FANUC

GE Fanuc Automation

Computer Numerical Control Products

Series 0i-Model C
Series 0i-Mate Model C

Connection Manual (Function)

GFZ-64113EN-1/01 August 2004

GFL-001

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

DEFINITION OF WARNING, CAUTION, AND NOTE

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

WARNING

Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE

The Note is used to indicate supplementary information other than Warning and Caution.

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

PREFACE

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

1. General
Describes feature of the function. Refer to Operator's manual as requied.

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

3. Parameters
Describes parameters related with a function.

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

5. Reference item
List the related items of the related manuals in a table.

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

Applicable models

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

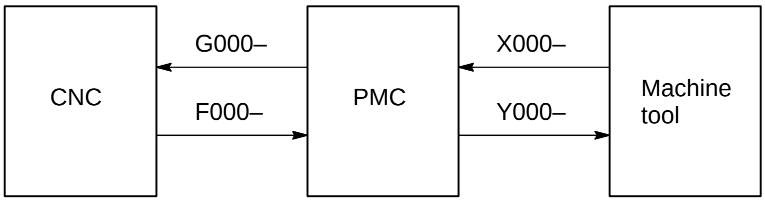
<table>
<tr><th>Model name</th><th colspan="3">Abbreviation</th></tr>
<tr><td>FANUC Series 0i–TC</td><td>0i–TC</td><td rowspan="2">Series 0i–C</td><td rowspan="2">0i</td></tr>
<tr><td>FANUC Series 0i–MC</td><td>0i–MC</td></tr>
<tr><td>FANUC Series 0i Mate–TC</td><td>0i Mate–TC</td><td rowspan="2">Series 0i Mate–C</td><td rowspan="2">0i Mate</td></tr>
<tr><td>FANUC Series 0i Mate–MC</td><td>0i Mate–MC</td></tr>
</table>

For ease of understanding, the models are categorized as follows:
T series: 0*i*–TC, 0*i* Mate–TC
M series: 0*i*–TC, 0*i* Mate–TC

NOTE
Some functions described in this manual may not be applied to some products.
For details, refer to the DESCRIPTIONS manual (B–64112EN).

Signal description

Relation of interface signals among the CNC, the PMC and the machine tool is shown below:



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

- **Expression of signals**

One address accommodates eight signals.

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

	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP	SA	STL	SPL				RWD

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

[Example 1]

Signal EXLM, ST is a common signal, STLK is for T series only and RLSOT and RVS are for M series only.

	#7	#6	...	#2	#1	#0	
G007	RLSOT	EXLM		ST	STLK	RVS	T series M series

Parameter description

Parameters are classified by data type as follows :

Dta type	Valid data range	Remarks
Bit	0 or 1	
Bit axis		
Byte	–127 to 127 0 to 255	In some parameters, signs are ignored.
Byte axis		
Word	–32767 to 32767 0 to 65535	
Word axis		
2–word	–99999999 to 99999999	
2–word axis		

NOTE

1 For the bit type and bit axis type parameters, a single data number is assigned to 8 bits. Each bit has a different meaning.

2 The axis type allows data to be set separately for each control axis.

3 The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

- **Notation of bit type and bit axis type parameters**

Data No. | Data (#0 to #7 indicates bit position)

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

- **Notation of parameters other than bit type and bit axis type**

Data No.	Data
1023	Servo axis number of a specific axis

> **NOTE**
> In an item where both T series and M series are described, parameters having different meanings between the T series and M series and parameters that are valid only for the T or M series are indicated in two levels as shown below. Parameters left blank are unavaliable.

[Example 1]

Parameter 5010 has different meanigs for the T series and M series.

5010	Tool nose radius compensation . . .	T series
	Cutter compensation C . . .	M series

[Example 2]

DPI is a parameter common to the M and T series, but GSB and GSC are parameters valid only for the T series.

	#7	#6			#0	
3401	GSC	GSB			DPI	T series
					DPI	M series

[Example 3]

The following parameter is provided only for the M series.

1450		T series
	F1 digit feed . . .	M series

Related manuals of Series 0*i*–C/0*i* Mate–C

The following table lists the manuals related to Series 0*i*–C and 0*i* Mate–C. This manual is indicated by an asterisk(*).

Manual name	Specification number	
DESCRIPTIONS	B–64112EN	
CONNECTION MANUAL (HARDWARE)	B–64113EN	
CONNECTION MANUAL (FUNCTION)	B–64113EN–1	*
Series 0*i*–TC OPERATOR'S MANUAL	B–64114EN	
Series 0*i*–MC OPERATOR'S MANUAL	B–64124EN	
Series 0*i* Mate–TC OPERATOR'S MANUAL	B–64134EN	
Series 0*i* Mate–MC OPERATOR'S MANUAL	B–64144EN	
MAINTENANCE MANUAL	B–64115EN	
PARAMETER MANUAL	B–64120EN	
PROGRAMMING MANUAL		
Macro Compiler/Macro Executor PROGRAMMING MANUAL	B–61803E–1	
FANUC MACRO COMPILER (For Personal Computer) PROGRAMMING MANUAL	B–66102E	
PMC		
PMC Ladder Language PROGRAMMING MANUAL	B–61863E	
PMC C Language PROGRAMMING MANUAL	B–61863E–1	
Network		
PROFIBUS–DP Board OPERATOR'S MANUAL	B–62924EN	
Ethernet Board/DATA SERVER Board OPERATOR'S MANUAL	B–63354EN	
FAST Ethernet Board/FAST DATA SERVER OPERATOR'S MANUAL	B–63644EN	
DeviceNet Board OPERATOR'S MANUAL	B–63404EN	
OPEN CNC		
FANUC OPEN CNC OPERATOR'S MANUAL Basic Operation Package 1 (For Windows 95/NT)	B–62994EN	
FANUC OPEN CNC OPERATOR'S MANUAL (DNC Operation Management Package)	B–63214EN	

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

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

Manual name	Specification number
FANUC AC SERVO MOTOR α*i*s/α*i* series DESCRIPTIONS	B–65262EN
FANUC AC SERVO MOTOR β*i*s series DESCRIPTIONS	B–65302EN
FANUC AC SERVO MOTOR α*i*s/α*i*/β*i*s series PARAMETER MANUAL	B–65270EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B–65272EN
FANUC AC SPINDLE MOTOR β*i*s series DESCRIPTIONS	B–65312EN
FANUC AC SPINDLE MOTOR α*i*/β*i* series PARAMETER MANUAL	B–65270EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B–65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B–65322EN
FANUC AC SERVO MOTOR α*i*s/α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B–65285EN
FANUC AC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B–65325EN

Either of the following servo motors and the corresponding spindle can be connected to the CNC covered in this manual.

- FANUC SERVO MOTOR α*i*s series
- FANUC SERVO MOTOR β*i*s series

This manual mainly assumes that the FANUC SERVO MOTOR α*i*s series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

Table of Contents

Volume 1

Volume 2

APPENDIX

1 AXIS CONTROL

1.1 CONTROLLED AXES

General

Series 0*i*–C

Item	M series	T series
Controlled path	1 path	1 path
Controlled axes	Max. 4 axes	Max. 4 axes
Simultaneously controlled axes	Max. 4 axes	Max. 4 axes
Axis control by PMC	Max. simultaneous 4 axes (Not available on Cs axis)	Max. simultaneous 4 axes (Not available on Cs axis)
Cs contouring control	Max. 1 axis	Max. 1 axis

Series 0*i* Mate–C

Item	M series	T series
Controlled path	1 path	1 path
Controlled axes	3 axes	Max. 3 axes
Simultaneously controlled axes	Max. 3 axes	Max. 3 axes
Axis control by PMC	–	–
Cs contouring control	–	Max. 1 axis

Parameter

1010	Number of CNC–controlled axes

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Byte

[Valid data range] 1, 2, 3, ..., the number of controlled axes

Set the maximum number of axes that can be controlled by the CNC.

[Example] Suppose that the first axis is the X axis, and the second and subsequent axes are the Y, Z, and A axes in that order, and that they are controlled as follows:

X, Y, and Z axes: Controlled by the CNC and PMC
A axis: Controlled by the PMC (cannot be controlled directly by the CNC)

Then set this parameter to 3 (total 3: X, Y, and Z)

Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED (M series)	The number of the commanded axes exceeded that of simultaneously controlled axes. Correct the program.
	TOO MANY AXES COMMANDED (T series)	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where the skip function activated by the torque–limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.2.1	Controlled Axes
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.2.1	Controlled Axes
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.2.1	Controlled Axes
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.2.1	Controlled Axes

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

General

Each axis that is controlled by the CNC (including those controlled by the PMC) must be named. Select and set names from among X, Y, Z, A, B, C, U, V, and W (with parameter 1020).
The names of the basic axes, however, are fixed (X, Y, and Z for the M series and X and Z for the T series). The names of additional axes can be selected, as desired, from the names other than those for the basic axes. The same name cannot be assigned to more than one axis.

Parameter

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

Axis name	Setting	Axis name	Setting	Axis name	Setting	Axis name	Setting
X	88	U	85	A	65	E	69
Y	89	V	86	B	66		
Z	90	W	87	C	67		

NOTE

1. With the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
2. The same axis name cannot be assigned to more then one axis.
3. The address used by the secondary auxilliary function (address B with the T series or, with the M series, the address specified in parameter No.3460) cannot be used as an axis name.
4. With the T series, when address C or A is used for chamfering, corner R, or direct drawing dimension programming (when the CCR parameter (bit 4 of parameter No.3405) is set to 1), addresses C or A cannot be used as an axis name.
5. Only with the T series, address E can be used as an axis name. Address E cannot be used with the M series. When address E is used as an axis name, note the following:
   – When G code system A is used, address E is always assigned to an absolute command.
   – When an equal–lead threading command (G32) is issued in the FS10/11 tape format. Use address F to specify the thread lead.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.2.2	NAMES OF AXES
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.2.2	NAMES OF AXES
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.2.2	NAMES OF AXES
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.2.2	NAMES OF AXES

1.2.2 Increment System

General

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or degrees.
The increment system is classified as either IS–B or IS–C (Tables 1.2.2(a) and 1.2.2 (b)). Select IS–B or IS–C using bit 1 (ISC) of parameter 1004.

Table 1.2.2 (a) Increment system IS–B

		Least input increment	Least command increment
Metric system machine	mm input	0.001mm(Diameter)	0.0005mm
		0.001mm(Radius)	0.001mm
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.0005mm
		0.0001inch(Radius)	0.001mm
		0.001deg	0.001deg
Inch system machine	mm input	0.001mm(Diameter)	0.00005inch
		0.001mm(Radius)	0.0001inch
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.00005inch
		0.0001inch(Radius)	0.0001inch
		0.001deg	0.001deg

Table 1.2.2 (b) Increment system IS–C

		Least input increment	Least command increment
Metric system machine	mm input	0.0001mm(Diameter)	0.00005mm
		0.0001mm(Radius)	0.0001mm
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.00005mm
		0.00001inch(Radius)	0.0001mm
		0.0001deg	0.0001deg
Inch system machine	mm input	0.0001mm(Diameter)	0.000005inch
		0.0001mm(Radius)	0.00001inch
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.000005inch
		0.00001inch(Radius)	0.00001inch
		0.0001deg	0.0001deg

> **NOTE**
> Diameter programming is used only for T series. Diameter programming or radius programming is determined by parameter DIAx (No. 1006#3) for each axis. Also, parameter IPR (No. 1004#7) can make the least input increment of IS–B and IS–C ten times the least command increment on each axis.

Parameter

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

The following parameter can be set at "Setting screen".

[Data type] Bit

INI Unit of input

0 : In mm
1 : In inches

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

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

[Data type] Bit

INM Least command increment on the linear axis

0 : In mm (metric system machine)
1 : In inches (inch system machine)

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

> **NOTE**
> After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit

ISA, ISC The least input increment and least command increment are set.

ISC	ISA	Least input increment and least command increment	Symbol
0	0	0.001mm, 0.001deg, or 0.0001inch	IS–B
0	1	0.01mm, 0.01deg, or 0.001inch	IS–A
1	0	0.0001mm, 0.0001deg, or 0.00001inch	IS–C

NOTE
IS–A is not available.

IPR Whether the least input increment for each axis is set to a value 10 times as large as the least command increment is specified, in increment systems of IS–B and IS–C, mm input.

0 : The least input increment is not set to a value 10 times as large as the least command increment.

1 : The least input increment is set to a value 10 times as large as the least command increment.

If IPR is set to 1, the least input increment is set as follows:

Input increment	Least input increment
IS–B	0.01 mm, 0.01 deg, or 0.0001 inch
IS–C	0.001 mm, 0.001 deg, or 0.00001 inch

NOTE
For IS–A, the least input increment cannot be set to a value 10 times as large as the least command increment.
When inch of input is specified, the least input increment does not become 10 times as large as the least command increment.

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

NOTE
When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

DIAx Either a diameter or radius is set to be used for specifying the amount of travel on each axis.

0 : Radius

1 : Diameter

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.2.3	Increment System
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.2.3	Increment System
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.2.3	Increment System
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.2.3	Increment System

1.2.3 Specifying the Rotation Axis

General

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

Parameter

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

> **NOTE**
> After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis.

ROSx	ROTx	Meaning
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.

ROSx	ROTx	Meaning
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) The rotation axis roll over function and index table indexing function (M series) cannot be used.

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

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit axis

ROAx The roll–over function of a rotation axis is
0 : Invalid
1 : Valid

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

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

NOTE
RABx is valid only when ROAx is 1.

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

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

1260	Amount of a shift per one rotation of a rotation axis

NOTE
1 After setting the parameter, turn off the power once and turn it on again to operate the machine.
2 This parameter is valid only when ROAx = 1.

[Data type] Two–word axis

[Unit of data]

Increment system	Unit of data	Standard value	Unit
IS–A	0.01	36000	deg
IS–B	0.001	360000	
IS–C	0.0001	3600000	

[Valid data range] 1000 to 9999999

Set the amount of a shift per one rotation of a rotation axis.

Note

NOTE
Rotary axis roll–over function cannot be used together with the indexing function of the index table.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.20.2	Rotary Axis Roll–over
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.19.2	Rotary Axis Roll–over
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.18.1	Rotary Axis Roll–over

1.2.4 Controlled Axes Detach

General

These signals release the specified control axes from control by the CNC. When attachments are used (such as a detachable rotary table), these signals are selected according to whether the attachments are mounted. The signals can also be used for switching the C axis and spindle on lathes.

When multiple rotary tables are used in turn, the tables must use motors of the same model. Absolute pulse coders cannot be used.

Signal

Controlled axis detach signals DTCH1 – DTCH4 <G124#0–#3>

[Classification] Input signal

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

DTCH 1

1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
: :
: :

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

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

Controlled axis detach status signals MDTCH1 – MDTCH4 <F110#0–#3>

[Classification] Output signal

[Function] These signals notify the PMC that the corresponding axes have been released from control.
These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.

MDTCH 1
1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
: :
: :

[Output condition] These signals are 1 in the following case:
– When the corresponding axes are released from control
These signals are 0 in the following case:
– When the corresponding axes are under control

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G124					DTCH4	DTCH3	DTCH2	DTCH1

	#7	#6	#5	#4	#3	#2	#1	#0
F110					MDTCH4	MDTCH3	MDTCH2	MDTCH1

Parameter

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

Setting entry is acceptable.

[Data type] Bit axis

RMVx Releasing the assignment of the control axis for each axis
0 : Not released
1 : Released

> **NOTE**
> RMVx is valid when the bit 7 (RMBx) in parameter 1005 is 1.

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

[Data type] Bit axis

RMBx Releasing the assignment of the control axis for each axis (signal input and setting input)
0 : Invalid
1 : Valid

MCCx When an axis is released from control, control for the MCC signal for the corresponding servo amplifier is

0 : Disabled
1 : Enabled

> **NOTE**
> If the servo motor for an axis is connected to a 2–axis or other multiaxis amplifier, releasing the axis from control causes servo alarm 401 (V ready off) to be output. This alarm can be disabled by this parameter. When the servo motor is disconnected from the CNC, however, servo alarm 401 is output, regardless of the value of the parameter, due to the nature of multiaxis amplifier.

Caution

> **CAUTION**
> When a 2–axis or 3–axis amplifier is used, releasing only one axis from control results in the output of servo alarm 401 (V ready off). Use 1–axis amplifiers for those axes to be released from control, e.g., by replacing the rotary table.

Note

> **NOTE**
> 1 Controlled axis detach signals DTCH1 <G124#0>, DTCH2 <G124#1>, DTCH3 <G124#2>, … can be changed from 1 to 0 or from 0 to 1 when the power is first turned on or when no movement is being executed along the corresponding axis. If these signals are changed from 0 to 1 when the tool is moving along the corresponding axis, the axis is released from control upon completion of the movement.
> 2 For these signals to be attached, parameter No. 1005#7 must be set, indicating the axes are detachable.
> 3 Setting parameter No. 0012#7 from the MDI panel detaches the axes in the same way as these signals.
> 4 Those axes that are released from control lose their reference positions. Reference position return must, therefore, be performed for the axes prior to executing move commands for the axes. Specifying a move command before reference position return has been performed causes alarm 224 to be output (the alarm can be disabled by setting bit 0 (ZRNx) of parameter 1005).

1.2.5 Outputting the Movement State of an Axis

General

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

Signal

Axis moving signals MV1 – MV4 <F102#0–#3>

[Classification] Output signal

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

MV 1

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

[Output condition] The signals turn to “1” in the following cases:

. The corresponding axis has started moving.

. In manual handle feed mode, the handle feed axis of the corresponding axis has been selected.

The signals turn to “0” in the following case:

. When the move command for the corresponding axis has been distributed (when bit 6 (MVX) of parameter 3003 is 0)

. When deceleration for the corresponding axis has been completed and the axis is set to the in–position condition. If in–position check is not performed, when the deceleration for the corresponding axis is completed. (When bit 6 (MVX) of parameter 3003 is 1)

Setting 1 in bit 7 (MVG) of parameter 3003 prevents these signals from being output during drawing in dynamic graphics mode (drawing without movement of the machine) in the T series.
The axis moving signals are output in the M series.

Axis moving direction signals MVD1 – MVD4 <F106#0–#3>

[Classification] Output signal

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

MVD 1

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

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

> **CAUTION**
> These signals maintain their condition during a stop, indicating the direction of the axes' movement before stopping.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F102					MV4	MV3	MV2	MV1
F106					MVD4	MVD3	MVD2	MVD1

Parameter

- **Setting the output format of the axis moving signal**

	#7	#6	#5	#4	#3	#2	#1	#0
3003	MVG	MVX						
		MVX						

[Data type] Bit

MVX The axis moving signal is set to 0 when:

0 : Distribution for the axis is completed. (The signal is set to 0 in deceleration.)

1 : Deceleration of the axis is terminated, and the current position is in the in–position.
When the deceleration–time in–position check is suppressed by setting bit 5 (NCI) of parameter No. 1601, the signal is set to 0 at the end of deceleration.

MVG While drawing using the dynamic graphics function (with no machine movement), the axis moving signal is:

0: Output
1: Not output

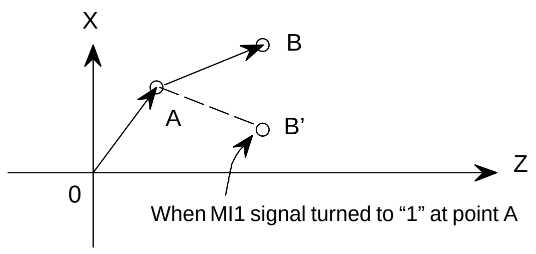
Caution

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

1.2.6 Mirror Image

General

Mirror image can be applied to each axis, either by signals or by parameters (setting input is acceptable). All movement directions are reversed during automatic operation along axes to which a mirror image is applied.



Mirror image (Example for T series)

However, the following directions are not reversed:

– Direction of manual operation and direction of movement, from the intermediate position to the reference position during automatic reference position return (for the M and T series)
– Approach direction for single direction positioning (G60) and shift direction for boring cycles (G76 and G87) (for M series only)

Mirror image check signals indicate whether mirror image is applied to each axis. System variable #3007 contains the same information (refer to the operator’s manual).

Signal

Mirror image signal MI1 – MI4 <G106#0–#3>

[Classification] Input signal

[Function] Apply mirror image to the specified axes.

[Operation] Apply mirror image to those axes for which the signals are 1.
These signals are provided for the controlled axes on a one–to–one basis.
A number appended to a signal represents the controlled axis number.

MI 1

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

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

a) During offset cancel;

b) When the CNC is in the automatic operation stop state and not in the feed hold state.

Mirror image check signal MMI1 – MMI4 <F108#0–#3>

[Classification] Output signal

[Function] These signals indicate the mirror image condition of each axis. The mirror image is set by taking the logical sum of the signal from the MDI panel and the input signal of the machine tool, then relaying the information to the machine tool.
These signals are provided for every control axis; the numeral in the signal name indicates the relevant control axis number.

MMI 1

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

[Output condition] These signals turn to “1” when:

· Mirror image signal MIn of the corresponding axis is “1”; or

· Mirror image of the corresponding axis is turned on by setting data from the MDI panel.

These signals turn to “0” when:

· Mirror image signal (MIn) of the corresponding axis is “0” and the setting of the mirror image in the control unit is turned off.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G106					MI4	MI3	MI2	MI1

	#7	#6	#5	#4	#3	#2	#1	#0
F108					MMI4	MMI3	MMI2	MMI1

Parameter

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

The following parameter can be set at “Setting screen.”

[Data type] Bit axis

MIRx Mirror image for each axis

0 : Mirror image is off.
1 : Mirror image is on.

Warning

WARNING

1 When programmable mirror image (M series) and ordinary mirror image are specified at the same time, programmable mirror image is applied first.

2 No programmable mirror image (M series) affects mirror image check signals MMI1 to MMI4 <F108>.

Caution

CAUTION

Even when the mirror image is applied, commands which do not actuate mirror image (such as automatic reference position return and manual operation) do not affect mirror image check signals MMI1 to MMI4 <F108>.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	III.4.8	Mirror Image
	OPERATOR’S MANUAL (T series) (B–64114EN)	III.4.8	Mirror Image
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	III.4.8	Mirror Image
	OPERATOR’S MANUAL (T series) (B–64134EN)	III.4.8	Mirror Image

1.2.7 Follow–up

General

When position control is disabled for the controlled axes (when the servo is off, during emergency stop, or during a servo alarm), if the machine is moved, a positional error occurs. Follow–up is a function for changing the current position of the CNC and reseting the error counter to zero. Assuming a command corresponding to the error has been specified.
You can select whether to perform follow–up for axes when the servo is turned off.
Follow–up is always performed during emergency stop or a servo alarm.

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

When signal *FLWU is 1 or bit 0 (FUPx) of parameter 1819 is 1, follow–up is not performed. The error is added to the error counter as a servo error. In this case, the machine moves to compensate for the error when the servo off signal changes to 0.
In general, follow–up is not used if the machine is mechanically clamped when position control is disabled for the controlled axes.

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

When *FLWU is "0", the follow-up function is engaged. The present position of the CNC is changed to reset the error counter to zero. The machine tool remains in a deviated position, but since the present position of the CNC changes correspondingly, the machine moves to the correct position when the absolute command is next applied.
In general, follow–up should be used when motors are driven by mechanical handles.

Signal

Follow–up signal *FLWU <G007#5>

[Classification] Input signal

[Function] Select whether to perform follow–up when the servo is turned off for those axes for which bit 0 (FUPx) of parameter 1819 is 0.

[Operation] 0: Performs follow–up.
1: Does not perform follow–up.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007			*FLWU					

Parameter

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

[Data type] Bit axis

FUPx To perform follow–up when the servo is off for each axis.

0 : The follow–up signal, *FLWU, determines whether follow–up is performed or not.
When *FLWU is 0, follow–up is performed.
When *FLWU is 1, follow–up is not performed.

1 : Follow–up is not performed.

CAUTION
When the index table indexing function (M series) is used, be sure to set FUPx of the 4th axis to 1.

Reference item

CONNECTION MANUAL (This manual)	1.2.8	Servo Off (Mechanical handle)

1.2.8 Servo Off (Mechanical Handle)

General

Place the controlled axes in the servo off state, stop the current to the servo motor, which disables position control. However, the position detection feature functions continuously, so the current position is not lost.
These signals are used to prevent the servo motors from overloading when the tools on the axes are mechanically clamped under certain machining conditions on the machine, or to move the machine by driving the motors by mechanical handles.

Signal

Servo off signal SVF1 – SVF4 <G126#0–#3>

[Classification] Input signal

[Function] Select whether to place each axis in the servo off state.

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

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

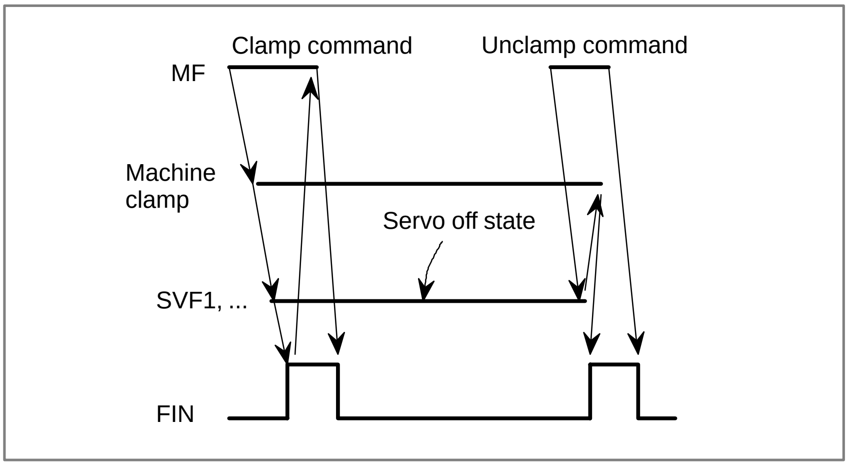
[Operation] These signals put the axes for which the signals are 1 in the servo off state (the current to the servo motor is stopped). This disables position control. However, the position detection feature continues to function, so the current position is not lost.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G126					SVF4	SVF3	SVF2	SVF1

Caution

CAUTION
1 In general, interlock is applied to an axis while the servo off signal for that axis is 1.
2 When one of these signals turns to “1”, the servo motor is turned off. The mechanical clamp is done by using the auxiliary function. Set the timing for the auxiliary function, mechanical clamp and servo off signals as shown in the diagram below. The clamp command auxiliary function should be executed only after the distribution end signal (DEN) turned to “1”.



Reference item

CONNECTION MANUAL (This manual)	1.2.7	Follow–up

1.2.9 Position Switch

General

Position switch signals can be output to the PMC while the machine coordinates along a controlled axes are within a specified ranges.

Signal

Position switch signal PSW01 – PSW16 <F070#0 – F071#7>

[Classification] Output signal

[Function] Indicates that the machine coordinates along the controlled axes specified by parameters (6910 to 6925) are within the ranges specified by parameters (6930 to 6945 and 6950 to 6965). Up to 16 position switch signals can be output.
(Using 11 or more position switches requires setting the EPS parameter (bit 1 of No. 6901.)

[Output condition] These signals are 1 in the following case:

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

These signals are 0 in the following case:

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

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071	PSW16	PSW15	PSW14	PSW13	PSW12	PSW11	PSW10	PSW09

Parameter

- **Increasing the number of position switch signals**

	#7	#6	#5	#4	#3	#2	#1	#0
6901					PSF	PCM	EPS	IGP

[Data type] Bit

IGP During follow–up for the absolute position detector, position switch signals are:
0 : Output
1 : Not output

EPS The number of position switches is:
0 : Up to 10.
1 : Up to 16.

PCM Position switch signals are output:
0 : Without considering acceleration/deceleration and servo delay.
1 : With considering acceleration/deceleration and servo delay.

PSF In AI contour control, AI advanced preview control or advanced preview control mode, position switches are:
0 : Not used.
1 : Used.

To use the position switches in any of the following modes, set this parameter:
AI contour control, AI advanced preview control mode or advanced preview control mode

NOTE
The position switch signals are output considering acceleration/deceleration after interpolation and servo delay. Acceleration/deceleration after interpolation and servo delay are considered even for position switch signal output in a mode other than the AI contour control, AI advanced preview control, and advanced preview control modes. When this parameter is set to 1, however, signals are output from the position switches at different times from the specified ones.

- **Setting the correspondence between the position switch signals and the controlled axes**

6910	Axis corresponding to the first position switch
6911	Axis corresponding to the second position switch
6912	Axis corresponding to the third position switch
6913	Axis corresponding to the fourth position switch
6914	Axis corresponding to the fifth position switch
6915	Axis corresponding to the sixth position switch
6916	Axis corresponding to the seventh position switch
6917	Axis corresponding to the eighth position switch
6918	Axis corresponding to the ninth position switch
6919	Axis corresponding to the tenth position switch
6920	Axis corresponding to the eleventh position switch
6921	Axis corresponding to the twelveth position switch
6922	Axis corresponding to the thirteenth position switch
6923	Axis corresponding to the fourteenth position switch
6924	Axis corresponding to the fifteenth position switch
6925	Axis corresponding to the sixteenth position switch

[Data type] Byte

[Valid data range] 0 to Number of controlled axes

These parameters sequentially specify the numbers of the controlled axes corresponding to the 1st through 16th position switch functions. The corresponding position switch signal is output to the PMC when the machine coordinate of the corresponding axis is within the range set in parameters.

NOTE
1 Set 0 for the number corresponding to a position switch which is not to be used.
2 Parameter Nos. 6920 to 6925 are valid only when bit 1 (EPS) of parameter No. 6901 is 1.

- **Setting the machine coordinate ranges for which the position switch signals are output**

- **Maximum operation range**

6930	Maximum operation range of the first position switch
6931	Maximum operation range of the second position switch
6932	Maximum operation range of the third position switch
6933	Maximum operation range of the fourth position switch
6934	Maximum operation range of the fifth position switch
6935	Maximum operation range of the sixth position switch
6936	Maximum operation range of the seventh position switch
6937	Maximum operation range of the eighth position switch
6938	Maximum operation range of the ninth position switch
6939	Maximum operation range of the tenth position switch
6940	Maximum operation range of the eleventh position switch
6941	Maximum operation range of the twelveth position switch
6942	Maximum operation range of the thirteenth position switch
6943	Maximum operation range of the fourteenth position switch
6944	Maximum operation range of the fifteenth position switch
6945	Maximum operation range of the sixteenth position switch

[Data type] 2–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

These parameters sequentially set the maximum operation ranges of the 1st through 16th position switches.

> **NOTE**
> Parameter Nos. 6940 to 6945 are valid only when bit 1 (EPS) of parameter No. 6901 is 1.

● Minimum operation range

6950	Minimum operation range of the first position switch
6951	Minimum operation range of the second position switch
6952	Minimum operation range of the third position switch
6953	Minimum operation range of the fourth position switch
6954	Minimum operation range of the fifth position switch
6955	Minimum operation range of the sixth position switch
6956	Minimum operation range of the seventh position switch
6957	Minimum operation range of the eighth position switch
6958	Minimum operation range of the ninth position switch
6959	Minimum operation range of the tenth position switch
6960	Minimum operation range of the eleventh position switch
6961	Minimum operation range of the twelveth position switch
6962	Minimum operation range of the thirteenth position switch
6963	Minimum operation range of the fourteenth position switch
6964	Minimum operation range of the fifteenth position switch
6965	Minimum operation range of the sixteenth position switch

[Data type] 2–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

These parameters sequentially set the minimum operation ranges of the 1st through 16th position switches.

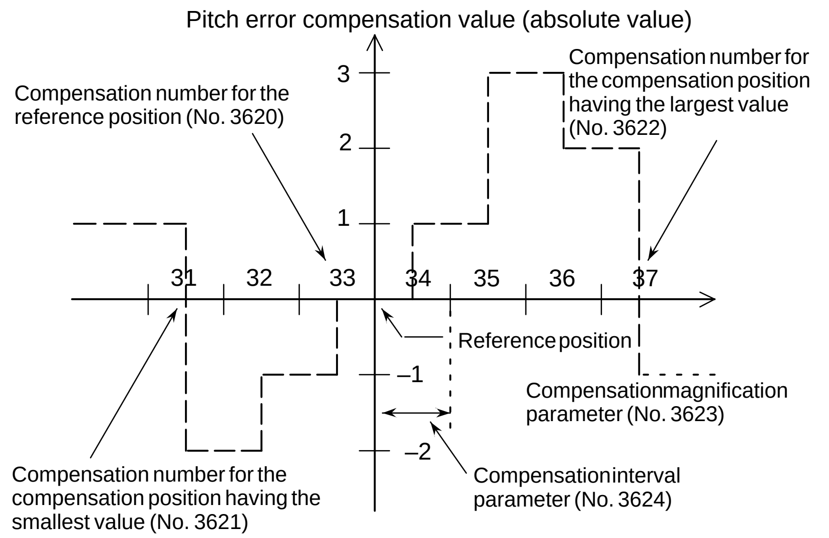
NOTE
Parameter Nos. 6960 to 6965 are valid only when bit 1 (EPS) of parameter No. 6901 is 1.

1.3 ERROR COMPENSATION

1.3.1 Stored Pitch Error Compensation

General

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



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

· Pitch error compensation position at the reference position (for each axis): Parameter 3620

· Pitch error compensation position having the smallest value (for each axis): Parameter 3621

· Pitch error compensation position having the largest value (for each axis): Parameter 3622

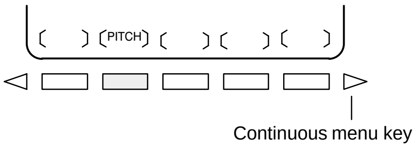
· Pitch error compensation magnification (for each axis): Parameter 3623

· Interval of the pitch error compensation positions (for each axis): Parameter 3624

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:

- Pitch error compensation position at the reference position (for each axis): Parameter 3620
- Pitch error compensation position having the smallest value (for each axis): Parameter 3621
- Pitch error compensation position having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation positions (for each axis): Parameter 3624



2 Press function key SYSTEM.

3 Press the continuous menu key ▷, then press chapter selection soft key **[PITCH]**.
The following screen is displayed:

```
PIT-ERROR SETTING                        O0000 N00000

     NO.      DATA      NO.      DATA      NO.      DATA
     0000     0         0010     0         0020     0
     0001     0         0011     0         0021     0
     0002     0         0012     0         0022     0
     0003     0         0013     0         0023     0
  (X)0004     0         0014     0         0024     0
     0005     0         0015     0         0025     0
     0006     0         0016     0         0026     0
     0007     0         0017     0         0027     0
     0008     0         0018     0         0028     0
     0009     0         0019     0         0029     0
 > _
 MEM **** *** ***       16:05:59
 [ NO.SRH ][ ON:1 ][ OFF:0 ][ +INPUT ][ -INPUT ]
```

4 Move the cursor to the compensation position number to be set in either of the following ways:

- Enter the compensation position number and press the [NO.SRH] soft key.
- Move the cursor to the compensation position number using the page keys, PAGE ↑ and PAGE ↓, and cursor keys, ↑, ↓, ←, and →.

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

Explanations

- **Specifying the compensation position**

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

- **Compensation position number**

1024 compensation positions from No. 0 to 1023 are available on the pitch error setting screen. Assign arbitrary positions for each axis using parameters.
The number of the compensation position at the reference position (parameter 3620), number of the compensation position having the smallest value (parameter 3621), and number of the compensation position having the largest value (parameter 3622) must be set for each axis.
The name of each axis is displayed before the smallest compensation position number on the pitch error setting screen.

- **Interval of compensation positions**

The pitch error compensation positions are equally spaced to parameter No. 3624. Set the space between two adjacent positions for each axis.

The minimum interval between pitch error compensation positions is limited and obtained from the following equation:

Minimum interval of pitch error compensation positions = maximum feedrate (rapid traverse rate)/3750

Unit:

· Minimum interval of pitch error compensation positions: mm, inches, deg.

· Maximum feed rate: mm/min, inch/min, deg/min

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

Examples

- **For linear axis**

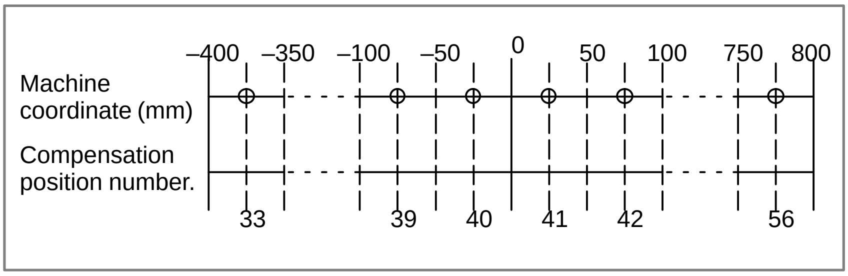
· Machine stroke: –400 mm to +800 mm
· Interval between the pitch error compensation positions: 50 mm
· No. of the compensation position of the reference position: 40
If the above is specified, the No. of the farthest compensation position in the negative direction is as follows:

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

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

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

The correspondence between the machine coordinate and the compensation position No. is as follows:



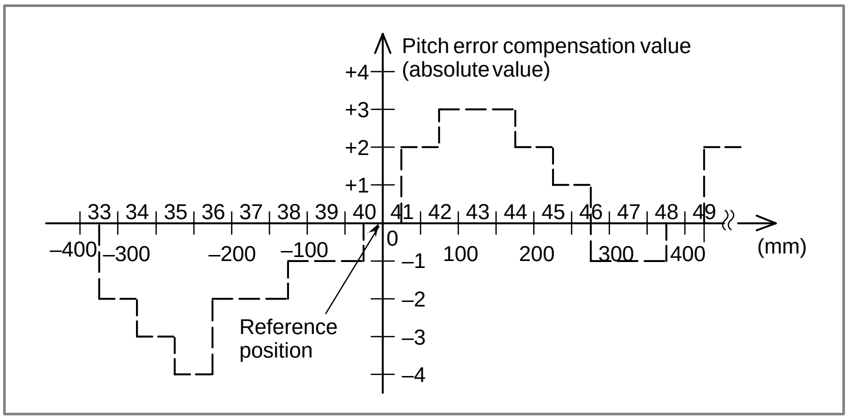
Compensation values are output at the positions indicated by ○ .

Therefore, set the parameters as follows:

Parameter	Setting value
3620 : Compensation number for the reference position	40
3621 : Smallest compensation position number	33
3622 : Largest compensation position number	56
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	50000

The compensation amount is output at the compensation position No. corresponding to each section between the coordinates.
The following is an example of the compensation amounts.

No	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49	56
Compensation value	–2	–1	–1	+2	0	+1	0	+1	+2	+1	0	–1	–1	–2	0	+1	+2	1

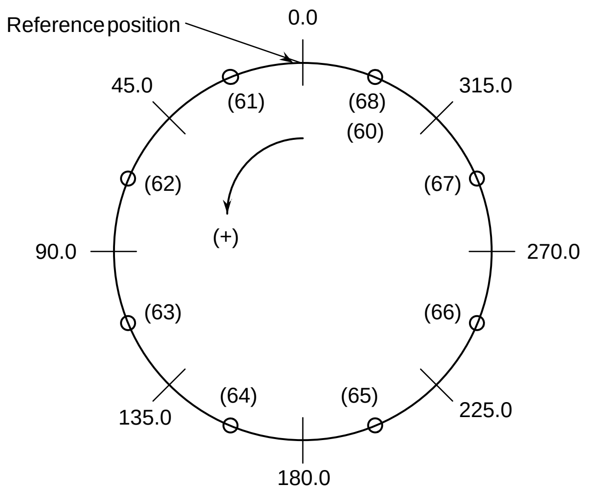


- **For rotary axis**

·Amount of movement per rotation: 360°
· Interval between pitch error compensation positions: 45°
· No. of the compensation position of the reference position: 60
If the above is specified, the No. of the farthest compensation position in the negative direction for the rotating axis is always equal to the compensation position No. of the reference position.
The No. of the farthest compensation position in the positive direction is as follows:

No. of the compensation position of the reference position + (Move amount per rotation/Interval between the compensation positions)
= 60 + 360/45= 68

The correspondence between the machine coordinate and the compensation position No. is as follows:



Compensation values are output at the positions indicated by ○.

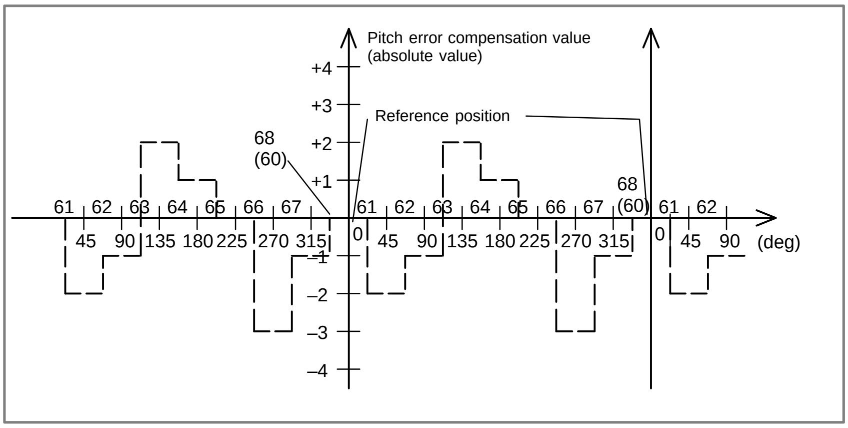
Therefore, set the parameters as follows:

Parameter	Setting value
3620 : Compensation number for the reference position	60
3621 : Smallest compensation position number	60
3622 : Largest compensation position number	68
3623 : Compensation magnification	1
3624 : Interval between pitch error compensation positions	45000
3625 : Movement value per rotation	360000

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

The following is an example of compensation amounts.

No	60	61	62	63	64	65	66	67	68
Compensation value	+1	–2	+1	+3	–1	–1	–3	+2	+1



Parameter

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

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

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

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

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

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

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

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

Set the number of the pitch error compensation position at the extreme positive position for each axis.

NOTE
This value must be larger than the value of parameter (No. 3620).

3623	Magnification for pitch error compensation for each axis

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Byte axis

[Unit of data] 1

[Valid data range] 0 to 100

Set the magnification for pitch error compensation for each axis.

If the magnification is set to 1, the magnification is the same as the detection unit.

3624	Interval between pitch error compensation positions for each axis

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 99999999

The pitch error compensation positions are equally spaced to parameter No. 3624. Set the space between two adjacent positions for each axis.

The minimum interval between pitch error compensation positions is limited and obtained from the following equation:

Minimum interval of pitch error compensation positions = maximum feedrate (rapid traverse rate)/7500

Unit:

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

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

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

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

[Data type] 2–word axis

[Valid data range] 0 to 99999999

If the pitch error compensation of rotation axis type is performed (bit 1 (ROSx) of parameter No. 1006 is set to 0 and bit 0 (ROTx) of parameter No. 1006 is set to 1), set the travel distance per revolution. The travel distance per revolution does not have to be 360 degrees, and a cycle of pitch error compensation of rotation axis type can be set.

However, the travel distance per revolution, compensation interval, and number of compensation points must satisfy the following condition:

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

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

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

Warning

WARNING

1 **Compensation value range**
Compensation values can be set within the range from –7 x compensation magnification (detection unit) to +7 x compensation magnification (detection unit). The compensation magnification can be set for each axis within the range from 0 to 100 in parameter 3623.

2 **Pitch error compensation of the rotary axis**
For the rotating axis, the interval between the pitch error compensation positions shall be set to one per integer of the amount of movement (normally 360°) per rotation. The sum of all pitch error compensation amounts per rotation must be made to 0. Also, set the same compensation value to a position and the same position with one rotation.

3 **Conditions where pitch error compensation is not performed**
Note that the pitch error is not compensated in the following cases:
- When the machine is not returned to the reference position after turning on the power. This excludes the case where an absolute position detector is employed.
- If the interval between the pitch error compensation positions is 0.
- If the compensation position Nos. on the positive or negative direction do not fall within the range of 0 to 1023.
- If the compensation position Nos. do not conform to the following relationship:
  Negative side ≦ Reference position < Positive side

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.8.6.3	Inputting pitch error compensation data
		III.8.6.4	Outputting pitch error compensation data
		III.11.5.2	Displaying and setting pitch error compensation data
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.8.6.3	Inputting pitch error compensation data
		III.8.6.4	Outputting pitch error compensation data
		III.11.5.2	Displaying and setting pitch error compensation data
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.8.6.3	Inputting pitch error compensation data
		III.8.6.4	Outputting pitch error compensation data
		III.11.5.2	Displaying and setting pitch error compensation data
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.8.6.3	Inputting pitch error compensation data
		III.8.6.4	Outputting pitch error compensation data
		III.11.5.2	Displaying and setting pitch error compensation data

1.3.2 Backlash Compensation

General

● **Backlash compensation**

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

● **Backlash compensation for each rapid traverse and cutting feed**

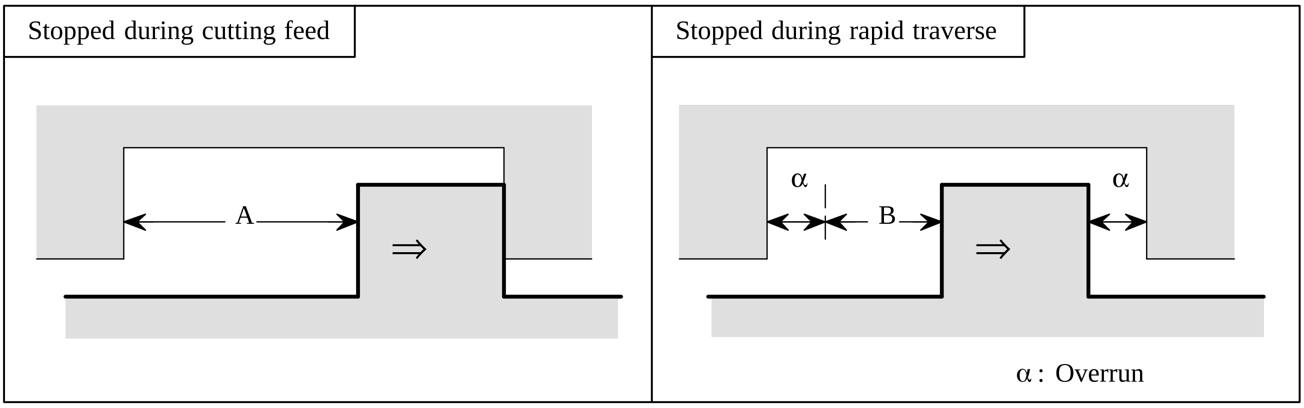
More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the rapid traverse or the cutting feed.

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

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

· α = (A–B)/2

· The positive or negative direction for compensating values is the direction of movement.



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

Parameter

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

[Data type] Bit

RBK Backlash compensation applied separately for cutting feed and rapid traverse

0 : Not performed
1 : Performed

1851	Backlash compensating value for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensating value for each axis.

When RBK is 1, set the backlash compensating value for cutting feed.

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

1852	Backlash compensating value used for rapid traverse for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensating value used in rapid traverse for each axis.

This parameter is valid when RBK is set to 1.

Caution

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

Note

NOTE
When backlash compensation is applied separately for cutting feed and rapid traverse, jog feed is regarded as cutting feed.

1.3.3 Bidirectional Pitch Error Compensation

Outline

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

Setting data

1. Setting parameters
   Set the following parameters for each axis.

Table 1.3.3 (a)

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

2. Pitch error compensation data
   The compensation point numbers can be from 0 to 1023 and from 3000 to 4023. This data may be used for both the positive and negative directions. Note, however, that the set of compensation data for a given axis cannot extend over 1023 and 3000.

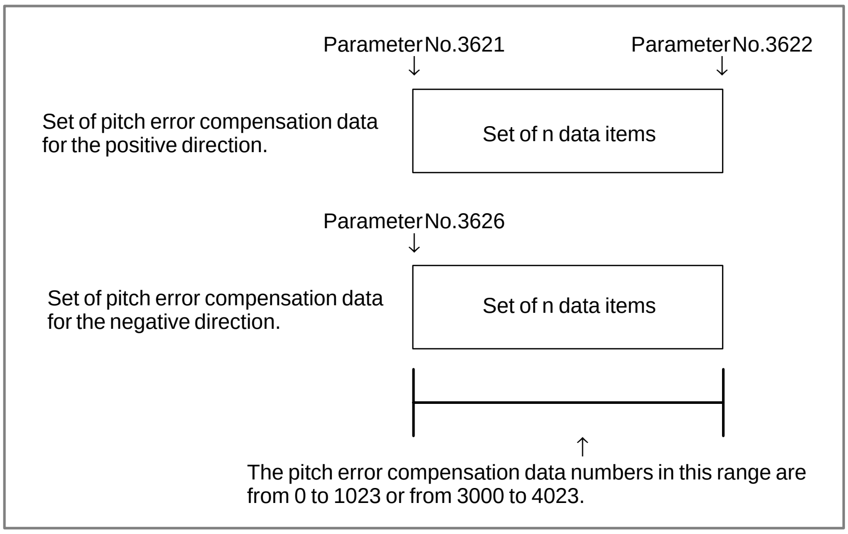


Fig. 1.3.3 (a)

Data setting example

If the direction of a manual reference position return is positive on an axis (linear axis) having the pitch error amounts shown in the figure below (Fig. 1.3.3 (b)), set the data given in the table below (Table 1.3.3 (b)).

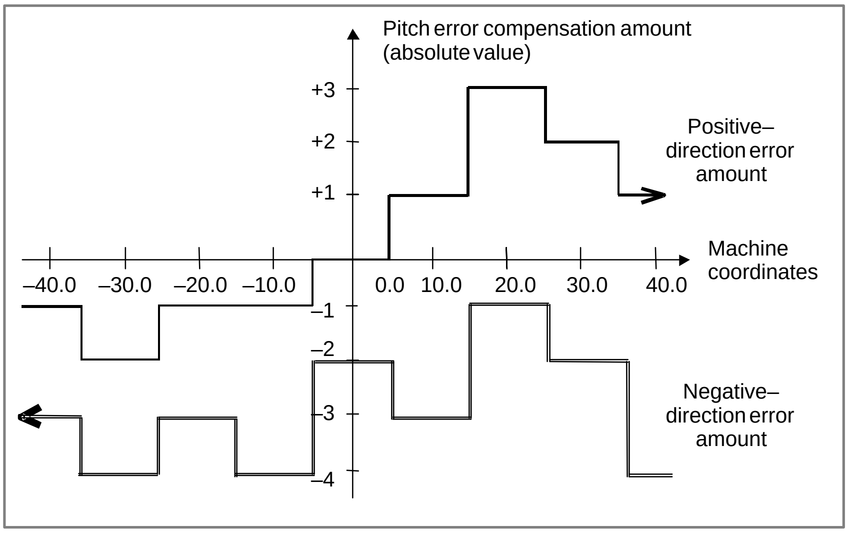


Fig. 1.3.3 (b)

Table 1.3.3 (b) Positive–direction pitch error data

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

As pitch error data, always set incremental values as viewed in the negative direction (direction toward the left in Fig. 1.3.3 (b)).

Table 1.3.3 (c) Negative–direction pitch error data

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

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

Table 1.3.3 (d)

Data number	Setting	Description
3605#0	1	Bidirectional pitch error compensation, 1: Enabled / 0: Disabled
3620	23	Number of the pitch error compensation point for the reference position
3621	20	Number of the most distant pitch error compensation point on the – side for travel in the positive direction
3622	27	Number of the most distant pitch error compensation point on the + side for travel in the positive direction
3623	1	Pitch error compensation magnification
3624	10000	Pitch error compensation point interval
3625	–	For a rotation axis, amount of travel per rotation in pitch error compensation
3626	30	Number of the most distant pitch error compensation point on the – side for travel in the negative direction
3627	–2	Pitch error compensation amount (absolute value) at the reference position when the machine moves to the reference position in the direction opposite to that of the reference position return

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

Compensation example

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

0.0 to 40.0,
40.0 to –40.0, and
–40.0 to 0.0

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

Machine coordinate	0.0	5.0	15.0	25.0	35.0	40.0
Compensa-tion pulse	–	–1	–2	+1	+1	+5

Machine coordinate	35.0	25.0	15.0	5.0	–5.0	–15.0	–25.0	–35.0	–40.0
Compensa-tion pulse	–2	–1	+2	–1	+2	–1	+1	–1	–2

Machine coordinate	–35.0	–25.0	–15.0	–5.0	0.0
Compensa-tion pulse	+1	–1	0	–1	–

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

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

+5 = – ((–4) – (+1))

Pitch error associated with the positive–direction absolute value at the position of 40.0

Pitch error associated with the negative–direction absolute value at the position of 40.0

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

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

–2 = – ((–1) – (–3))

Pitch error associated with the negative–direction absolute value at the position of –40.0

Pitch error associated with the positive–direction absolute value at the position of –40.0

Setting and displaying data

All the compensation data can be displayed and set on the conventional screen for the pitch error compensation data.
And those data can be input and output by the following methods.

* Input by MDI
* Input by G10
* Input and output by input/output device interface
* Input by PMC window (function code 18)
  (It is not possible to input and output by the method other than the above methods.)

Output format : The output format is as follows:
N20000 P.... ;
N21023 P.... ;
N23000 P.... ;
N24023 P.... ;
N : Pitch error compensation point No. + 20000
P : Pitch error compensation data

Parameter

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

[Data type] Bit axis

BDP Specifies whether to use bidirectional pitch error compensation.
0 : Do not use.
1 : Use.

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

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

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023, 3000 to 4023

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

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

3621	Number of the pitch error compensation position at extremely negative position for each axis (In case of positive–direction movement)

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023, 3000 to 4023

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

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

3622	Number of the pitch error compensation position at extremely positive position for each axis (In case of positive–direction movement)

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023, 3000 to 4023

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

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

3623	Magnification for pitch error compensation for each axis

[Data type] Byte axis

[Unit of data] 1

[Valid data range] 0 to 100

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

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

3624	Interval between pitch error compensation positions for each axis

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 99999999

The pitch error compensation positions are arranged with equal spacing. The space between two adjacent positions is set for each axis. The minimum interval between pitch error compensation positions is limited and obtained from the following equation:

Minimum interval between pitch error compensation positions =
Maximum feedrate (rapid traverse rate) / 7500

Units: Minimum interval between pitch error compensation positions: mm, inch, deg
Maximum feedrate : mm/min, inch/min, deg/min

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

3625	Angular displacement per rotation in rotation–axis pitch error compensation

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 99999999

For rotation–axis pitch error compensation (with bit 0 (ROT) of parameter No. 1006 set to 1 and bit 1 (ROS) of parameter No.1006 set to 0), set angular displacement per rotation for each axis. The angular displacement per rotation need not always be 360 degrees. The period in rotation–axis pitch error compensation can be set.

The angular displacement per rotation, compensation interval, and number of compensation points must satisfy the following equation:

Angular displacement per rotation =
compensation interval $\times$ number of compensation points

The sum of the compensation values per rotation must always be 0.
When this parameter is set to 0, it assumes that 360 degree is set.

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

3626	Number of the pitch error compensation position at extremely negative position for each axis (In case of negative–direction movement)

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023, 3000 to 4023

When using bidirectional pitch error compensation, set the number of the pitch error compensation position at the extremely negative position for each axis in the case of negative–direction movement.

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

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

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –32768 to 32767

By using an absolute value, set a pitch error compensation value at the reference position when a movement is made to the reference position in the negative direction if the reference position return direction (bit 5 (ZMI) of parameter No. 1006) is positive, or when a movement is made to the reference position in the positive direction if the reference position return direction is negative.

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

Note

(1) This function is enabled after a manual reference position return or an automatic reference position return with the same sequence as that of a manual reference position return is performed. When an absolute position detector is used, however, the function is enabled after the power is turned on.

(2) When the machine moves to the reference position in the reference position return direction, set the absolute value of the pitch error compensation pulse to 0.

(3) When this function and backlash compensation are used at the same time, the pulse resulting from backlash compensation is superimposed on the compensation pulse when the travel direction is reversed.

(4) When this function is used for a rotation axis, the sum of the pitch error compensation amounts per rotation about the rotation axis must be 0 for both the positive and negative directions.

(5) The function cannot be used with the distance coded linear scale function.

1.4 SETTINGS RELATED TO SERVO–CONTROLLED AXES

The servo interface of the Series 16 features the following:

- Digitally controlled AC servo motor
- Motor feedback with serial pulse coders
  (1) Absolute pulse coder with a resolution of 1,000,000 pulses/rev
  (2) Absolute pulse coder with a resolution of 65,536 pulses/rev
  (3) Incremental pulse coder with a resolution of 10,000 pulses/rev
- Scale feedback with A/B/Z signal interface

1.4.1 Parameters Related to Servo

General

Explanation of terms frequently used in CNC

Least command increment
The minimum unit of a command to be given from CNC to the machine tool

Detection unit
The minimum unit which can detect the machine tool position

Command multiplier (CMR)
A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

Detection multiplier (DMR)
A constant to enable the weight of CNC command pulses to meet the weight of pulses from the detector

CAUTION
The relations among the least command increment, detection unit, CMR, and DMR are as specified below.
Least command increment = CMR × detection unit

$$\text{Detection unit} = \frac{\text{Move amount per revolution of motor}}{\text{DMR} \times \text{number of pulses of detector per revolution}}$$

The flexible feed gear function in the digital servo defines constant DMR using two parameters (Nos. 2084 and 2085) n and m (DMR = n/m).

Parameter

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

[Data type] Bit

CVR When velocity control ready signal VRDY is set ON before position control ready signal PRDY comes ON

0 : A servo alarm is generated.
1 : A servo alarm is not generated.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx			OPTx	

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

[Data type] Bit axis

OPTx Position detector

0 : A separate pulse coder is not used.
1 : A separate pulse coder is used.

APZx Machine position and position on absolute position detector when the absolute position detector is used

0 : Not corresponding
1 : Corresponding

> **WARNING**
> When an absolute position detector is used, after primary adjustment is performed or after the absolute position detector is replaced, this parameter must be set to 0, power must be turned off and on, then manual reference position return must be performed. This completes the positional correspondence between the machine position and the position on the absolute position detector, and sets this parameter to 1 automatically.

APCx Position detector

0 : Other than absolute position detector
1 : Absolute position detector (absolute pulse coder)

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				

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

[Data type] Bit axis

DM1x to DM3x Setting of detection multiplier

Set value			Detection multiplier
DM3x	DM2x	DM1x	
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

NOTE
When the flexible feed gear is used, do not use these parameters. Set the numerator and denominator of DMR to an appropriate values in parameters 2084 and 2085 respectively.

1820	Command multiplier for each axis (CMR)

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

[Data type] Byte axis

Set a command multiplier indicating the ratio of the least command increment to the detection unit for each axis.

Least command increment = detection unit x command multiplier

Relationship between the increment system and the least command increment

	Least command increment			
Increment system	**IS–A**	**IS–B**	**IS–C**	**Unit**
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

The value set in the parameter is obtained as follows:

(1) When command multiplier is 1/2 to 1/27

$$\text{Set value} = \frac{1}{(\text{Command multiplier})} + 100$$

Valid data range: 102 to 127

(2) When command multiplier is 1 to 48

Set value = 2 × command multiplier

Valid data range: 2 to 96

NOTE
When command multiplier is 1 to 48, the set value must be determined so that an integer can be set for command multiplier.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

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

1825	Servo loop gain for each axis

[Data type] Word axis

[Unit of data] 0.01 s^{-1}

[Valid data range] 1 to 9999

Set the loop gain for position control for each axis.

When the machine performs linear and circular interpolation (cutting), the same value must be set for all axes. When the machine requires positioning only, the values set for the axes may differ from one another. As the loop gain increases, the response by position control is improved. A too large loop gain, however, makes the servo system unstable.

The relationship between the positioning deviation (the number of pulses counted by the error counter) and the feedrate is expressed as follows:

$$\text{Positioning deviation} = \frac{\text{feedrate}}{60 \times (\text{loop gain})}$$

Unit : Positioning deviation: mm, inches, or deg
Feedrate: mm/min, inches/min, or deg/min
Loop gain: s^{-1}

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

If the positioning deviation exceeds the positioning deviation limit during movement, a servo alarm is generated, and operation is stopped immediately (as in emergency stop).
Generally, set the positioning deviation for rapid traverse plus some margin in this parameter.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for each axis.

If, in the stopped state and the positioning deviation exceeds the positioning deviation limit set for stopped state, a servo alarm is generated, and operation is stopped immediately (as in emergency stop).

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to +99999999

A grid shift is set for each axis.

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.

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

1.4.2 Absolute Position Detection

General

Even when the power to the CNC is turned off, a battery–powered pulse coder stores the current position. No reference position return is required when the power to the CNC is restored.

Signal

Absolute position detector battery voltage low alarm signal PBATL <F172#7>

[Classification] Output signal

[Function] Notifies that the life of the absolute position detector battery, which is used to keep the machine position even when the CNC power is off, is about to expire.
Generally, this signal is used to turn on a lamp for calling the operator's attention.

[Operation] The signal becomes 1 when:

- The battery voltage for the absolute position detector becomes lower than or equal to the rating.
  The battery need be replaced in the immediate future.

The signal becomes 0 when:

- The battery voltage for the absolute position detector is higher than or equal to the rating.

Absolute position detector battery voltage zero alarm signal PBATZ <F172#6>

[Classification] Output signal

[Function] Notifies that the life of the absolute position detector battery, which is used to keep the machine position even when the CNC power is off, has expired.

[Operation] The signal becomes 1 when:

- The battery voltage for the absolute position detector becomes 0 V.
  The battery need be replaced before the CNC power is turned off.

The signal becomes 0 when:

- The battery voltage for the absolute position detector is higher than or equal to 0 V.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F172	PBATL	PBATZ						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx					

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

[Data type] Bit axis

APCx Position detector

0: Other than absolute position detector
1: Absolute position detector (absolute pulse coder)

1.4.3 FSSB Setting

Overview

Connecting the CNC control section to servo amplifiers via a high–speed serial bus (FANUC Serial Servo Bus, or FSSB), which uses only one fiber optics cable, can significantly reduce the amount of cabling in machine tool electrical sections.

In a system using the FSSB, it is necessary to set up the following parameters to specify its axes. (The other parameters should be specified as usual.)

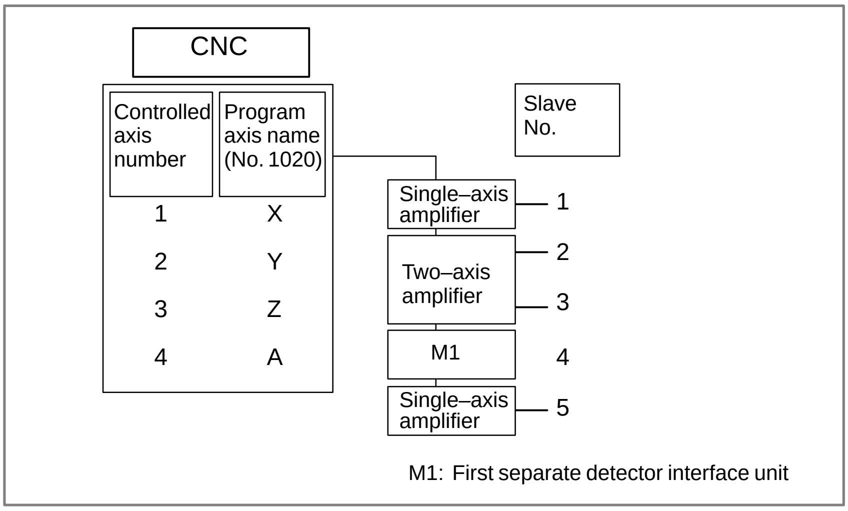
- Parameter No. 1023
- Parameter No. 1905
- Parameter Nos. 1910 to 1919
- Parameter Nos. 1936 and 1937

These parameters can be specified using the following methods:

1. Manual setting 1
   Parameters are defaulted according to the setting of parameter No. 1023. There is no need to specify parameter Nos. 1905, 1910 to 1919, 1936 and 1937. No automatic setting is used. Note that some functions are unusable.
2. Automatic setting
   Axis settings are calculated automatically according to the interrelationships between axes and amplifiers entered on the FSSB setting screen. Parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are specified automatically according to the results of the calculation.
3. Manual setting 2
   Parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are specified according to manually entered values. The user must be totally familiar with the meaning of each parameter before entering any values.

Slave

In an FSSB–based system, a fiber optics cable is used to connect the CNC to servo amplifiers and separate detector interface units. These amplifiers and separate detector interface units are called slaves. The two–axis amplifier consists of two slaves, and the three–axis amplifier consists of three slaves. The slaves are numbered 1, 2, ..., 10 (slave number) sequentially, with that nearest to the CNC starting at number 1.

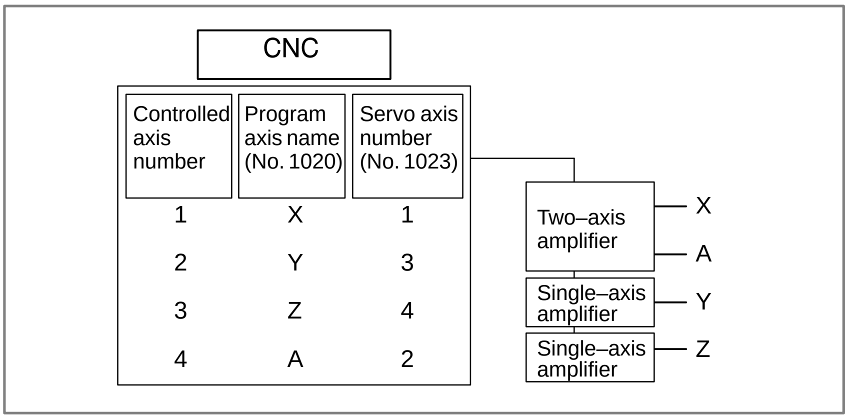


Manual setting 1

The manual setting 1 is valid when the following parameter have the following values:

Bit 0 of parameter No. 1902 = 1
Bit 1 of parameter No. 1902 = 0
Parameter Nos. 1910 to 1919 = all 0s

By manual setting 1, the value set for parameter No. 1023 when the power is switched on is regarded as a slave number. Specifically, an axis for which parameter No. 1023 is set to 1 is connected to the amplifier nearest to the CNC, while an axis for which parameter No. 1023 is set to 2 is the second one from the CNC.



By manual setting 1, some of the following functions and values cannot be used, as described below. They should be used with automatic setting or manual setting 2.

- No separate detector interface unit can be used; hence, no separate position detectors can be used.
- No number can be skipped in parameter No. 1023; for example, number 3 cannot be used for any axis unless number 2 is used.

- The following servo functions cannot be used:
  - ❑ High–speed current loop
  - ❑ High–speed interface axis

Automatic setting

Automatic setting can be used on the FSSB setting screen, if the following parameter is set as follows:

Bit 0 of parameter No. 1902 = 0

On the FSSB setting screen, automatic setting should be enabled by means of the following procedure:

1. On the amplifier setting screen, specify the axis number of a controlled axis to be connected to each amplifier.
2. Press the [SETING] soft key. (If a warning message is displayed, restart from step 1.)
3. On the axis setting screen, specify information about each axis, such as a separate detector interface unit connector No.
4. Press the [SETING] soft key. (If a warning message is displayed, repeat the procedure, starting from step 3.)

In this way, parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are set according to the results of automatic calculation. In addition, bit 1 of parameter No. 1902 is set to 1 to indicate that each parameter has been set up. Switching the power off then back on again causes axis setting to be performed according to these parameter settings.

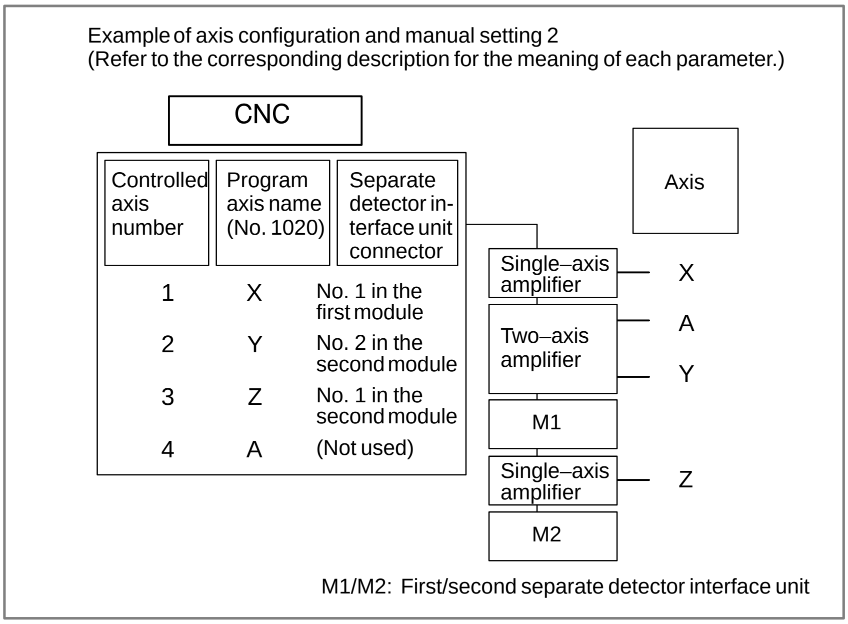
For details of the FSSB setting screen, see the FSSB data display and setting procedure, described below.

Manual setting 2

If the following parameter is set, manual setting 2 can be used for each parameter axis setting.

Bit 0 of parameter No. 1902 = 1

To perform manual setting 2, set parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937. Refer to the Parameter Manual for the definition of each parameter.



No.	1902#0 FMD
	1

No.	1910	1911	1912	1913	1914	1915	1916	1917	1918	1919
	0	1	2	16	3	48	40	40	40	40

No.	1023	1905#0 FSL	1905#6 PM1	1905#7 PM2	1936	1937
X	1	0	1	0	0	0
Y	3	0	0	1	0	1
Z	4	1	0	1	0	0
A	2	1	0	0	0	0

FSSB display and setting procedure

• Display

The FSSB setting screen displays FSSB–based amplifier and axis information. This information can also be specified by the operator.

1. Press the SYSTEM function key.
2. To display [FSSB], press the ▷ next menu page key several times.
3. Pressing the [FSSB] soft key causes the **[AMPLIFIER SETTING]** screen (or the previously selected FSSB setting screen) to appear, with the following soft keys displayed.

```
[  AMP  ][  AXIS  ][ MAINTE ][      ][ (OPRT) ]
```

The FSSB setting screens include: **[AMPLIFIER SETTING]**, **[AXIS SETTING]**, and **[AMPLIFIER MAINTENANCE]**

Pressing the [AMP] soft key causes the **[AMPLIFIER SETTING]** screen to appear.
Pressing the [AXIS] soft key causes the **[AXIS SETTING]** screen to appear.
Pressing the [MAINTE] soft key causes the **[AMPLIFIER MAINTENANCE]** screen to appear.

(1) Amplifier setting screen
The amplifier setting screen consists of two sections: the first section displays information about the amplifiers, while the second section displays information about the separate detector interface units.

```
AMPLIFIER SETTING                       O1000 N00001
  NO.  AMP   SERIES    UNIT     CUR.   AXIS NAME
   1   A1-L    α      SVM-HV    40AL     1    X
   2   A2-L    α         SVM     12A     2    Y
   3   A2-M    α         SVM     40A     3    Z
   5   A3-L    β         SVM     20A     4    A




  NO.  EXTRA  TYPE   PCB ID
   4    M1     A     0000 DETECTOR(4AXES)
   6    M2     B     12AB
>_
MDI **** *** ***      13:11:56
[  AMP  ][  AXIS  ][ MAINTE ][      ][ (OPRT) ]
```

The amplifier setting screen consists of the following items:

- NO. . . slave number
  The numbers of up to ten slaves (up to six amplifiers and up to two separate detector interface units) connected via the FSSB are displayed sequentially, with the one nearest the CNC being number 1.

- AMP . . . . amplifier type
  The amplifier type display consists of the letter A, which stands for “amplifier”, a number that indicates the placing of the amplifier, as counted from that nearest to the CNC, and a letter such as L (first axis) or M (second axis) indicating the placing of the axis in the amplifier.
- AXIS . . . . controlled axis number
  The axis number of each controlled axis specified in parameters (Nos. 1920 to 1929) is displayed. If a number specified in these parameters falls outside the range of 1 and the maximum number of controlled axes, 0 is displayed.
- NAME . . . controlled axis name
  The axis name assigned to a parameter (No. 1020) corresponding to a particular controlled axis number is displayed. If the controlled axis number is 0, – is displayed.
- The following items are displayed as amplifier information:
  - UNIT . . . . . servo amplifier unit type
  - SERIES . . . servo amplifier series
  - CUR. . . . . . maximum rating current
- The following items are displayed as separate detector interface unit information:
  - EXTRA
    This consists of the letter M, which stands for “separate detector interface unit”, and a number indicating the placing of the separate detector interface unit, as counted from that nearest to the CNC.
  - TYPE
    This is a letter indicating the type of the separate detector interface unit.
  - PCB ID
    This consists of four digits indicating the separate detector interface unit ID (hexadecimal). The separate detector interface unit ID is followed by DETECTOR (8AXES) when 8–axes separate detector interface unit or DETECTOR (4AXES) when 4–axes separate detector interface unit.

(2) Axis setting screen

The axis setting screen displays the information shown below:

```
AXIS SETTING                             O1000 N00001
  AXIS   NAME   AMP      M1  M2   1-DSP   CS   TNDM
    1      X    A1-L     0   0      0     0     1
    2      Y    A2-L     1   0      1     0     0
    3      Z    A2-M     0   0      0     1     0
    4      A    A3-L     0   0      0     0     2

>_
MDI **** *** ***      13:11:56
[  AMP   ][  AXIS   ][ MAINTE ][      ][ (OPRT) ]
```

The axis setting screen displays the following items:

- AXIS . . . . controlled axis number
  This item is the placing of the NC controlled axis.
- NAME . . . controlled axis name
- AMP . . . . type of the amplifier connected to each axis
- M1 . . . . . . connector number for separate detector interface unit 1
  This item is the number of the connector for separate detector interface unit 1, specified in parameter No. 1931.
- M2 . . . . . . connector number for separate detector interface unit 2
  This item is the number of the connector for separate detector interface unit 2, specified in parameter No. 1932.
- 1–DSP
  This item is the value specified in bit 0 (parameter 1DSP) of parameter No. 1904. It is 1 for an axis (such as a learning control axis, high–speed current loop axis, or high–speed interface axis) that exclusively uses a DSP, which is usually shared by two axes.
- CS . . . . . Cs contour controlled axis
  This item is the value specified in parameter No. 1933. It is 1 for the Cs contour controlled axis.
- TNDM
  This item is the number specified in parameter No. 1934. Consecutive odd and even numbers are displayed for the master and slave axes for tandem control. This item is not used in Series 0*i* Mate.

(3) Amplifier maintenance screen

The amplifier maintenance screen displays maintenance information for servo amplifiers. This screen consists of the following two pages, either of which can be selected by pressing the PAGE ↑ or PAGE ↓ key.

```
AMPLIFIER MAINTENANCE               O1000 N00001
  AXIS   NAME AMP    SERIES  UNIT   AXES   CUR.
   1       X  A1-L     α    SVM-HV    2    40AL
   2       Y  A1-M     α       SVM    2     12A
   3       Z  A2-L     β       SVM    1     40A
   4       A  A3-L     α       SVM    2     20A
   5       B  A3-M     α       SVM    2     40A
   6       C  A4-L     α       SVU    1    240A






>_
MDI **** *** ***     13:11:56
[  AMP  ][  AXIS  ][ MAINTE ][      ][       ]
```

```
AMPLIFIER MAINTENANCE               O1000 N00001
  AXIS   NAME    EDITION      TEST    MEINTE-NO.
   1       X       01A       010123       01
   2       Y       01A       010123       01
   3       Z       01A       010123       01
   4       A       02B       010123       01
   5       B       02B       010123       01
   6       C       02B       010123       01






>_
MDI **** *** ***     13:11:56
[  AMP  ][ AXIS ][ MAINTE ][       ][       ]
```

The amplifier maintenance screen displays the following items:

- AXIS . . . . controlled axis number
- NAME . . . . controlled axis name
- AMP . . . . type of amplifier connected to each axis
- SERIES . . . . servo amplifier series of an amplifier connected to each axis
- UNIT . . . . unit type of a servo amplifier connected to each axis
- AXES . . . . maximum number of axes controlled by an amplifier connected to each axis
- CUR. . . . . maximum rating current for amplifiers connected to each axis
- EDITION . . . . . . unit version number of an amplifier connected to each axis

- TEST . . . . date of test performed on an amplifier connected to each axis
  Example) 010123 = January 23, 2001
- MEINTE–No. . . engineering change number for an amplifier connected to each axis

- **Setting**

On an FSSB setting screen (other than the amplifier maintenance screen), pressing the [(OPRT)] soft key displays the following soft keys:

```
[ SETING ][        ][ CANCEL ][        ][ INPUT ]
```

To enter data, place the machine in MDI mode or the emergency stop state, position the cursor to the point where a desired item is to be input, then enter the desired data and press the [INPUT] soft key (or the INPUT key on the MDI panel).

When the [SETING] key is pressed after data has been entered, a warning message is displayed if the entered data contains an error. When the data is valid, the corresponding parameter is set up.

To restore the previous value of a parameter if, for example, an entered value is incorrect, press the [CANCEL] soft key.

When the power is switched on, values are read from the parameters and displayed on the screen.

CAUTION

1 For the parameters to be specified on the FSSB setting screen, do not attempt to enter values on the parameter screen using the MDI or a G10 command. Use only the FSSB screen to enter values for these parameters.

2 If pressing the [SETING] key results in a warning message being displayed, retry data entry, or press the [CANCEL] key to clear the warning message. Note that pressing the reset key does not clear the warning message.

(1) Amplifier setting screen

```
AMPLIFIER SETING                              O1000 N00001
  NO.  AMP   SERIES    UNIT     CUR.   AXIS NAME
   1   A1-L    α     SVM-HV     40AL    1     X
   2   A2-L    α        SVM      12A    2     Y
   3   A2-M    α        SVM      40A    3     Z
   5   A3-L    β        SVM      20A    4     A



  NO.  EXTRA  TYPE   PCB ID
   4    M1     A     0000 DETECTOR(4AXES)
   6    M2     B     12AB
>_
MDI **** *** ***      13:11:56
[  AMP  ][  AXIS  ][ MAINTE ][        ][ (OPRT) ]
```

The amplifier setting screen displays the following items:

- AXIS . . . controlled axis number
  For this item, enter a value of between 1 and the maximum number of controlled axes. If a number that falls outside this range is entered, the warning message FORMAT ERROR appears. If the entered controlled axis number is duplicate or 0, the warning message DATA IS OUT OF RANGE appears when the [SETING] soft key is pressed to assert the entered value. In this case, no value can be entered for the parameter.

(2) Axis setting screen

```
AXIS SETTING                            O1000 N00001
  AXIS   NAME   AMP     M1  M2   1-DSP CS   TNDM
    1      X    A1-L     0   0     0    0     1
    2      Y    A2-L     1   0     1    0     0
    3      Z    A2-M     0   0     0    1     0
    4      A    A3-L     0   0     0    0     2









>_
MDI **** *** ***      13:11:56
[ SETING ] [       ] [ CANCEL ] [       ] [ INPUT ]
```

On the axis setting screen, the following items can be specified:

- M1 . . . connector number for separate detector interface unit 1
  For an axis that uses separate detector interface unit 1, enter a connector number using a number in the range of between 1 and the maximum number of axes for separate detector interface unit 1. When separate detector interface unit 1 need not be used, enter 0. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- M2 . . . connector number for separate detector interface unit 2
  For an axis that uses separate detector interface unit 2, enter a connector number using a number in the range of between 1 and the maximum number of axes for separate detector interface unit 2. When separate detector interface unit 2 need not be used, enter 0. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- 1–DSP
  Enter 1 for the following axes, each of which exclusively uses a DSP, which is usually shared by two axes. If a number other than 0 or 1 is entered, the message FORMAT ERROR is displayed.
  - High–speed current loop axis
  - High–speed interface axis
- CS . . Cs contour controlled axis
  Enter 1 for the Cs contour controlled axis. If a number other than 0 or 1 is entered, the message FORMAT ERROR is displayed.

- TNDM
  Enter odd and even numbers for the master and slave axes for tandem control. These numbers must be consecutive and in the range of between 1 and 4. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
  This item is not used in Series 0*i* Mate.

When the [SETING] soft key is pressed on the axis setting screen after data entry, the message DATA IS OUT OF RANGE is displayed if any of the following conditions is satisfied.

- Both M1 and M2 are nonzero for an axis.
- Any two of the 1–DSP, CS, and TNDM are nonzero for an axis.
- A duplicate value is specified for M1.
- A duplicate value is specified for M2.
- A duplicate value is specified for CS.
- A duplicate value is specified for TNDM.
- An invalid master/slave axis pair is specified for TNDM.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1902							ASE	FMD

[Data type] Bit

FMD Specifies the FSSB setting mode.

0 : Automatic setting mode.
(If the interrelationships between axes and amplifiers are specified on the FSSB setting screen, parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 are set automatically.)
1 : Manual setting 2 mode.
(Parameter Nos. 1023, 1905, 1910 to 1919, 1936, and 1937 must be set manually.)

ASE Indicates whether automatic setting is complete, if bit 0 of parameter No. 1902 is 0 (automatic setting mode).

0 : Incomplete.
1 : Complete.

(This bit automatically becomes 1 upon the completion of automatic setting.)

	#7	#6	#5	#4	#3	#2	#1	#0
1904								DSP

[Data type] Bit axis

DSP 0 : Two axes share a DSP. (Ordinary axis)
1 : One axis occupies a DSP. (Learning control axis, and so on)

Usually, the user should not attempt to manipulate this bit, because it is set using the FSSB setting screen. It need not be used in FSSB manual setting 2 mode.

	#7	#6	#5	#4	#3	#2	#1	#0
1905	PM2	PM1						FSL

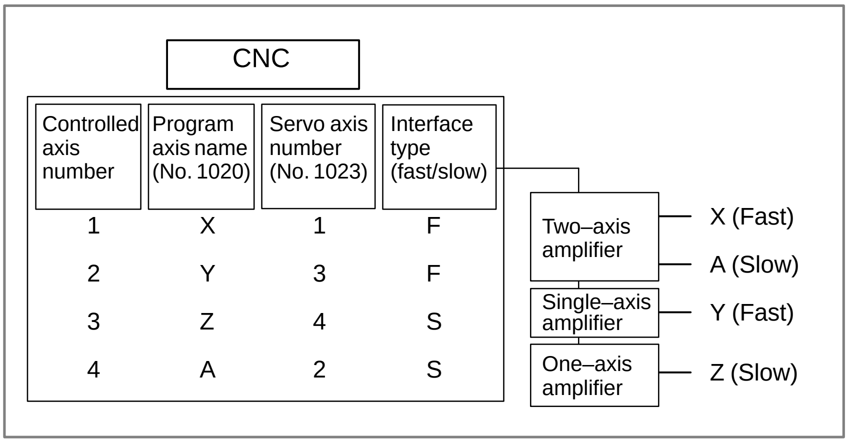
[Data type] Bit axis

FSL Specifies whether to use a fast or slow interface between a servo amplifier and the servo software.

0 : Fast type
1 : Slow type

There are two types of servo data transfer interfaces: fast and slow types. They are selected as described below.

- Both types are usable for single–axis amplifiers.
- For two–axis amplifiers, do not use fast type interfaces for both axes simultaneously. Slow types can be used simultaneously for both axes.
- For three–axis amplifiers, the same rules as those for two–axis amplifiers apply for the first and second axes, while the same rules as those for single–axis amplifiers apply for the third axis.
- For those axes for which an odd number is set for parameter No. 1023, the fast type must be used, except for the high–speed current loop and high–speed interface axes, for which the slow type can also be used.
- For those axes for which an even number is set for parameter No. 1023, only the slow type is usable; this bit must be set to 1.



PM1 Specifies whether the first separate detector unit is to be used.

0 : Not used.
1 : Used.

PM2 Specifies whether the second separate detector unit is to be used.

0 : Not used.
1 : Used.

If automatic setting is set as the FSSB setting mode (bit 0 of parameter No. 1902 = 0), this parameter is set automatically when data is entered using the FSSB setting screen. For manual setting 2 (bit 0 of parameter No. 1902 = 1), the user must set this parameter. When using a separate detector interface unit, a connection number must be specified separately (parameter Nos. 1936 and 1937).

1910	Address conversion table value for slave 1 (ATR)
1911	Address conversion table value for slave 2 (ATR)
1912	Address conversion table value for slave 3 (ATR)
1913	Address conversion table value for slave 4 (ATR)
1914	Address conversion table value for slave 5 (ATR)
1915	Address conversion table value for slave 6 (ATR)
1916	Address conversion table value for slave 7 (ATR)
1917	Address conversion table value for slave 8 (ATR)
1918	Address conversion table value for slave 9 (ATR)
1919	Address conversion table value for slave 10 (ATR)

[Data type] Byte

[Valid data range] 0 to 7, 16, 40, and 48

An address conversion table value must be specified for each of slaves 1 to 10. The term “slave” refers to any of the servo amplifiers and separate detector interface units connected to the CNC. Each slave is assigned a number of between 1 and 10 sequentially, with the one nearest to the CNC assigned number 1. A two–axis amplifier is regarded as being two slaves, while a three–axis amplifier is regarded as being three slaves. Each of these parameters is set depending on whether the slave is an amplifier or separate detector interface unit, as follows:

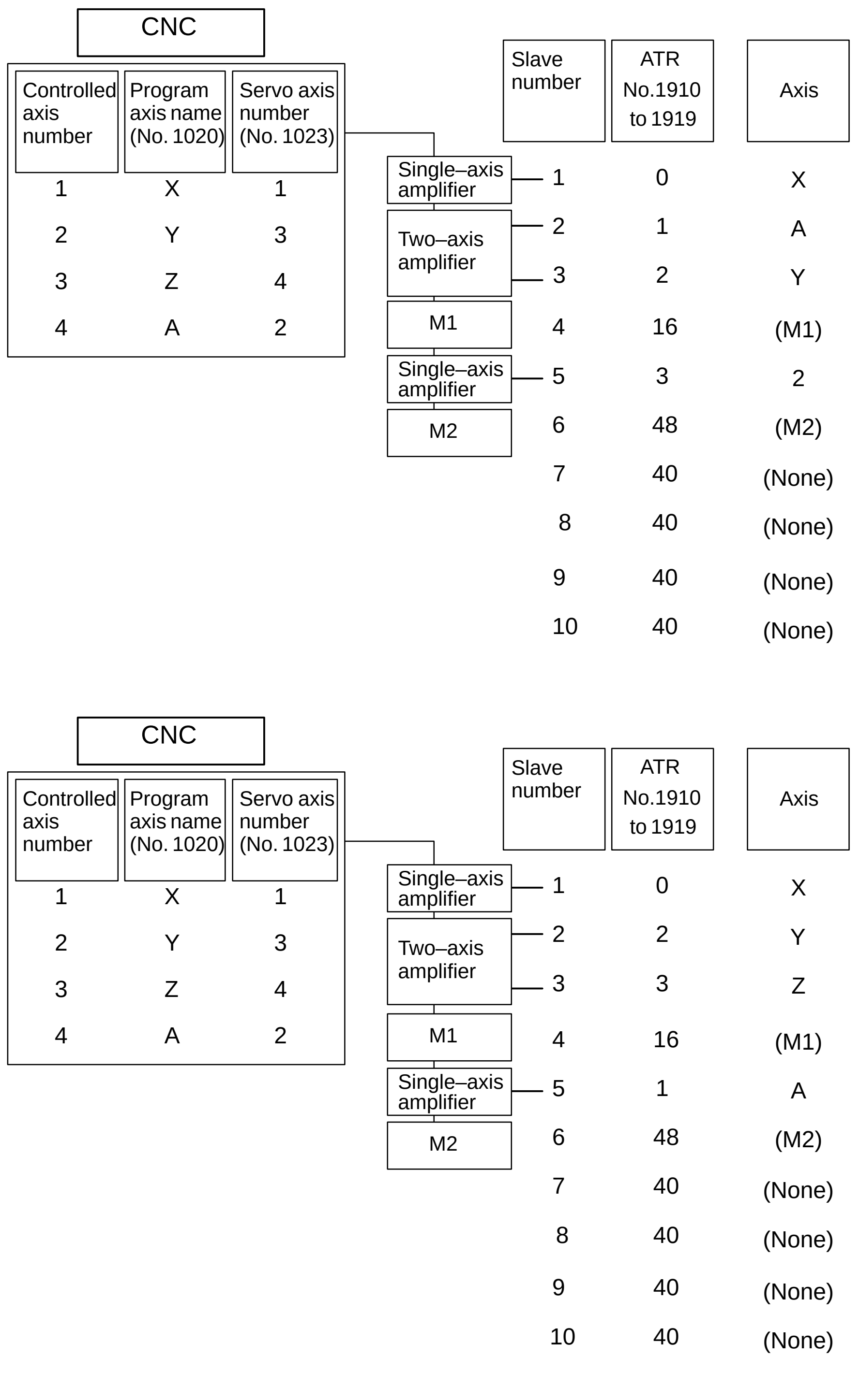
○ When the slave is an amplifier:
The parameter is set to the “value in parameter No. 1023 for an axis to which the amplifier is assigned,” minus 1.

○ When the slave is a separate detector interface unit:
The parameter is set to 16 for the first separate detector interface unit (that nearest to the CNC) or to 48 for the second separate detector interface unit (that farthest from the CNC).

○ When there is no slave:
The parameter is set to 40.

If automatic setting is set as the FSSB setting mode (bit 0 of parameter No. 1902 = 0), this parameter is set automatically when data is entered on the FSSB setting screen. For manual setting 2 (bit 0 of parameter No. 1902 = 1), the parameter must be set manually.

○ Example of axis configuration and parameter setting



M1/M2: First/second separate detector interface unit

1920	Controlled axis number for slave 1 (FSSB setting screen only)
1921	Controlled axis number for slave 2 (FSSB setting screen only)
1922	Controlled axis number for slave 3 (FSSB setting screen only)
1923	Controlled axis number for slave 4 (FSSB setting screen only)
1924	Controlled axis number for slave 5 (FSSB setting screen only)
1925	Controlled axis number for slave 6 (FSSB setting screen only)
1926	Controlled axis number for slave 7 (FSSB setting screen only)
1927	Controlled axis number for slave 8 (FSSB setting screen only)
1928	Controlled axis number for slave 9 (FSSB setting screen only)
1929	Controlled axis number for slave 10 (FSSB setting screen only)

[Data type] Byte

[Valid data range] 0 to 8

Each of these parameters is set using a controlled axis number for a slave numbered 1 to 10. This parameter is set automatically when data is entered using the FSSB setting screen; do not specify it manually. For manual setting 2, the parameter need not be set.

1931	Connector number for first separate detector interface unit (FSSB setting screen only)
1932	Connector number for second separate detector interface unit (FSSB setting screen only)

[Data type] Byte axis

[Valid data range] 0 to the number of connectors in each separate detector interface unit

To use a separate detector interface unit, the user must specify a connector number for the separate detector interface unit on each axis. This parameter is set automatically when data is entered using the FSSB setting screen; do not specify it manually. For manual setting 2, the parameter need not be set.

1933	Cs contour controlled axis (FSSB setting screen only)

[Data type] Byte axis

[Valid data range] 0 and 1

To use Cs contour control, this parameter must be set to 1 for the corresponding axis. This parameter is set automatically when data is entered using the FSSB setting screen; do not specify it manually. For manual setting 2, the parameter need not be set.

1934	Master/slave number for tandem–controlled axes (FSSB setting screen only)

[Data type] Byte axis

[Valid data range] 0 to 8

To use tandem control, this parameter must be set to an odd number (for the master axis) or to an even number (slave axis). This parameter is set automatically when data is entered using the FSSB setting screen; do not set it manually. For manual setting 2, the parameter need not be set.

1936	Connector number for first separate detector interface unit
1937	Connector number for second separate detector interface unit

[Data type] Byte axis

[Valid data range] 0 to 7

To use a separate detector interface unit, this parameter must be set to "connection number for the separate detector interface unit," minus 1; that is, 0 for connector number 1, 1 for connector number 2, and so on. It is also necessary to set up bits 6 and 7 of parameter No. 1905. For an axis that does not use a separate detector interface unit, 0 is specified for the parameter. Any connector can be used for any axis, however the connectors in a single separate detector interface unit should be used in ascending order of connector number. For instance, connector 4 of a separate detector interface unit cannot be used without using connector 3 of the same separate detector interface unit.

Example)

Controlled axis	Connector number for first separate detector interface unit	Connector number for second separate detector interface unit	No.1936	No.1937	No.1905 (#7, #6)
X	1	Not used	0	0	0,1
Y	Not used	2	0	1	1,0
Z	Not used	1	0	0	1,0
A	Not used	Not used	0	0	0,0

If the FSSB setting mode is automatic setting (bit 0 of parameter No. 1902 = 0), this parameter is set automatically when data is entered using the FSSB setting screen. For manual setting 2 (bit 0 of parameter No. 1902 = 1), the user must set this parameter.

Alarm and message

- **Servo alarms**

Number	Message	Description
456	ILLEGAL CURRENT LOOP	The current control cycle settings (parameter No. 2004, bit 0 of parameter No. 2003, and bit 0 of parameter No. 2013) are incorrect. Possible problems are as follows. – For the two axes whose servo axis numbers (settings of parameter No. 1023) are an odd number followed by an even number (a pair of axes 1 and 2, for example), a different current control cycle is set for each of the axes. – The requirements for slaves needed for the set current control cycle, including the number, type, and connection method of them, are not satisfied.
457	ILLEGAL HI HRV (250US)	Use of high–speed HRV is specified although the current control cycle is 200 μs.
458	CURRENT LOOP ERROR	The current control cycle setting does not match the actual current control cycle.
459	HI HRV SETTING ERROR	For the two axes whose servo axis numbers (settings of parameter No. 1023) are an odd number followed by an even number (a pair of axes 1 and 2, for example), the SVM for one of the axes supports high–speed HRV control but the SVM for the other does not. Refer to the SVM specification.
460	n AXIS : FSSB DISCONNECT	FSSB communication was interrupted. The most likely causes are: 1. The FSSB communication cable is disconnected or has a broken conductor. 2. The amplifier power supply was turned off. 3. A low–voltage alarm condition occurred in the amplifier.
461	n AXIS : ILLEGAL AMP INTERFACE	The fast type interface was assigned to both axes of a two–axis amplifier.
462	n AXIS : SEND CNC DATA FAILED	The slave could not receive data correctly because of an FSSB communication error.
463	n AXIS : SEND SLAVE DATA FAILED	The servo section failed to receive data correctly because of an FSSB communication error.
465	n AXIS : READ ID DATA FAILED	An attempt to read the initial ID information for the amplifier failed when the power was switched on.

Number	Message	Description
466	n AXIS : MOTOR/AMP COMBINATION	The maximum current rating for the amplifier does not match that for the motor.
467	n AXIS : ILLEGAL SETTING OF AXIS	The servo function for the following has not been enabled when an axis occupying a single DSP is specified on the axis setting screen. 1. High–speed current loop (bit 0 of parameter No. 2004 = 1) 2. High–speed interface axis (bit 4 of parameter No. 2005 = 1)
468	HI HRV SETTING ERROR(AMP)	Use of high–speed HRV is specified for a controlled axis of an amplifier which does not support high–speed HRV.

- **P/S alarms**

Number	Message	Description
5134	FSSB : OPEN READY TIME OUT	The FSSB did not become ready to open during initialization.
5135	FSSB : ERROR MODE	The FSSB entered an error mode.
5136	FSSB : NUMBER OF AMPS IS SMALL	The number of amplifiers recognized by the FSSB is insufficient, compared with the number of controlled axes.
5137	FSSB : CONFIGURATION ERROR	The FSSB detected a configuration error. The address conversion table value (ATR) setting (parameter Nos. 1910 to 1919 and 1970 to 1979) for a slave axis does not match the type of a slave axis actually connected to the FSSB.
5138	FSSB : AXIS SETTING NOT COMPLETE	Axis setting has not been performed in automatic setting mode. Perform axis setting using the FSSB setting screen.
5139	FSSB : ERROR	The initialization of the servo was not completed normally. Probable cases are a defect of optical cable or a mistake of the connection between the amplifier and other modules.
5197	FSSB : OPEN TIME OUT	The FSSB did not open when the CNC had allowed the FSSB to open.

Number	Message	Description
5198	FSSB : ID DATA NOT READ	The initial ID information for the amplifier cannot be read because of a failure in the temporary assignment.
5311	FSSB: ILLEGAL CONNECTION	A connection related to FSSB is illegal. This alarm is issued when either of the following is found: 1 Two axes having adjacent servo axis numbers (parameter No. 1023), odd number and even number, are assigned to amplifiers to which different FSSB systems are connected. 2 The system does not satisfy the requirements for performing HRV control, and use of two pulse modules connected to different FSSB systems having different FSSB current control cycles is specified.

1.4.4 Tentative Absolute Coordinate Setting

General

In a full–closed system with a built–in absolute position detector (serial pulse coder) and incremental linear scale, a coordinate system is set up, using absolute position data received from the built–in absolute position detector when the power is switched on. After this, position control is carried out using the linear scale incremental data. Because the machine position obtained immediately after the power is switched on is tentative, obtaining the accurate machine position requires making a manual reference position return.

Even before a reference position return is made, using this function enables a stroke limit, although the machine position obtained when the power is switched on is approximate.

Note that this function is not intended to use an incremental linear scale as an absolute position detector.

This function is an option.

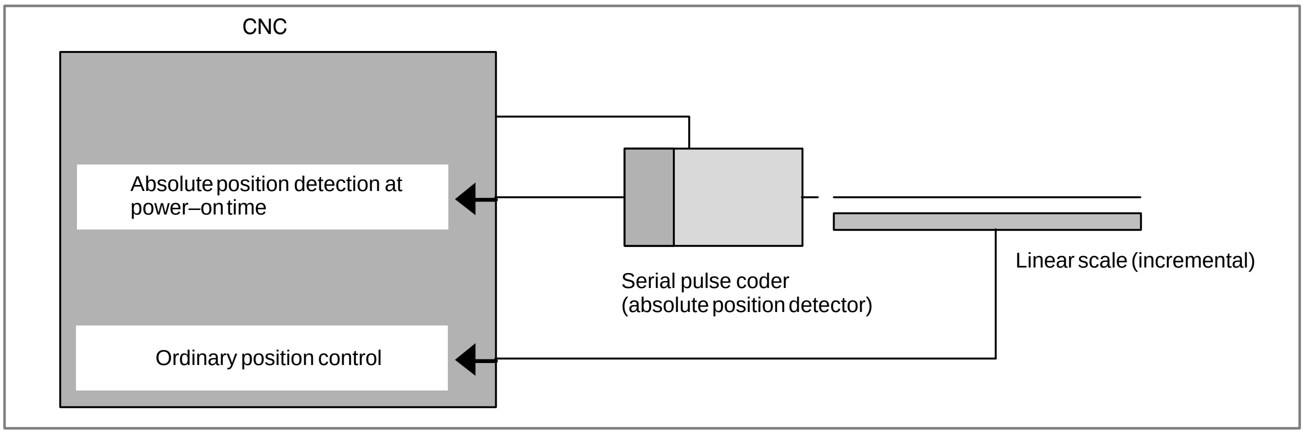


Fig. 1.4.4 System using tentative absolute coordinate system setting

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1801					INA			

[Data type] Bit

INA Specifies whether to perform absolute position communication for re–setting up the machine position at a reset after a servo alarm other than alarm 413 (LSI overflow), 416, 445, 446, or 447 (broken–wire alarm) occurs in a system that uses an absolute position detector, as follows:

0 : To perform.
1 : Not to perform.

> **NOTE**
> 1 If INA = 0 in a system that uses tentative absolute coordinate system setting, the machine position is approximate after a servo alarm other than 413, 416, 445, 446, or 447 is reset, because it has been re–set up using data received from the built–in absolute position coder.
> 2 When a reset is performed after a servo alarm 413, 416, 445, 446, or 447 occurs, absolute position communication for re–setting up the machine position always takes place.

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx				OPTx	

[Data type] Bit axis

OPTx Specifies whether to use a separate position detector, as follows:

0 : Not to use.
1 : To use.

APCx Specifies whether to use a absolute position detector, as follows:

0 : Not to use.
1 : To use.

> **NOTE**
> 1 When using tentative absolute coordinate system setting, set both OPTx and APCx to 1.
> 2 After setting any of these parameters, turn the power off then on again so that the setting will take effect.

1874	Flexible feed gear numerator for built–in position detector
1875	Flexible feed gear denominator for built–in position detector

[Data type] Word axis

[Valid data range] 1 to 32,767

Specifies a flexible feed gear for a built–in position detector for each axis when using tentative coordinate system setting, according to the following expression:

$$\frac{\text{NO.1874}}{\text{NO.1875}} = \frac{\text{Number of position feedback pulses per motor revolution}}{1,000,000}$$

> **NOTE**
> 1 These parameters can be used also in a system that uses the Inductosyn.
> 2 After setting any of these parameters, turn the power off then on again so that the setting will take effect.

	#7	#6	#5	#4	#3	#2	#1	#0
2011	XIAx							

[Data type] Bit axis

INI Specifies whether to enable tentative absolute coordinate system setting, as follows:

0 : To disable.
1 : To enable.

> **NOTE**
> Using tentative absolute coordinate system setting requires setting bit 1 (OPTx) of parameter No. 1815, bit 5 (APCx) of parameter No. 1815, parameter No. 1874, and parameter No. 1875.

1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS

1.5.1 Machine Coordinate System

General

Machine coordinate system is a coordinate system set with a zero point proper to the machine system.
A coordinate system in which the reference position becomes the parameter-preset (No. 1240) coordinate value when manual reference position return is performed, is set. With G53 command, the machine coordinate system is selected and the axis can be moved at rapid traverse to the position expressed by the machine coordinates.

High–speed G53 function

This function enables the inter–rapid traverse block overlap function between machine coordinate system select command (G53) and positioning (rapid traverse) command (G00) blocks and allows the next rapid traverse command (G00) to be executed at the end of the machine coordinate system select command (G53) without decelerating to a stop. This way, high–speed positioning becomes possible even when the machine coordinate system select command (G53) is used.
Specifying P1 in the G53 bock enables the high–speed G53 function.

G53 IP_ P1;

G53: Machine coordinate system select command G code (00 group)
IP_: End–point dimension word
P1: Enable high–speed G53 function

Parameter

1240	Coordinate value of the reference position on each axis in the machine coordinate system

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

1722	Rapid traverse deceleration rate at inter–rapid traverse block overlap

[Data type] Byte axis

[Unit of data] %

[Valid data range] 0 to 100

If a high–speed G53 command (G53 P1) block is followed by a rapid traverse block, the latter block (rapid traverse) is executed after the feedrate specified for each axis in the G53 P1 block has decreased to the deceleration rate specified in this parameter.

> **NOTE**
> 1 Enabling the high–speed G53 function does not require setting the RTO parameter (bit 4 of parameter No. 1601) for enabling inter–rapid traverse block overlap to 1.
> 2 If the RTO parameter (bit 4 of parameter No. 1601) is set to 1, a value specified in parameter No. 1722 is used also for ordinary inter–rapid traverse block overlap.

Warning

> **WARNING**
> Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute–position detector is attached.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.7.1	MACHINE COORDINATE SYSTEM
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.7.1	MACHINE COORDINATE SYSTEM
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.7.1	MACHINE COORDINATE SYSTEM
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.7.1	MACHINE COORDINATE SYSTEM

1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair

General

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

Setting a workpiece coordinate system

A workpiece coordinate system can be set using one of three methods:

(1) **Method using G92** (G50 for G code system A)
A workpiece coordinate system is set by specifying a value after G92 (G50) in the program.

(2) **Method of using G54 to G59**
Six workpiece coordinate systems are set in advance, using the MDI panel, and the workpiece coordinate system to be used is selected using program commands G54 to G59.

(3) **Method of specifying the workpiece coordinate system counter**
If the WKINC parameter (bit 4 of parameter No. 3108) has been set, pressing an axis address and the [INP.C.] soft key on the workpiece coordinate system screen in succession causes the relative coordinate value of the specified axis to be set as workpiece coordinate system data at the cursor position.

Selecting a workpiece coordinate system

The user can choose from set workpiece coordinate systems as described below.

(1) **Selecting a workpiece coordinate system set by G92 (G50)**
Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

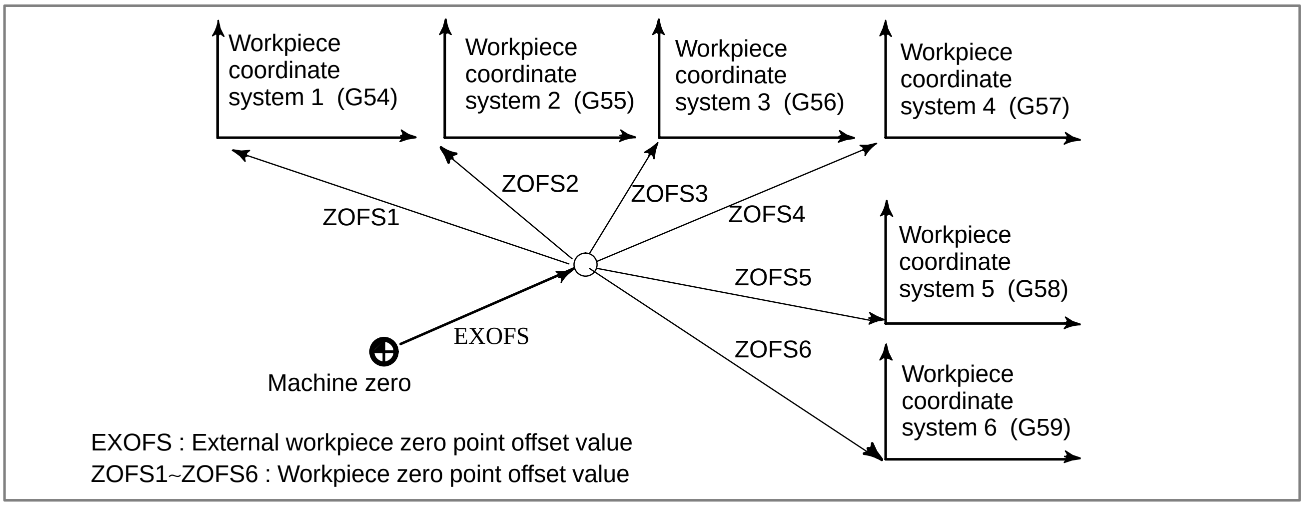
(2) **Choosing from six workpiece coordinate systems set using the MDI panel**
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 Workpiece coordinate system 1
G55 Workpiece coordinate system 2
G56 Workpiece coordinate system 3
G57 Workpiece coordinate system 4
G58 Workpiece coordinate system 5
G59 Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected as default.

Changing workpiece coordinate system

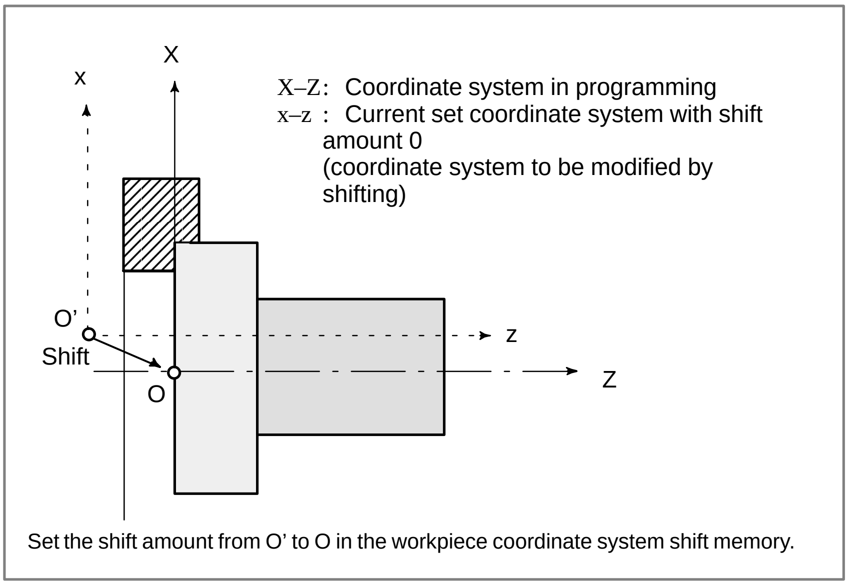
The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.
(1) Inputting from the MDI panel
(2) Programming by G10 or G92 (G50)
(3) Using external data input (refer to 15.2)



Changing an external workpiece zero point offset value or workpiece zero point offset value

Workpiece coordinate system shift (T series)

When the coordinate system actually set by the G92 (G50) command deviates from the programmed workpiece coordinate, the set coordinate system can be shifted.
Set the desired shift amount in the workpiece coordinate system shift memory.



Workpiece Coordinate System shift

Addition of workpiece coordinate system pair (M series)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1201			AWK					

[Data type] Bit

AWK Action taken after the workpiece zero point offset value is changed

0 : The absolute coordinate value is changed when the first automatic operation is performed.
1 : The absolute coordinate value is changed immediately.(If automatic operation is not in the start–up sequence)

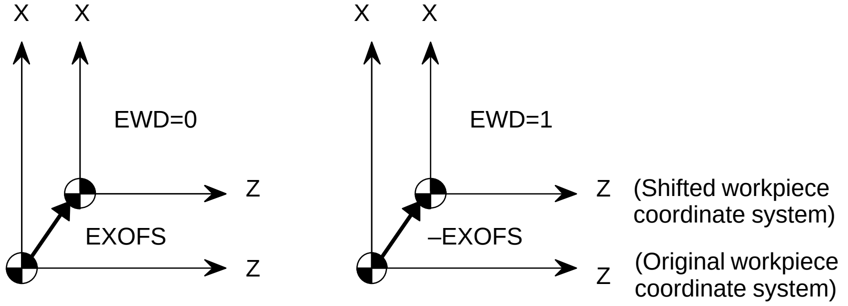
NOTE
In either case, the change becomes effective when the next block is buffered.

	#7	#6	#5	#4	#3	#2	#1	#0
1202						G50	EWS	EWD

[Data type] Bit

EWD The shift direction of the workpiece coordinate system is:

0 : The direction specified by the external workpiece zero point offset value
1 : In the opposite direction to that specified by the external workpiece zero point offset value



EXOFS : External workpiece zero point offset value

EWS Shift value of the workpiece coordinate system and external workpiece zero point offset value are

0 : Stored in the separate memory areas.
1 : Stored in the same memory area, that is, the shift and the offset values are the same.

G50 When the CNC has commands G54 to G59 specifying workpiece coordinate systems (optional function), if the G50 command for setting a coordinate system (or the G92 command in G command system B or C) is specified,

0 : The G50 (or G92) command is executed without an alarm.
1 : P/S alarm No. 010 is issued and the G50 (or G92) command is not executed.

1220	External workpiece zero point offset value

[Data type] Two–word axis

[Unit of data]

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

This is one of the parameters that give the position of workpiece coordinate system (G54 to G59). It gives an offset of the workpiece zero point common to all workpiece coordinate systems. In general, the offset varies depending on the workpiece coordinate systems. The parameter value can also be set from the PMC by using the external data input function.

1221	Workpiece zero point offset value in workpiece coordinate system1 (G54)
1222	Workpiece zero point offset value in workpiece coordinate system2 (G55)
1223	Workpiece zero point offset value in workpiece coordinate system3 (G56)
1224	Workpiece zero point offset value in workpiece coordinate system4 (G57)
1225	Workpiece zero point offset value in workpiece coordinate system5 (G58)
1226	Workpiece zero point offset value in workpiece coordinate system6 (G59)

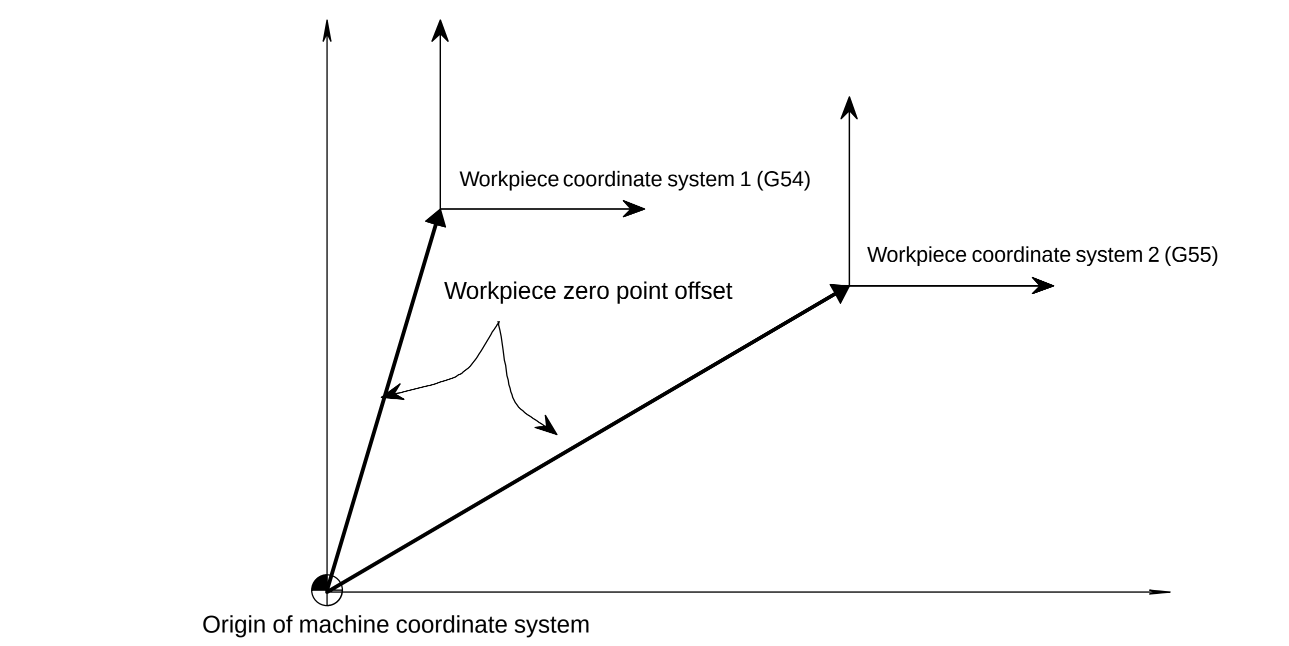
[Data type] Two–word axis

[Unit of data]

Input increment	IS–A	IS–B	IS–C	Unit
Linear axis (input in mm)	0.01	0.001	0.0001	mm
Linear axis (input in inches)	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

The workpiece zero point offset values in workpiece coordinate systems 1 to 6 (G54 to G59) are set.



	#7	#6	#5	#4	#3	#2	#1	#0
3108				WCI				

[Data type] Bit

WCI On the workpiece coordinate system screen, a counter input is:

0 : Disabled.
1 : Enabled.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.7.2	WORK PIECE COORDINATE SYSTEM
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.7.2	WORK PIECE COORDINATE SYSTEM
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.7.2	WORK PIECE COORDINATE SYSTEM
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.7.2	WORK PIECE COORDINATE SYSTEM

1.5.3 Rotary Axis Roll Over

General

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 0 (ROAx) of parameter 1008 to 1.

For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values rounded by the angle corresponding to one rotation set in parameter No. 1260. The tool moves in the direction in which the final coordinates are closest when bit 1 (RABx) of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (RRLx) of parameter No. 1008 is set to 1.

Parameter

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

> **NOTE**
> After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis (1) Inch/metric conversion is done. (2) All coordinate values are linear axis type. (Not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A Type) (1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008 #0 and #2. (3) Stored pitch error compensation is the rotation type. (Refer to parameter No. 3624) (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) (1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°) (3) Stored pitch error compensation is linear axis type (Refer to parameter No. 3624) (4) The rotation axis roll over function and index table indexing function (M series) cannot be used.

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

> **NOTE**
> After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit axis

ROAx The roll–over function of a rotation axis is
0 : Invalid
1 : Valid

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

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

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

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

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

1260	Move amount per rotation of rotary axis

> **NOTE**
> When this parameter is changed, turn off the power before continuing operation.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Unit of data	0.01	0.001	0.0001	deg
Standard setting value	36000	360000	3600000	

[Valid data range] 1000 to 99999999

Set move amount per rotation of rotation axis.

Note

NOTE
This function cannot be used together with the indexing function of the index table (M series).

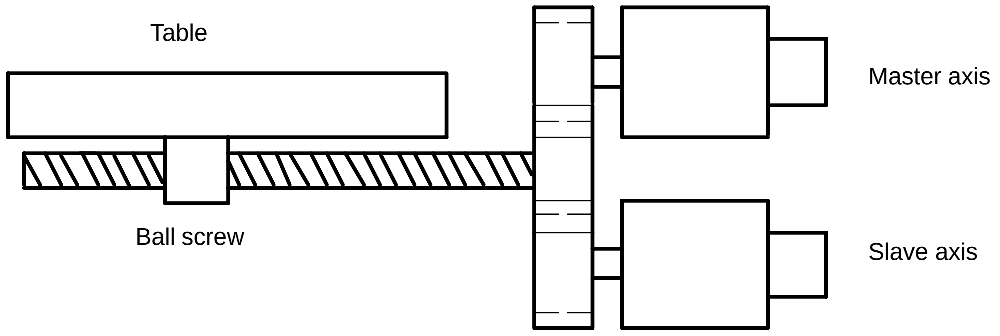
Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.20.2	ROTARY AXIS ROLL–OVER
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.19.2	ROTARY AXIS ROLL–OVER
Series 0*i* Mate–C	OPERATOR’S MANUAL (T series) (B–64134EN)	II.18.1	ROTARY AXIS ROLL–OVER

1.6 TANDEM CONTROL

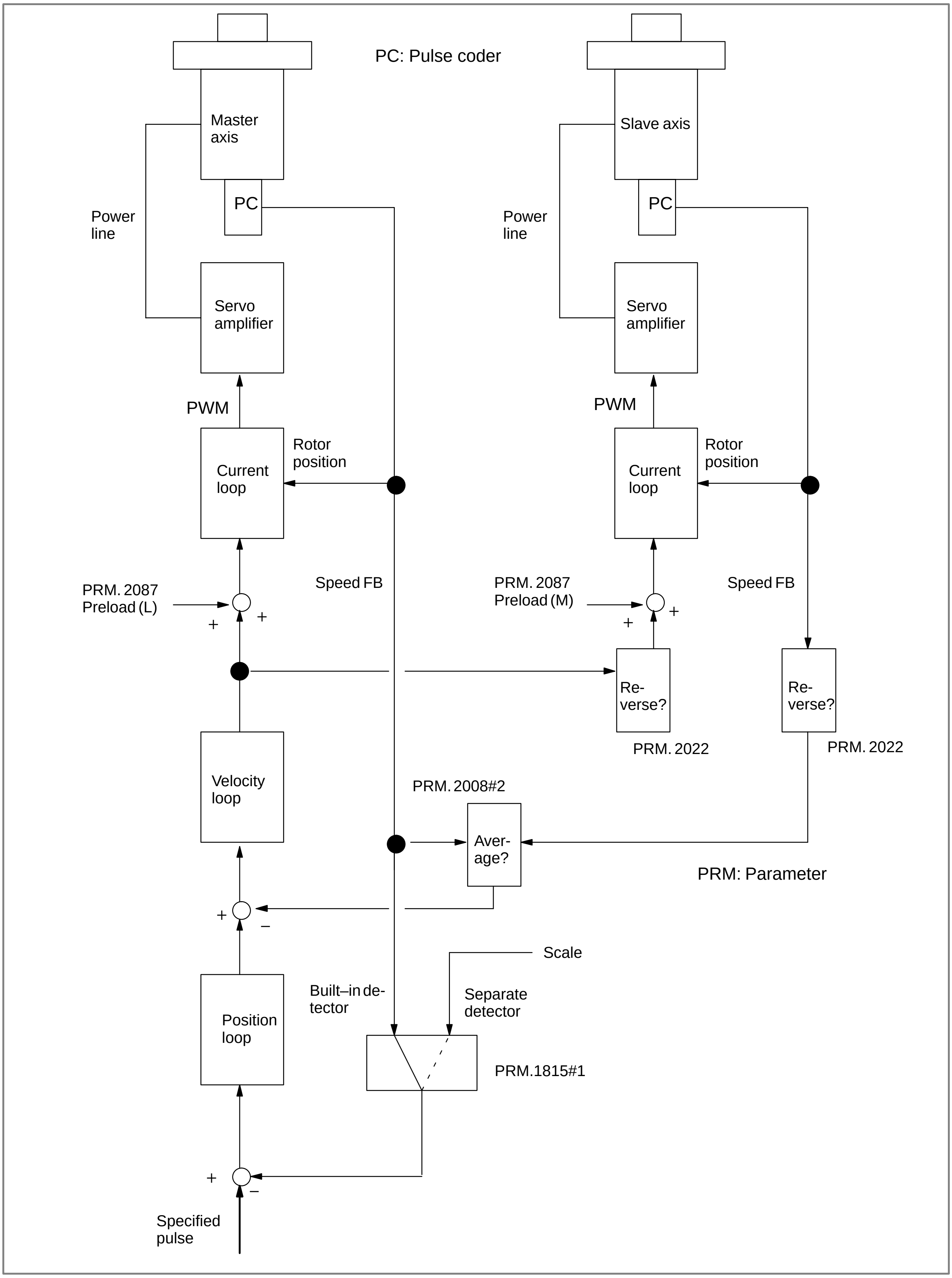
General

If a single motor cannot produce sufficient torque to move a large table, for example, this function allows two motors to be used. By means of this function, two motors can be used to perform movement along a single axis.
Positioning is carried out only for the master axis. The slave axis is used only to produce a torque. By means of this function, double the amount of torque can be obtained.



Sample application

The CNC generally processes the two axes of tandem control as a single axis. In the management of servo parameters and the monitoring of servo alarms, however, the two axes are handled individually.



Block Diagram of Tandem Control

Explanations

To specify the axis configuration in tandem control, follow the procedure below:

- **Axis configuration in tandem control**

(1) Tandem control can be performed for up to two pairs of axes.

(2) In terms of controlled axes, the pair of axes is handled as two separate axes. In terms of CNC–controlled axes (command axes), the pair of axes is handled as a single axis (master axis). Specify the number of CNC–controlled axes with parameter 1010, excluding the slave axis of tandem control. The slave axis must be handled as if it were controlled only by the PMC.

(3) The pair of axes is handled as two separate axes in the management of servo parameters and the monitoring of servo alarms.

(4) Assign two consecutive numbers, that is one odd and one even number, to the master and slave axes as their servo axis numbers (parameter 1023). Assign the smaller number to the master axis.

(Example) If the servo axis number of the master axis (parameter 1023) is set to 1, specify servo axis number 2 for the corresponding slave axis. If the servo axis number of the master axis is set to 3, specify servo axis number 4 for the corresponding slave axis.

(5) If tandem control is performed for two or more pairs of axes, assign servo axis numbers to the master and slave axes in identical order.

(6) Specify a unique axis name for the slave axis.

(7) The slave axis is handled as a controlled axis. Set the NDPx bit (bit 0 of parameter 3115) to 1 to suppress the position display.

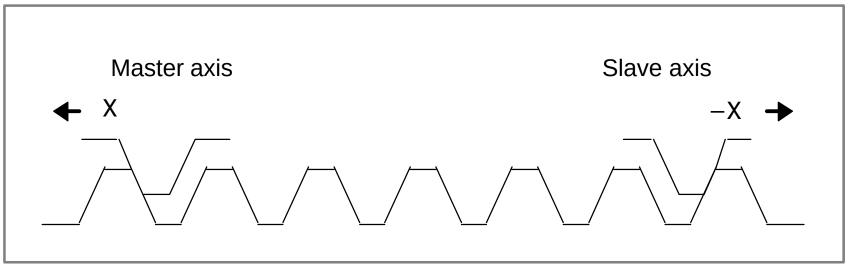
The following sample axis configuration is for a machine with four axes X, Y, Z, and A. The X–axis is the master axis of tandem control.

Number of CNC controlled axes (parameter 1010): 3

Axis number	Axis name	Servo axis number (PRM. 1023)	
1	X	3	CNC axis (master axis of tandem control)
2	Y	1	CNC axis
3	Z	2	CNC axis
4	A	4	Slave axis of tandem control (master axis: X–axis)

• Preload function

By adding an offset to the torque controlled by the position (velocity) feedback device, the function can apply opposite torques to the master and slave axes so that equal and opposite movements are performed for both axes. This function can reduce the effect of backlash on the master and slave axes caused by the tandem connection of the two motors via a gear. This function, however, cannot reduce backlash between the ball screw and table or other backlash inherent to the machine.
If a preload of x is set for the master axis and –x for the slave axis, the opposing preload torques are continuously applied to the two axes, even at rest, as shown below:



> **CAUTION**
> 1 Specify as low a preload as possible. Avoid specifying a preload higher than the rated torque. Too high a preload will trigger an overload alarm because the specified torques continue to be applied, even at rest. A preload that is only slightly higher than the frictional force is recommended. Thus, the recommended preload may be about one–third of the rated torque.
> 2 If the motors rotate in opposite directions (different signs are specified in parameter 2022), specify the preload values with the same sign.

• Velocity feedback average function

As shown in the block diagram of tandem control, the motor of the slave axis is not subject to velocity control. A machine with a large amount of backlash may become unstable if the motor of the slave axis vibrates as a result of backlash in the gear. This can be overcome by applying velocity control to the slave axis also. This velocity feedback average function is enabled when bit 2 of parameter 2008 is set to 1.

• Improved stability of a closed–loop system

The following two functions can increase the stability and position gain of a closed–loop system having a linear scale:

· Dual position feedback function
· Machine velocity feedback function

For details of these functions, refer to FANUC AC SERVO MOTOR α*is*/α*i*/β*is* series PARAMETER MANUAL (B–65270EN).

• Notes on stability of tandem control

An important factor affecting stability in tandem control is the capability of back feed. Back feed is to cause movement along either the master or slave axis from the other axis, via the transmission mechanism connecting the two axes. A machine without this capability may be inclined to become unstable and require adjustments.

- **Connection of axis signals**

The DI/DO signals, generally connected to each axis, must be connected only to the master axis of two axes of tandem control. The signals need not be connected to the slave axis. The following signals, however, may have to be connected depending on the application.

i) Controlled axis detach signal and servo off signal
Connect these signals so that the master and slave axis signals are simultaneously input.

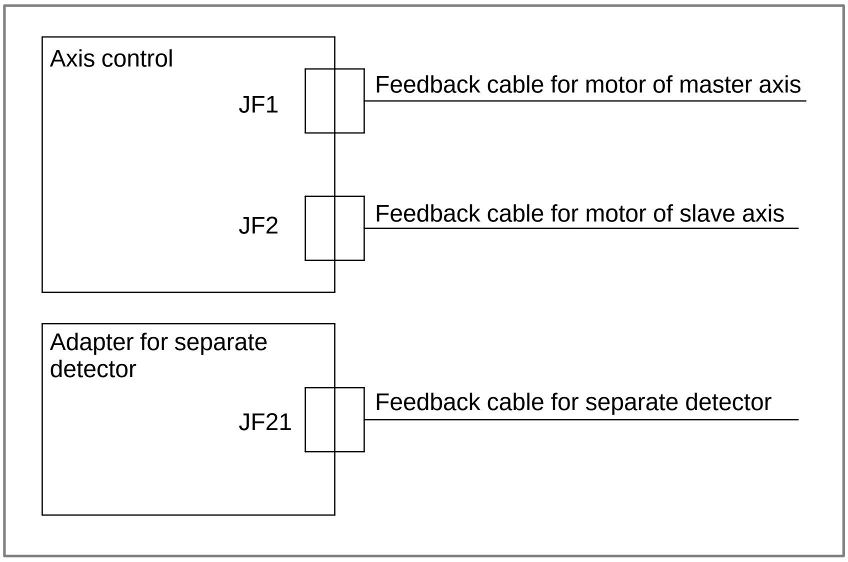
ii) Overtravel limit signal
Connect the signal so that 1 is always output as the overtravel limit signal for the slave axis.

If the slave axis stroke limit must also be detected, connect the signals so that the signal detected on the slave axis is sent to the overtravel limit signal of the master axis.

- **Connecting motors**

Connect the motors according to the servo axis numbers. Connect the feedback cable of the slave axis.

(Sample connection for position feedback cable)



- **Servo alarms**

Motor overload and other servo alarms are displayed separately for the master and slave axes.

Parameter

Setting data (parameters)

The parameters that are generally set for each axis can, when set for axes under tandem control, be classified into the following three groups:

i) Parameters in which identical values must be set for the master and slave axes

ii) Parameters that must be specified only for the master axis (The corresponding parameter for the slave axis is not used.)

iii) Parameters for which different values may be set for the master and slave axes

The classifications of the parameters are described below. Any parameter that is not listed in the tables for the three classifications should be processed as a parameter of type i) and, specify identical values for the master and slave axes.

WARNING

Note that, if different values are set for the master and slave axes in a parameter of type i), the operations for the two axes of tandem control will not be performed correctly.

– Care must be taken to specify the following two servo parameters, according to the directions of rotation around the master and slave axes.

  Parameter 2022: Direction of rotation of the motor

  Parameter 2087: Preload value

  In parameter 2022, specify 111 for forward rotation and –111 for the reverse rotation.

  In parameter 2087, specify values having identical signs when the motors of the master and slave axes rotate in opposite directions. Specify values having different signs when the motors of the master and slave axes rotate in the same direction.

– If a separate pulse coder is used, use of the separate pulse coder must be set for the master axis. For the slave axis, use of a built–in pulse coder must be set. Therefore, pay particular attention to setting the following parameters.

  Bit 1 of parameter 1815: Separate pulse coder

  Bits 6 to 4 of parameter 1816: Detection multiplier (DMR)

  Parameter 2024: Number of position detection feedback pulses (PPLS)

  Parameter 1821: Capacity of an optional reference counter

  Parameter 2084: Numerator of flexible feed gear ratio

  Parameter 2085: Denominator of flexible feed gear ratio

  If, for example, a motor with serial pulse coder A is used with a linear scale capable of detecting a position in 1–μm units, and if a single rotation of the motor produces a movement of 4 mm, specify the parameters as shown below:

		Master axis	Slave axis
No. 1815#1	=	1	0
No. 1816	=	01110000	01110000
No. 2024	=	4000	12500
No. 1821	=	4000	4000
No. 2084	=	0	4
No. 2085	=	0	1000

Parameters that should be set only for the master axis

Parameter No.	Meaning of parameters
0012#0	Mirror image
0012#7	Servo control off
1004#7	Input unit 10 times
1005#4	External deceleration in plus direction
1005#5	External deceleration in minus direction
1005#7	Servo control off
1022	Parallel axis specification
1220	External workpiece coordinate shift
1221	Workpiece zero point offset by G54
1222	Workpiece zero point offset by G55
1223	Workpiece zero point offset by G56
1224	Workpiece zero point offset by G57
1225	Workpiece zero point offset by G58
1226	Workpiece zero point offset by G59
1423	Jog feedrate
1424	Manual rapid traverse
1425	FL rate in manual reference position return
1427	External deceleration rate at rapid traverse
1430	Maximum feedrate
1815#1	Separate type pulse coder
1815#5	Absolute pulse coder
2008#2	Velocity feedback average function

Parameters that may be set to different values for the master and slave axes

Parameter No.	Meaning of parameters
1020	Axis name
1023	Servo axis number
2022	Motor rotation direction
2087	Preload value
3115	Current position display
1310#0	Soft OT2
1310#1	Soft OT3
1320	1st stroke limit of plus side
1321	1st stroke limit of minus side
1322	2nd stroke limit of plus side
1323	2nd stroke limit of minus side
1815#1	Separate type pulse coder
1816#6 to #4	Detection multiplier (DMR)
1821	Arbitrary reference counter capacity
2024	Position detection feedback pulses (PPLS)
2084	Numerator of flexible feed gear ratio
2085	Denominator of flexible feed gear ratio

Parameters that should be set to the same values for the master and slave axes

Parameter No.	Meaning of parameters
1005#0	Movement before reference position return
1005#1	Dogless reference position setting
1006#0	Rotary axis
1006#1	Machine coordinate of rotary axis is rotary type
1006#3	Diameter/radius specification
1006#5	Direction of reference position return
1006#7	Least input increment (0.0001 mm)
1240	Reference position as viewed from machine zero
1241	Coordinate of 2nd reference position
1242	Coordinate of 3rd reference position
1243	Coordinate of 4th reference position
1260	Move distance per rotation of rotary axis
1420	Rapid traverse rate
1421	F0 of rapid traverse override
1620	Time constant of rapid traverse linear acceleration/deceleration
1621	Time constant of rapid traverse bell shaped acceleration/deceleration
1622	Time constant of feed exponential acceleration/deceleration
1623	FL of feed exponential acceleration/deceleration
1624	Time constant of manual continuous exponential acceleration/deceleration
1625	FL of manual continuous exponential acceleration/deceleration
1626	Time constant of exponential acceleration/deceleration during thread cutting cycle
1627	FL of exponential acceleration/deceleration during thread cutting cycle
1820	Command multiplier (CMR)
18XX	Digital servo parameters
20XX	Digital servo parameters

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

[Data type] Bit axis (set to each axis)
Set for both master and slave axes.

TAN Tandem control is
0 : not performed
1 : performed

	#7	#6	#5	#4	#3	#2	#1	#0
2008						VFBAVE		

[Data type] Bit axis (set to each axis)
Set only for the master axes.

VFBAVE Velocity feedback average function
0 : invalid
1 : valid

2087	Preload of each axis (Tcmd offset)

[Data type] Word axis

[Unit of data] (Preamplifier limit) /7282

[Valid data range] –1821 to 1821

An offset is added to a torque command to reduce backlash.
Set a slightly large value than that of the friction torque of the motor.
As a reference set a value one–third the rated torque.

[Example] To set a torque of 3A in the opposing directions for the master and slave amplifiers that have current limit of 40A:
3 / (40 / 7282) = 546
Master side = 546
Slave side = –546

2021	Load inertia

[Data type] Word axis

Set the same value to the master and slave axes.

[Unit of data] $\dfrac{\text{(All load inertia)}}{2}$ /(Motor inertia) $\times$ 256

2022	Direction of rotation of motor

[Data type] Word axis

Set the direction of motor rotation.
If the rotation directions of master and slave axes are opposite, set them by this parameter.

Alarm and message

Number	Message	Description
417	SERVO ALARM: n AXIS DGTL PARAM	Illegal values are set for parameter 1010, 1023, or 1817 when tandem control is performed.

1.7 SIMPLE SYNCHRONOUS CONTROL

General

A movement along an axis can be executed simply by executing a move command specified for that axis or by synchronizing the movement with another axis. Either of these two types can be selected by means of a signal sent from the machine.

In synchronous operation, that axis for which move commands can be specified is called the master axis, while an axis along which the tool moves in sync with the master axis is called a slave axis.

• Simple synchronous control for the M series and T series

The M series and T series support different simple synchronization control functions. One of the greatest differences is that:

<T series> The function can synchronize only automatic operations. It cannot synchronize manual operations. Only one master/slave axis pair can be used.

<M series> The function can synchronize both automatic and manual operations. A maximum of two pairs can be synchronized.

The following functions are provided only for the M series:

- Synchronization error check function
- Synchronization compensation function

• Synchronization error check based on positional deviation (M series)

Any difference between the servo positional deviation of the master axis and that of the slave axis is monitored constantly. A P/S alarm condition (No. 213) is detected if a limit set in parameter No. 8313 (if only one master/slave axis pair is in sync) or 8323 (if two master/slave axis pair is in sync) is exceeded.

• Synchronization error check based on machine coordinates (M series)

The function monitors the difference between the machine coordinates on the master and slave axes. If the function detects a difference greater than or equal to a preset value, it stops the machine. This function constantly monitors the difference. Even if the synchronization control signal is erroneously set to 0, thus disabling synchronization control, the function can issue an alarm, stop the machine, and thus prevent damage.

If the detected difference is greater than or equal to the maximum error set in parameter 8314, servo alarm 407 is output.

WARNING
1 Before using the synchronization error check function, set identical values for the reference positions of the master and slave axes.
2 To clear the alarm, first increase the maximum synchronization error set in parameter 8314, then press the reset key. Next, perform handle operations or other manual operations so that the machine coordinates agree. Then, restore the original value in parameter 8314.
3 If an alarm is detected during a synchronous operation, set the signals indicating that a synchronous operation is in progress (G138, G140) to off, then follow the procedure for clearing an alarm.

NOTE
If the synchronization error check function is not used, set parameter 8314 to 0.

- **Synchronization compensation function (M series)**

If the synchronization between the positions of the master and slave axes is lost when the system power is turned off, the function compensates for the difference between them. After performing a follow–up at power on, the function sends compensation pulses to the slave axis to adjust its position such that it agrees with that of the master axis. This function is enabled only when the slave axis of synchronization control supports the absolute–position detection function.

This function, however, cannot be used for rotation axes.

WARNING
1 The synchronization compensation function is enabled after reference position returns have been performed. The function is not executed if the parameter is set before reference position returns are performed.
2 The synchronization compensation function is not executed when the servo alarm is eliminated.

CAUTION
The synchronization deviation is processed as a position error on the slave axis while at rest. The position error is displayed as diagnostic data 300, in the same units as used to detect the error. If the error exceeds the value set in parameter 8315 (if only one master/slave axis pair is in sync) or 8325 (if two master/slave axis pair is in sync), servo alarm 410 is triggered. The alarm can be cleared by pressing the reset key. As the position error for the slave axis remains even after the alarm is cleared, however, the positions must be adjusted.

> **NOTE**
> 1 To use the synchronization compensation function, set the SOF bit, bit 7 of parameter 8301 (if only one master/slave axis pair is in sync) or SOFx bit, bit 7 of parameter 8303 (if two master/slave axis pair is in sync), to 1.
> 2 The synchronization compensation function is also enabled when emergency stop is canceled.

- **Automatic setting of grid positioning (M series)**

To use simple synchronous control, it is necessary to perform reference position return for the master and slave axes. This function causes the CNC to automatically perform reference position return (grid position) for both the master and slave axes in simple synchronization.

[Operating procedure]

This procedure can be applied only when one master/slave axis pair is in sync, and when bit 0 (ATE) of parameter No. 8302 is set to 1. If two master/slave axis pair is in sync, it is necessary to use parameters ATEx (bit 0 of parameter No. 8303) and ATSx (bit 1 of parameter No. 8303).

1 Set bit 1 (ATS) of parameter No. 8302 to 1.

2 Power off/on.

3 Enter REF mode (or JOG mode for reference position setting without dogs), and move along the axis toward the reference position.

4 Motion along the master and slave axes stops automatically, and the grid deviation is set in parameter No. 8316. At the same time, bit 1 (ATS) of parameter No. 8302 becomes 0, and a power–off request alarm (No. 000) occurs.

5 Switch the power off then back on.

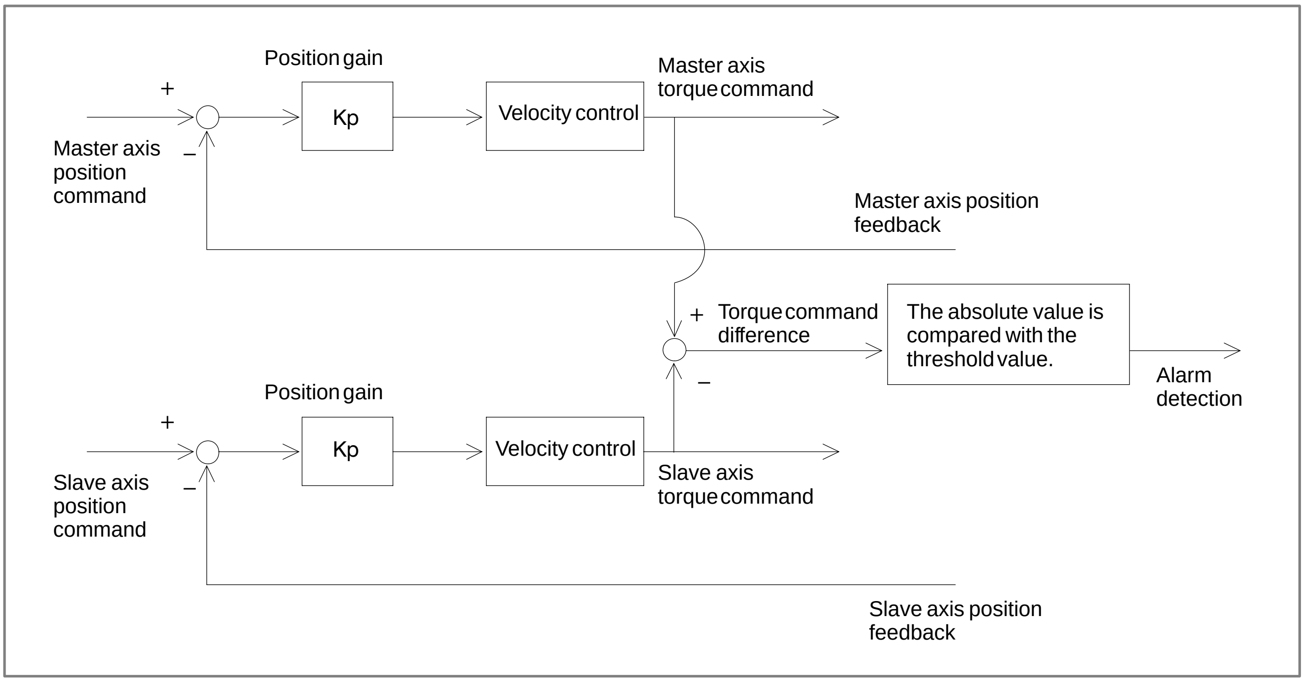
6 Perform ordinary reference position return.

> **NOTE**
> Parameter setting
> When parameter ATS (bit 1 of parameter No. 8302) or ATSx (bit 1 of parameter No. 8303) is set, parameter APZ (bit 4 of parameter No. 1815) for the master and slave axes and parameter No. 8316 become 0. If the operator specifies parameter No. 8316 (MDI, G10L50), parameter ATE (bit 0 of parameter No. 8302) becomes 0.

- **Torque difference alarm detection (M series)**

If the master and slave axes operate independently while simple synchronous control is applied, the machine may be damaged. To prevent this, the torque command difference between the axes is monitored. If the difference is found to be abnormal, an alarm can be issued.

[System configuration]



[How to use]

Determine the threshold parameter using the following procedure.

1 Set up the following parameters:
Parameter No. 2031 = 0 : Disable torque difference alarm detection.

Parameter Nos. 2115 and 2151: Display the absolute value of a torque difference between axes in synchronization on the diagnosis screen. Set the same value for the two axes in simplified synchronization.
Setting
Parameter No. 2115 = 0
Parameter No. 2151 = 178

2 Cause the diagnosis screen to appear, using the <SYSTEM> function key → the [DGNOS] soft key.
The diagnosis screen No. 353 displays the absolute value of a torque difference between the two axes in synchronization.

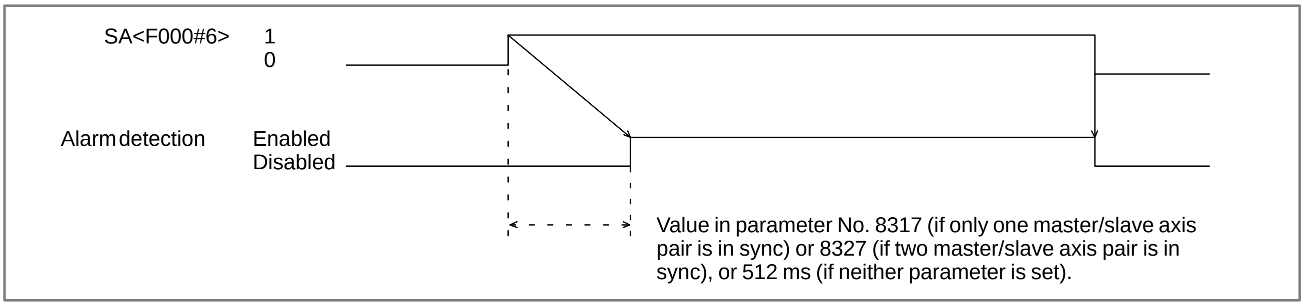
3 Read a maximum of the absolute values of torque differences during a normal operation.
Set the threshold parameter with the maximum absolute value with some margin allowed.
If it is difficult to read the absolute values of torque differences on the diagnosis screen, observe the absolute values of torque differences with an oscilloscope, using the following method:

4 Set parameter Nos. 2115 and 2151 with:
Setting
Parameter No. 2115 = 4
Parameter No. 2151 = 178

5 Connect a check board to observe torque differences.
If an analog check board is used, set the rotary switch of the check board to 1, and observe signals on CH7.
If the oscilloscope is a combined analog/digital model, set the DATA digit for CH1 to 5, and observe signals on CH1.

6 Convert the observed value, using the formula 1 V = 410 (specified threshold value). Read the maximum value during ordinary operation, and allow an appropriate margin.

[Timing chart]



If the servo preparation completed signal SA <F000#6> is 0, torque difference alarm detection is not performed.

The simple synchronous control functions are described separately for the T series and M series in the following explanations.

Signal

<T series and M series>

Signals to select the slave axis for simple synchronous control SYNC1 to SYNC4 <G138#0–#3>

[Classification] Input signal

[Function] synchronous control is performed for memory or MDI operation.
The signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNC 1

1. ... The first axis becomes the slave axis for synchronous control.
2. ... The second axis becomes the slave axis for synchronous control.
3. ... The third axis becomes the slave axis for synchronous control.

:
:

[Operation] When the signal is set to 1, the control unit operates as described below:

– During memory or MDI operation, the control unit supplies the move command, specified for the master axis, to both the master and slave axes of synchronous control.

The master axis is specified with a parameter.

<M series>

Signals for selecting the manual feed axis for simple synchronous control SYNCJ1 to SYNCJ4 <G140#0–#3>

[Classification] Input signal

[Function] synchronous control is performed in jog, handle, or incremental feed mode.
The signal is provided for each controlled axis. The number at the end of the signal name represents the number of the controlled axis.

SYNCJ 1

1. ... The first axis becomes the slave axis for synchronous control.
2. ... The second axis becomes the slave axis for synchronous control.
3. ... The third axis becomes the slave axis for synchronous control.
:
:

[Operation] When the signal is set to 1, the control unit operates as described below:

– In jog, handle, or incremental feed mode, the control unit supplies the move command, specified for the master axis, to both the master and slave axes of synchronous control.

The master axis is specified with a parameter.

Signal address

T series

	#7	#6	#5	#4	#3	#2	#1	#0
G138					SYNC4	SYNC3	SYNC2	SYNC1

M series

	#7	#6	#5	#4	#3	#2	#1	#0
G138					SYNC4	SYNC3	SYNC2	SYNC1
G140					SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1

Parameter

T series

1010	Number of CNC–controlled axes

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

[Data type] Byte

[Valid data range] 1, 2, 3, ..., the number of controlled axes

Set the maximum number of axes that can be controlled by the CNC.

[Example]
Suppose that the first axis is the X axis, and the second and subsequent axes are the Y, Z, and A axes in that order, and that they are controlled as follows:

X and Y axes: Controlled by the CNC

Z axis: Controlled by the CNC and PMC
A axis: Controlled by the PMC

Then set this parameter to 3 (total 3: first to third axes)

NOTE
When using simplified synchronization control, specify slave axes as well as the master axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1015			SVS					

[Data type] Bit

SVS When the servo along an axis is turned off, simple synchronous control is:

0 : Released.
1 : Not released.

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 to 3

Select a master axis for simple synchronous control. Set a master axis number for the axis used as a slave axis. If the value of this parameter is 0, the first axis is the master axis. In this case, when the synchronous control select signal G138 is set to 1, operation starts with the 1st axis being the master axis.

Units digit in the parameter for the first axis
→ Set the axis number for the master axis when the first axis is used as a slave axis.

Tens digit in the parameter for the first axis
→ Set the axis number for the master axis when the second axis is used as a slave axis.

Units digit in the parameter for the second axis
→ Set the axis number for the master axis when the third axis is used as a slave axis.

Tens digit in the parameter for the second axis
→ Set the axis number for the master axis when the fourth axis is used as a slave axis.

Number	Tens digit	Units digit
First Second	Second axis Fourth axis	First axis Third axis

NOTE
The axis number settings are: 0 for the first axis, 1 for the second axis, 2 for the third axis, and so on.

Example) To set the 3rd axis as the master axis and the 4th axis to the slave axis, set as follows:

No. 8311
1st axis 00
2nd axis 20
3rd axis 00
4th axis 00

8312	Enabling/disablingmirror image in synchronous control

[Data type] Byte axis

[Valid data range] –127 to +128

This parameter sets the mirror image function. When 100 or a greater value is set with this parameter, the mirror image function is applied to synchronous control. Set this parameter to the slave axis.

Example: To establish reversed synchronization when using the third axis as the master axis and the fourth axis as the slave axis, set parameter No. 8312 as follows:

Parameter No. 8312 (first axis) = 0
Parameter No. 8312 (second axis) = 0
Parameter No. 8312 (third axis) = 0
Parameter No. 8312 (fourth axis) = 100

M series

1010	Number of CNC–controlled axes

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

[Data type] Byte

[Valid data range] 1, 2, 3, ..., the number of controlled axes

Set the maximum number of axes that can be controlled by the CNC.

[Example]
Suppose that the first axis is the X axis, and the second and subsequent axes are the Y, Z, and A axes in that order, and that they are controlled as follows:

X and Y axes: Controlled by the CNC
Z axis: Controlled by the CNC and PMC
A axis: Controlled by the PMC
Then set this parameter to 3 (total 3: first to third axes)

NOTE
When using simplified synchronization control, specify slave axes as well as the master axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3105	SMF							

[Data type] Bit

SMF During simple synchronous contorol, movement along a slave axis is:
0 : Included in the actual speed display
1 : Not included in the actual speed display

NOTE
This parameter is valid when simple syncronous contorol is applied according to the setting of parameter No. 8311 (master and slave axes can be arbitrarily selected).

	#7	#6	#5	#4	#3	#2	#1	#0
8301	SOF							

[Data type] Bit

SOF The synchronization compensation funciton under simple synchronous control (one master/slave axis pair) is:
0 : Not used.
1 : Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8302							ATS	ATE

NOTE
The system power must be turned off then back on in order for this parameter setting to become effective.

[Data type] Bit

ATE Specify whether to enable the automatic setting of grid positioning under simple synchronous control (one master/slave axis pair)

0 : Disabled
1 : Enabled

ATS Specify whether to start the automatic setting of grid positioning under simple synchronous control (one master/slave axis pair)

0 : Not started
1 : Started

NOTE
1 Setting this parameter resets parameter APZx (bit 4 of parameter No. 1815) for the master and slave axes and parameter No. 8316 to 0.
2 This parameter automatically becomes 0 upon the completion of grid positioning.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOFx						ATSx	ATEx

[Data type] Bit axis

ATEx Specify whether to enable the automatic setting of grid positioning under simple synchronous control (two master/slave axis pair)

0 : Disabled
1 : Enabled

ASTx Specify whether to start the automatic setting of grid positioning under simple synchronous control (two master/slave axis pair)

0 : Not started
1 : Started

NOTE
To start the automatic setting of grid positioning, set ATSx to 1. ATSx automatically becomes 0 upon the completion of automatic setting.

SOFx Specify whether to enable synchronization compensation during simple synchronous control (two master/slave axis pair)

0 : Disabled
1 : Enabled

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 to 3

Select a master axis and slave axis for simple synchronous control. Set a master axis number for the slave axis side. A maximum of two pairs can be synchronized.

Example1: When using the first axis (X–axis) as the master axis, and the third axis (Z–axis) as the slave axis, set parameter No. 8311 as follows:

Parameter No. 8311 X (first axis) = 0
Parameter No. 8311 Y (second axis) = 0
Parameter No. 8311 Z (third axis) = 1
Parameter No. 8311 A (fourth axis) = 0

Example2: If there are two master/slave axis pairs under simple synchronous control:

To specify the:
master axis as the 1st axis, and the slave axis as the 4th axis
master axis as the 2nd axis, and the slave axis as the 3rd axis,
set the following:
Parameter No. 8311 X(1st axis) = 0
Y(2nd axis) = 0
Z (3rd axis) = 2
(4th axis) = 1

Specifying the third axis (Z–axis) as the master axis, and the first axis (X–axis) as the slave axis is not allowed. The master axis number must always be smaller than the slave axis number.
It is impossible to specify more than one slave axis for a master axis.

NOTE
In Example 2 above, the Z axis, which is the master axis of the M Series, is used as the slave axis of another axis. This usage may prevent other functions from operating normally. When exercising simple synchronous control with two pairs, take care.

8313	Limit of the difference between the amount of positioning deviation of the master and slave axes (one master/slave axis pair under synchronous control)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies a limit imposed on the positional deviation difference between the master and slave axes. If the limit is exceeded, a P/S alarm (No. 213) is issued.

8314	Allowable error in synchronization error check

[Data type] Word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 32767

This parameter sets, in the detection unit, the allowable error when a synchronization error check is made. The mechanical coordinates of the master axis and slave axis are monitored. When a synchronization error equal to or greater than the value set in this parameter is detected, servo alarm No. 407 is issued, and the machine is stopped. Set this parameter to the master axis. When 0 is set with this parameter, no synchronization error check is performed.

8315	Maximum compensation value for synchronization compensation

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the maximum compensation value for synchronization. When a compensation value greater than the value set in this parameter is used, servo alarm No. 410 is issued.

8316	Reference counter difference between the master and slave axes (one master/slave axis pair under synchronous control)

NOTE
The system power must be turned off then back on in order for this setting to become effective.

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

This parameter is set to the reference counter difference between the master and slave axes.

NOTE
Upon the completion of grid positioning, the reference counter difference is set automatically. At the same time, parameter ATS (bit 1 of parameter 8302) is reset to 0.

8317	Torque difference alarm detection timer (one master/slave axis pair under synchronous control)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000 (if 0 is specified, the system assumes 512 ms.)

This parameter specifies the time between the servo preparation completed signal SA <F000#6> becoming 1 and the torque difference alarm detection function starting to check for a torque difference alarm condition. The specified value is rounded up to the nearest multiple of 16 ms.

(Example) If 100 is specified, 112 ms is assumed.

8323	Limit imposed on the positional deviation difference between the master and slave axes (two master/slave axis pair under synchronous control)

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies a limit imposed on the positional deviation difference between the compensation and slave axes. If the limit is exceeded, a P/S alarm (No. 213) is issued. This parameter must be set for the master axis. If 0 is specified, a positional deviation difference check is not performed.

8325	Maximum compensation to be applied during synchronization matching (two master/slave axis pair under synchronous control)

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies the maximum compensation to be applied during synchronous compensation. If the compensation exceeds the set value, a servo alarm (No. 410) is issued. This parameter must be set for the master axis. For this setting to become effective, parameter SOFx (bit 7 of parameter No. 8303) must be set to 1.

8326	Reference counter difference between the master and slave axes (two master/slave axis pair under synchronous control)

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

This parameter is automatically set to the reference counter difference (grid deviation) between the master and slave axes, when automatic grid position setting is performed. This parameter setting, together with an ordinary grid shift, is transferred to the servo section, when the user subsequently switches the power off then back on. This parameter must be set for the master axis.

8327	Torque difference alarm detection timer (two pair under synchronous control)

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

This parameter specifies the time between the servo preparation completed signal SA <F000#6> becoming 1 and the torque difference alarm detection function starting to check for a torque difference alarm condition during simple synchronous control. The specified value is rounded up to the nearest multiple of 16 ms.

(Example) If 100 is specified, 112 ms is assumed.

This parameter must be specified for the master axis. If the set value is 0, then 512 ms is assumed.

Alarm and message

T series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	A move command was specified for the slave axis of synchronous control.
214	ILLEGAL COMMAND IN SYNCHRO–MODE	A command for coordinate system setting or shift–type tool compensation was executed during synchronous control. Correct the program.

M series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	One of the following errors occurred during simple synchronous control operation: (1) The program contains a move command for the slave axis. (2) A command for jog feed, manual handle feed, or incremental feed was issued for the slave axis. (3) After power on, the command for automatic reference position return was specified before a manual reference position return had been performed. (4) The difference in position error between the master and slave axes exceeded the value set in parameter 8313 or 8323. Or, switching between synchronous mode and asynchronous mode was performed during automatic operation (when the automatic operation signal (OP) is 1).

Servo alarm

Number	Message	Description
407	SERVO ALARM: EXCESS ERROR	The following error has occurred during an operation under simplified synchronization control: The difference in machine coordinate value between axes in synchronization has exceeded a value specified in parameter No. 8314.
410	SERVO ALARM: n AXIS EXCESS ERR	The most likely causes are: 1 For the n axis, the positional deviation observed when the axis is stopped has exceeded the value set in parameter No. 1829. 2 In simple synchronous control, the compensation used during synchronous compensation has exceeded the value set in parameter No. 8315 or 8325. This alarm occurs only for the slave axis.
420	SERVO ALARM: n AXIS SYNC TORQUE	A torque command issued for the master or slave axis is greater than the value specified in parameter No. 2031. This alarm condition occurs only for the master axis.

Diagnostic data

Number	Message	Description
540	SYNCHRO ERROR	The data represents the difference in position error between the master and slave axes during synchronous control. (One master/slave axis pair under synchronous control)
541	SYNCHRO ERROR	The positional difference between the master and slave axes under synchronous control is displayed. (Two master/slave axis pair under synchronous control)

Caution

CAUTION

1 Set the same detection unit for both the master and slave axes.
2 When a manual reference position return is executed, identical movements are performed along the master and slave axes until deceleration commences. Subsequently, grids are detected separately.
3 Pitch error compensation and backlash compensation are executed separately for the master and slave axes.
4 If control of two master/slave axis pair is specified in parameter No. 8311, parameters to perform setting for, and display diagnostic information about, only one master/slave axis pair are invalid, and if control of only one master/slave axis pair is specified, parameters to perform setting for, and display diagnostic information about, only one master/slave axis pair are valid (except for common parameter Nos. 8311 and 8314).

Reference item

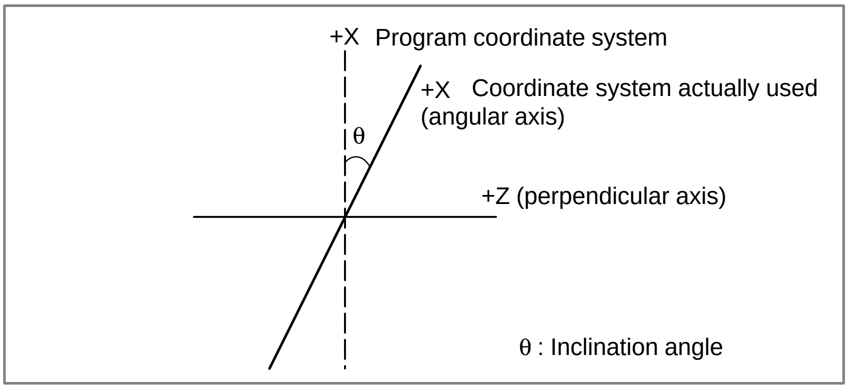
Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.20.1	Simple synchronous control
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.19.3	Simple synchronous control

1.8 ANGULAR AXIS CONTROL

1.8.1 Angular Axis Control/Arbitrary Angular Axis Control

General

When the angular axis makes an angle other than 90° with the perpendicular axis, the angular axis control function controls the distance traveled along each axis according to the inclination angle. For the ordinary angular axis control function, the X–axis is always used as the angular axis and the Z–axis is always used as the perpendicular axis. For angular axis control B, however, arbitrary axes can be specified as the angular and perpendicular axes, by specifying parameters accordingly. A program, when created, assumes that the angular axis and perpendicular axis intersect at right angles. However, the actual distance traveled is controlled according to an inclination angle.



Explanations

When the angular axis is the X–axis and the perpendicular axis is the Z–axis, the amount of travel along each axis is controlled according to the formulas shown below.
The distance to be traveled along the X–axis is determined by the following formula :

$$Xa = \frac{Xp}{\cos\theta}$$

The distance traveled along the Z–axis is corrected by the inclination of the X–axis, and is determined by the following formula:

$$Za = Zp – Xp\tan\theta$$

The speed component along the X–axis of feed rate is determined by the following formula:

$$Fa = \frac{Fp}{\cos\theta}$$

Xa, Za, Fa: Actual distance and speed
Xp, Zp, Fp: Programmed distance and speed

- **Method of use**

The angular and perpendicular axes to which angular axis control is to be applied must be specified beforehand, using parameters (No. 8211 and 8212).
Parameter AAC (No. 8200#0) enables or disables the angular axis control function. If the function is enabled, the distance traveled along each axis is controlled according to an inclination angle (No. 8210).
Parameter AZR (No. 8200#2) enables angular axis manual reference point return only with a distance along the angular axis.
If perpendicular/angular axis control disable signal NOZAGC has been set to 1, the angular axis control function is enabled only for the angular axis. In that case, the move command for the angular axis is converted to angular coordinates. The perpendicular axis is not affected by the move command for the angular axis.

- **Absolute and relative position display**

An absolute and a relative position are indicated in the programmed Cartesian coordinate system.

- **Machine position display**

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle. However, when inch/metric conversion is performed, a position is indicated which incorporates inch/metric conversion applied to the results of the inclination angle operation.

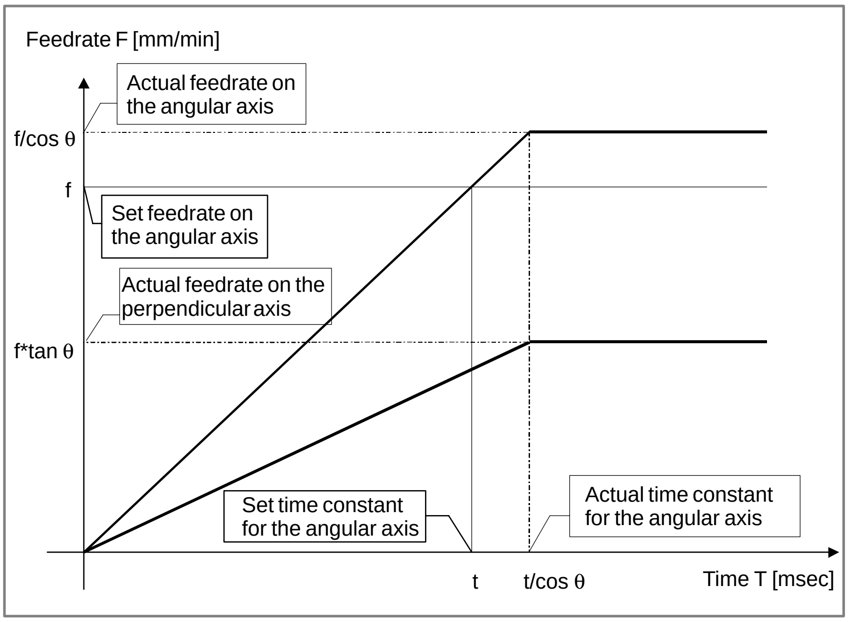
- **Path on an angular axis and perpendicular axis**

When a manual rapid traverse operation or a reference position return operation without dogs is performed on an angular axis during angular axis control, the path on the angular axis can be linear.
Control is exercised so that the acceleration/deceleration time is the same between the angular axis and perpendicular axis.
Suppose the following:

Rapid traverse rate on the angular axis (parameter No. 1420): f
Time constant for rapid traverse on the angular axis
(parameter No. 1620): t
Angle (parameter No. 8210): θ
Then, the actual rapid traverse rate on the perpendicular axis and the actual acceleration/deceleration time on the angular axis are as follows:

Actual rapid traverse rate on the perpendicular axis = $f*\tan\theta$
Actual acceleration/deceleration time on the angular axis = $t/\cos\theta$

At the time of rapid traverse on the perpendicular axis, acceleration/deceleration is performed according to the acceleration rate found from these two values.



Signal

Perpendicular/angular axis control disable signal NOZAGC <G063#5>

[Classification] Input signal

[Function] Disables angular axis control for the perpendicular axis.

[Operation] When this signal is set to 1, the control unit behaves as follows:

- Converts an angular axis move command to angular coordinates. The perpendicular axis is, however, not affected by an angular axis move command.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G063			NOZAGC					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8200						AZR		AAC

[Data type] Bit

AAC 0 : Does not perform angular axis control.
1 : Performs angular axis control.

AZR 0 : The machine tool is moved along the Cartesian axis during manual reference position return along the angular axis under angular axis control.

1 : The machine tool peforms manual reference position return along the angular axis under angular axis control.

	#7	#6	#5	#4	#3	#2	#1	#0
8201				ALN				

[Data type] Bit

ALN When manual rapid traverse or reference position return without dogs is performed for an angular axis during angular axis control:

0 : The acceleration/deceleration time for a Cartesian axis is not controlled.

1 : The acceleration/deceleration time for a Cartesian axis is controlled so that it matches the acceleration/deceleration time for the angular axis.

8210	Inclination angle for angular axis control

[Data type] 2 words

[Unit of data] 0.001 degree

[Valid data range] 20000 to 60000

8211	Axis number of a slanted axis subject to angular axis control
8212	Axis number of a Cartesian axis subject to slanted axis control

[Data type] Byte

[Unit of data] Axis number

[Valid data range] 1 to number of controlled axes

When angular axis control is to be applied to an arbitrary axis, these parameters set the axis numbers of a slanted axis and Cartesian axis.

Warning

WARNING

1 After angular axis control parameter setting, be sure to perform manual reference point return operation.
2 If bit 2 (AZK) of parameter No. 8200 has been set to 0, such that manual reference position return along the angular axis also causes movement along the perpendicular axis, once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis.
3 Once the tool has been moved along the angular axis with perpendicular/angular axis control disable signal NOZAGC set to 1, manual reference position return must be performed.
4 Before attempting to manually move the tool along both the angular and perpendicular axes simultaneously, set perpendicular/angular axis control disable signal NOZAGC to 1.

Note

NOTE

1 If an inclination angle close to 0° or ±90° is set, an error can occur. A range from ±20° to ±60° should be used.
2 Before a perpendicular axis reference point return check (G27) can be made, angular axis reference point return operation must be completed.
3 Before a perpendicular axis reference point return check (G27) can be made, angular axis reference point return operation must be completed.

Notes

1 To make the path linear on the angular axis and perpendicular axis, there must be a match in acceleration/deceleration type, time constant, and gain.

2 To make the path linear in jog feed as well, use linear acceleration/deceleration after interpolation. (Set bit 0 (CTL) and bit 4 (JGL) of parameter No. 1610 to 1.)

3 To make the path linear in automatic operation as well:

Cutting feed: Use linear acceleration/deceleration after interpolation.
(Set bit 0 (CTL) of parameter No. 1610 to 1.)

Rapid traverse: Use linear position of constant time type.
(Set bit 1 (LRP) of parameter No. 1401 to 1, and set bit 4 of parameter No. 1603 to 1.)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.20.4	Angular axis control
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.19.5	Angular axis control

1.8.2 Stored Stroke Limits in a Cartesian Coordinate System

General

This function is used to set stored stroke limits under angular axis control not in an angular coordinate system but in a Cartesian coordinate system.

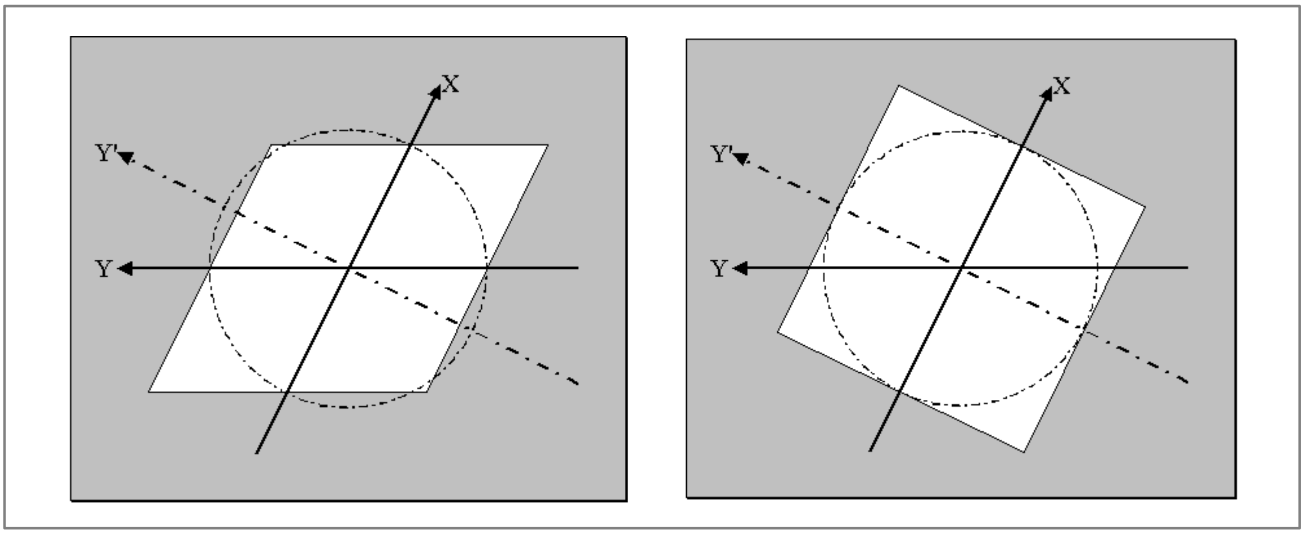


Fig. 1.8.2 (a) OT area in an angular coordinate system **Fig. 1.8.2 (b) OT area in a Cartesian coordinate system**

Explanation

When the move command Yp is specified for the angular axis under angular axis control, the actual travel distance Ya is:
$Ya = Yp/\cos\theta$
At this time, the compensation value Xa is output onto the perpendicular axis. The compensation value Xa is:
$Xa = -Yp \cdot \tan\theta$

The machine coordinates include the value converted for the angular axis and the compensation value for the perpendicular axis mentioned above. So, the machine coordinate system becomes an inclined coordinate system. Stored stroke limits are checked using a machine coordinate, so that the limit area also becomes an inclined area. In this case, it is not easy to recognize the area intuitively. So, a stroke check is made not in an actual angular machine coordinate system but in a virtual Cartesian machine coordinate system.

Diagnosis screen

306	Machine coordinate on the angular axis in a Cartesian coordinate system
307	Machine coordinate on the perpendicular axis in a Cartesian coordinate system

[Data type] 2–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

When angular axis control is used, machine coordinates in a Cartesian coordinate system are indicated.
The order of display can be changed using bit 7 (ADG) of parameter No. 8201.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8201	ADG	A53				AO3	AO2	AOT

[Data type] Bit

AOT Stored stroke limit 1 under angular axis control is:

0 : Handled as a value in an angular coordinate system
1 : Handled as a value in a Cartesian coordinate system

AO2 Stored stroke limit 2 under angular axis control is:

0 : Handled as a value in an angular coordinate system
1 : Handled as a value in a Cartesian coordinate system

AO3 Stored stroke limit 3 under angular axis control is:

0 : Handled as a value in an angular coordinate system
1 : Handled as a value in a Cartesian coordinate system

A53 When the machine coordinate system command (G53) specifies an angular axis singly under angular axis control:

0 : A movement is made on the perpendicular axis as well.
1 : A movement is made on the angular axis only.

ADG The diagnostic data of No. 306 and No. 307 is:

0 : Not exchanged with each other. The data is indicated in the order of the angular axis then the perpendicular axis.
1 : Exchanged with each other. The data is indicated in the order of the perpendicular axis then the angular axis.

Functions operating in a Cartesian machine coordinate system

- Stored stroke limit 1 (both I and II)
- Stored stroke limit 2 (G22/G23)
- Stored stroke limit 3
- Stroke check before movement(*1)
- External setting of stroke limits (available with the M series only. OT1 only is valid.)

  These functions are usable also with the function for issuing an alarm before an OT area is exceeded (bit 7 (BFA) of parameter No. 1300) and the OT deceleration function during linear acceleration/deceleration before cutting feed interpolation (parameter No. 1784).
  OT1/OT3 only is valid with the former, and OT1 only is valid with the latter.

The functions other than the above operate in an angular machine coordinate system.

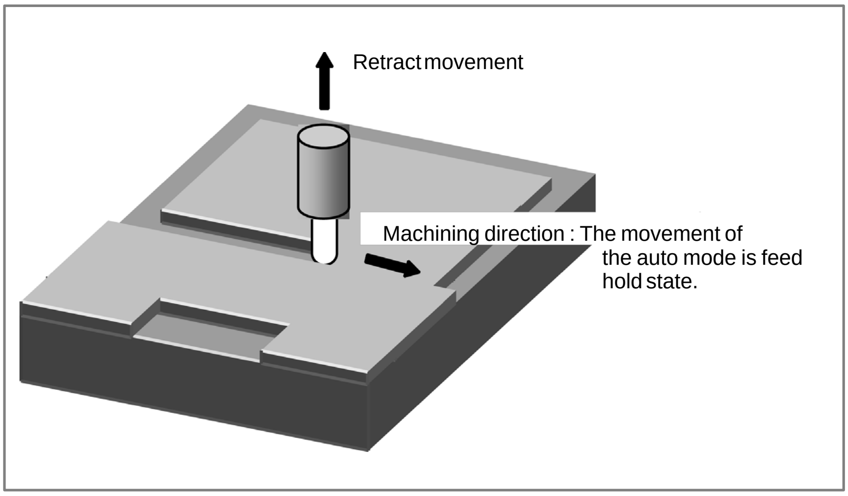
NOTE
When the optional function for angular axis control is selected, no stroke check before movement is made if this function is not enabled.

1.9 GENERAL PURPOSE RETRACT

General

When the retract signal RTRCT is turned to ”1” (the rising edge is detected) in auto mode or manual mode, the axis set in bit 0 (RTR) of the parameter No.7730 moves (retracts) by the amount set in the parameter No.7741.
Upon the completion of retraction, the retract completion signal RTRCTF is output.

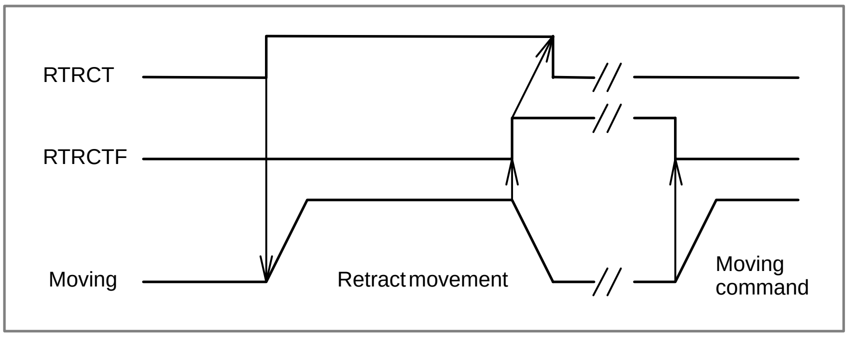
- Feedrate in retract is set to the parameter No.7740. Then the feedrate override is not effective.
- During the movement in retract, feed hold is not effective.
- In case that retract signal is turned to ”1” in auto mode, the movement of the auto mode is feed hold state and the retract movement is done.
- The retract completion signal is turned to ”0” when any retract axis is moved.



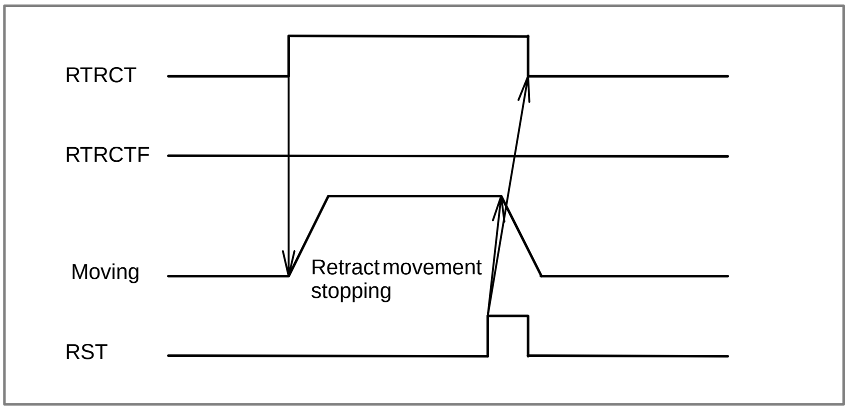
This function is used, for example, to prevent the damage of tool or workpiece when unexpected disturbance occurs during machining, and so on.

Timing chart

(1) ON/OFF timing of RTRCT and RTRCTF signalsRTRCT



(2) The stopping of retract by the reset



Signal

Retract signal RTRCT <G066#4>

[Classification] Input signal

[Function] Performs retraction for the axis specified with a parameter.

[Operation] When this signal turns to ”1”, the control unit performs the following:

Detects the rising edge of this signal, and performs retraction on the axis specified with bit 0 (RTRx) of parameter No.7730. The retract speed and amount of retraction are specified with parameter Nos.7740 and 7741. The retract signal is effective both in automatic operation mode and manual operation mode. When the retract signal turns to ”1” during automatic operation, retraction is performed and the CNC enters feed hold state.

Retract completion signal RTRCTF <F065#4>

[Classification] Output signal

[Function] Posts notification of the completion of retraction.

[Operation] The signal becomes ”1” when :

- Upon the completion of retraction.
  In case that there are two or more retract axes, upon the completion of retraction of all retract axes.

The signal becomes ”0” when :

- Upon the completion of retraction, when one of the retract axis is specified after that.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066				RTRCT				

	#7	#6	#5	#4	#3	#2	#1	#0
F065				RTRCTF				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7730								RTRx

[Data type] Bit axis

RTRx The retract function is :

0 : Disabled.
1 : Enabled.

7740	Feedrate during retraction for each axis

[Data type] 2 Words axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000

This parameter sets the feedrate during retraction for each axis

7741	Retracted distance for each axis

[Data type] 2 Words axis

[Unit of data]

Increment system	Unit of data	
	IS–B	IS–C
Millimeter input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

[Valid data range] –99999999 to 99999999

This parameter sets the retracted distance for each axis

Notes

(1) Feedrate override is not supported for retracting.

(2) Interlock is supported for retracting.

(3) Machine lock is supported for retracting. Retract completion signal is output when retract operation is completed in the machine lock condition.

(4) Feed hold is not supported for retracting.

(5) The mirror image (the signal or the setting) is invalid. Therefore, the direction of retract is the direction of the machine coordinate. (The mirror image is valid for update of the absolute coordinate.)

(6) When retract is executed during automatic operation, the control unit enters the feed hold state at the same time as the retract operation starts.

(7) The acc./dec. of retract is the condition of the acc./dec. at retract execution.

(8) Retract operation stops, when Reset or Emergency–stop is executed during retract movement. At this time, the retract completion signal is not turned to ”1”.

(9) When the servo alarm or the OT alarm of the retract axis occurs during retract movement, retract operation stops. At this time, the retract completion signal is not turned to ”1”. However, when an alarm except the OT alarm or the servo alarm occurs, retract operation does not stop.

(10) Even if the retract signal RTRCT is turned to ”0” after retract operation starting, retract operation does not stop.

(11) The retract signal RTRCT is not accepted while the retract completion signal RTRCTF is set to ”1”.

(12) Even if thread cutting is executed, retract is effective. When the retract signal is input, the thread cutting operation is stopped at once and retract operation is executed.

(13) Even if the retract is executed to the axis controlled by PMC, the movement command for the PMC axis is not canceled. In this case, the PMC axis must be canceled by the PMC axis reset signal ECLRg at the same time as the retract signal RTRCT is turned to ”1”.

(14) During advanced preview control mode, AI advanced preview control mode, or AI contour control mode, this function is not used.

2 PREPARATIONS FOR OPERATION

2.1 EMERGENCY STOP

General

If you press Emergency Stop button on the machine operator’s panel, the machine movement stops in a moment.

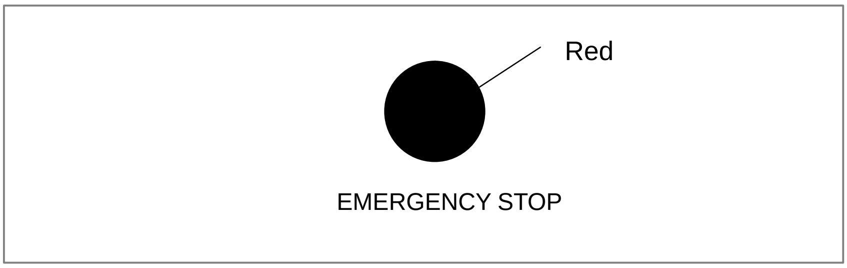


Fig. 2.1 (a) EMERGENCY STOP

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Signal

Emergency stop *ESP<X008#4,G008#4>

[Classification] Input signal

[Function] Activating an emergency stop signal stops the machine instantly.

[Operation] When the emergency stop signal *ESP turns to “0”, the emergency stop is applied to the machine and the CNC is reset. This signal is controlled by the B contacts of a pushbutton switch. The emergency stop signal turns the servo ready signal (SA) to “0”.

Overtravel detection by this CNC is handled by the stored stroke check function, and a limit switch for normal overtravel detection is not needed. To prevent the machine from moving beyond the software limit through servo feedback error, always install a stroke end limit switch (shown in Fig. 2.1 (b) as follows).

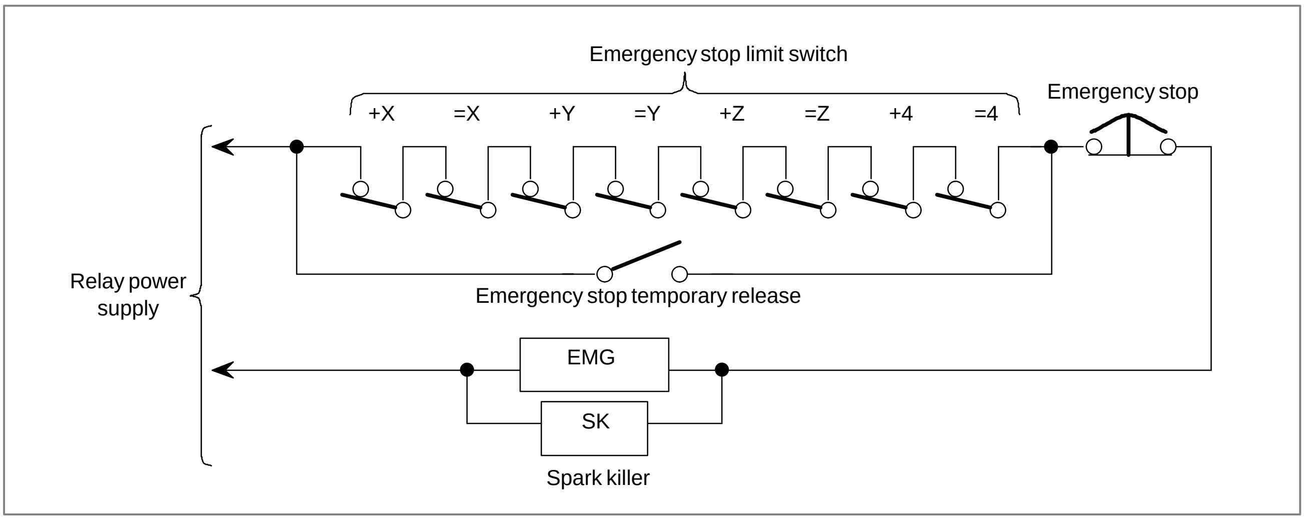


Fig. 2.1 (b) Connection of emergency stop limit switch

The distance from the position where the dynamic brake is applied to that where the tool stops moving is given in the "AC Servo Motor αi series Descriptions."

WARNING

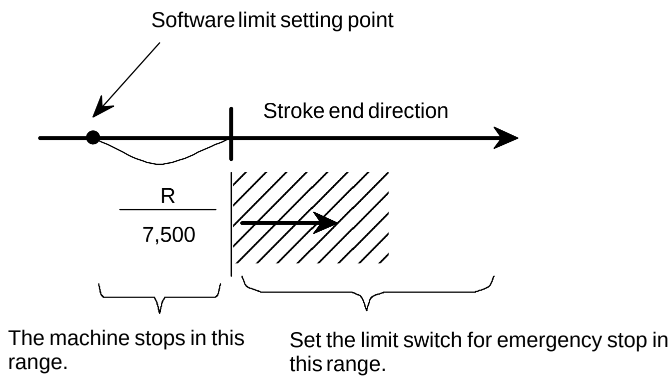
Software limit setting point and operating point of limit switch for emergency stop

The stop point by the software limit goes beyond the setting point by as much as the following distance.

$$\frac{R}{7,500} \text{ (mm)}$$

R: Rapid traverse rate (mm/min)

The actual stopping point may exceed the position set by a parameter (Nos.1320 and 1321) by as much as R/7500 (mm). Set the limit switch for emergency stop including the allowance for the above value.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP				

	#7	#6	#5	#4	#3	#2	#1	#0
G008				*ESP				

Reference item

FANUC AC SERVO MOTOR α*i*s/α*i* series DESCRIPTIONS	B–65262EN

2.2 CNC READY SIGNAL

General

When the CNC is turned on and becomes ready for operation, the CNC ready signal is set to 1.

Signal

CNC Ready Signal MA<F001#7>

[Classification] Output signal

[Function] The CNC ready signal indicates that the CNC is ready.

[Output condition] When the CNC is turned on and becomes ready for operation, the signal is set to 1. Normally, it takes several seconds to establish this state after the power is turned on. If a system alarm is issued, the signal is set to 0. The signal remains set to 1, however, when an emergency stop or a similar operation is performed.

Servo Ready Signal
SA <F000#6>

[Classification] Output signal

[Function] Signal SA turns to "1" when the servo system is ready to operate. For an axis that is to be braked, release the brake when this signal is "1" and apply the brake when this signal is "0".
Time chart of this signal is as follows:

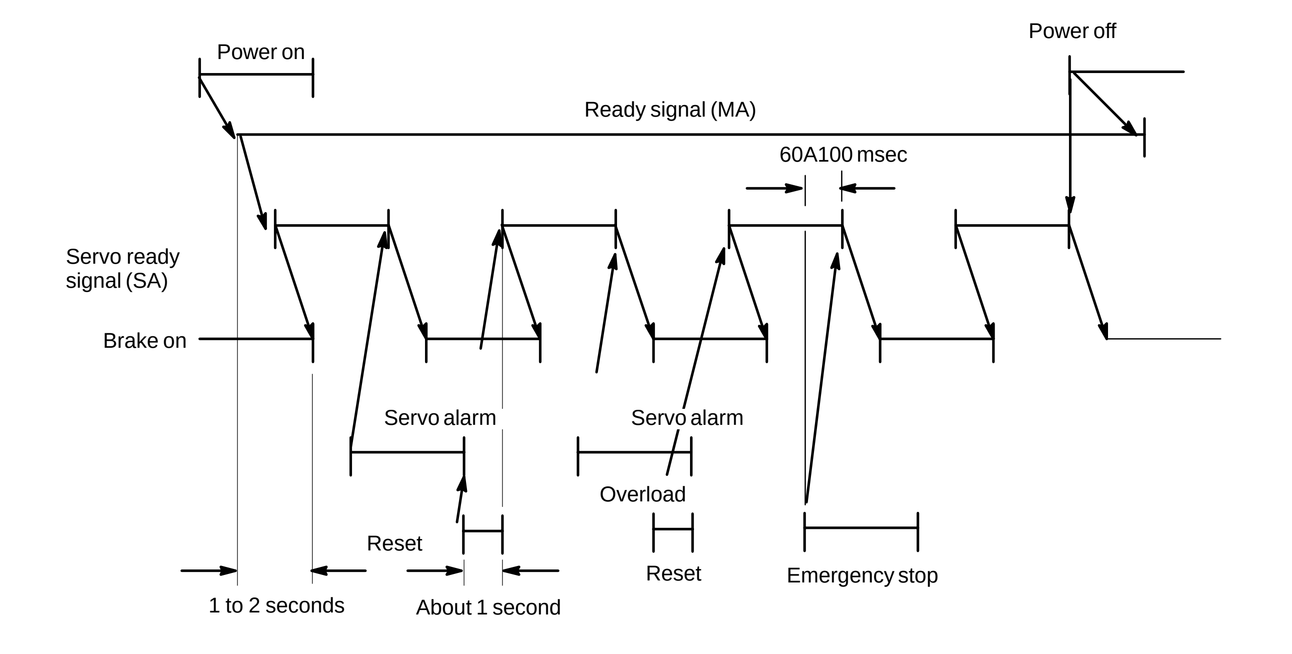


Fig. 2.2 Time chart for servo ready signal

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F000		SA						

	#7	#6	#5	#4	#3	#2	#1	#0
F001	MA							

2.3 OVERTRAVEL CHECK

2.3.1 Overtravel Signal

General

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops as a result of tripping the limit switch, and an OVER TRAVEL is displayed. The signal can be output with an alarm.

Signal

Overtravel signal *+L1 to *+L4<G114> *+L1 to *–L4<G116>

[Classification] Input signal

[Function] Indicates that the control axis has reached its stroke limit. There are individual signals for each direction in every control axis. The +/– in the signal name indicates the direction and the number corresponds to the control
axis.

+ L 1

1 No. 1 axis is at stroke limit.
2 No. 2 axis is at stroke limit.
3 No. 3 axis is at stroke limit.
: :

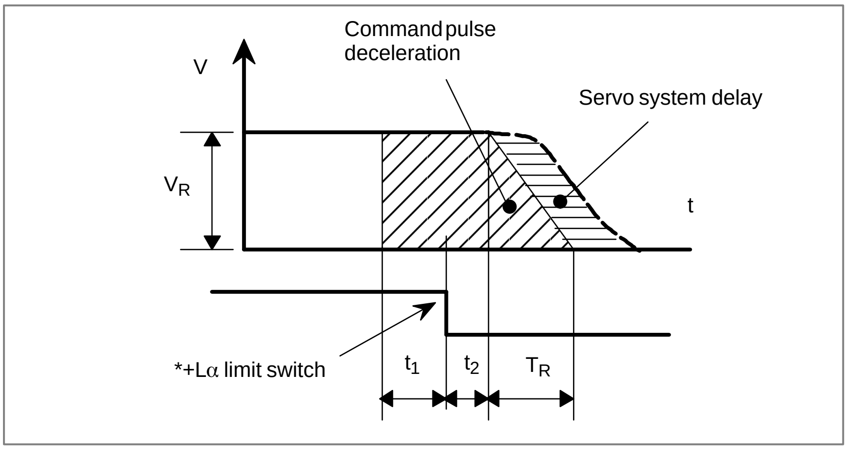
\+ Limit reached in + direction.
– Limit reached in – direction.

[Operation] When it is "0" , the control unit operates as given below.

- In automatic operation, if even one axis overtravel signal turns to "0", all axes are decelerated to stop, an alarm is given and operation is halted.
- In manual operation, only the axis whose overtravel signal has turned to "0" is decelerated to a stop, and the axis can be moved in the opposite direction.
- Once the axis overtravel signal has turned to "0", the axis direction is registered. Even if the signal returns to "1", it is not possible to move that axis in that direction until the alarm is cleared.

The following shows the deceleration distance at overtravel.

(i) Rapid traverse



$$L_1=V_R(t_1+t_2+\frac{T_R}{2}+T_S)\cdot\frac{1}{60000}\ \text{[mm or inch]}$$

L_1:Deceleration distance

V_R: Rapid traverse speed (mm/min or inch/min)

t_1: Limit switch signal delay time (from limit switch operation to *+Lα signal turn off (ms))

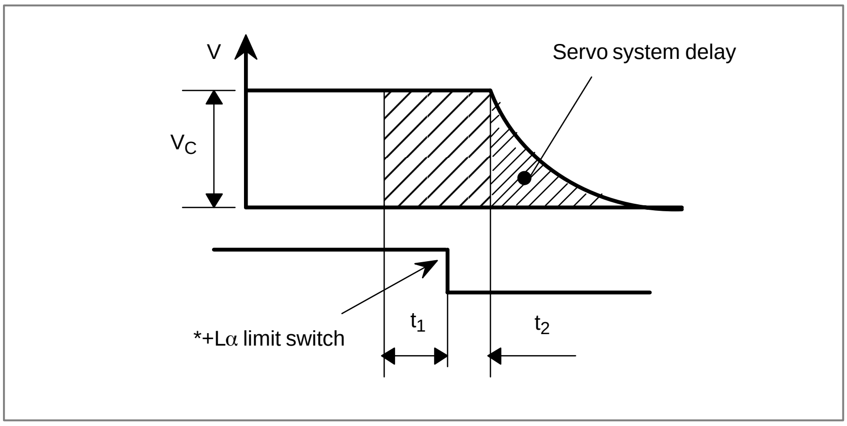
t_2: Receiver delay time 30ms

T_R: Rapid traverse acceleration/deceleration time constant (ms)

T_S: Servo system time constant (ms)

NOTE
Servo system time constant TS is 33 msec when the servo unit is adjusted to the standard setting.

(ii)Cutting feed



$$L_2=V_C(t_1+t_2+\frac{T_R}{2}+T_S)\cdot\frac{1}{60000}\ \text{[mm or inch]}$$

L_2:Deceleration distance

Vc: Maximum feedrate (mm/min or inch/min)

t_1, t_2, Ts: Same as (i).

- **Releasing overtravel**

First, move the tool into a safe zone under manual operation. Then press the reset button to reset the alarm.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G114					*+L4	*+L3	*+L2	*+L1
G116					*–L4	*–L3	*–L2	*–L1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

[Data type] Bit

OTH The overtravel signal is:

0 : Checked
1 : Not checked

WARNING
For safety, set to 0 at checking.

Alarm and message

Number	Message	Description
506	OVER TRAVEL : +n	Tool has moved beyond overtravel limit of the n–th axis in positive direction.
507	OVER TRAVEL : –n	Tool has moved beyond overtravel limit of the n–th axis in negative direction.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.6.2	Overtravel
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.6.2	Overtravel
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.6.2	Overtravel
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.6.2	Overtravel

2.3.2
Stored Stroke Check 1

General

When the tool tries to moved beyond a stored stroke check limit, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden zone and an alarm is generated, the tool may only be moved in the direction from which the tool came.

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) define the boundary. The forbidden zone lies outside the defined check limits. The machine tool builder usually sets this zone to permit maximum stroke.
The parameters used for stroke check can be switched by a signal.
Parameters can be used to change the stroke check method as follows (M series only):

- For a manual operation, setting both the BFA (bit 7 of No. 1300) and OTF (bit 5 of No. 1301) parameters to 1 causes an alarm to be issued on the border of the forbidden area and the machine to stop accordingly.
- For a manual operation, setting the NAL (bit 1 of No. 1300) to 1 causes the stroke limit reached signals +OT1 to +OT4 and –OT1 to –OT4 <F124 and F126> to be output without detecting an alarm condition.

Automatic alarm releasing

Setting the OF1 (bit 4 of No. 1301) parameter to 1 causes an alarm to be released when the axis enters a movable range without using a reset.

Signal

Stored stroke check select signal EXLM <G007#6>

[Classification] Input signal

[Function] Selects stroke check 1–I (parameter Nos. 1320 and 1321) or stroke check 1–II (parameter Nos. 1326 and 1327).

[Operation] When this signal is set to 1, the control unit operates as follows:
– Checks stroke check 1 on the basis of parameter Nos. 1326 and 1327, instead of parameter Nos. 1320 and 1321.

NOTE
If the bit 0 (DLM) of parameter No.1301 is set to 1, this signal is disabled.

Axis direction dependent stored stroke limit switch signal +EXL1 to +EXL4<G104> –EXL1 to –EXL4<G105>

[Classification] Input signal

[Function] Switches between stroke limit 1–I (parameter No. 1320 and No. 1321) and stroke limit 1–II (parameter No. 1326 and No. 1327) for each axis direction.

[Operation] When this signal goes 1 while DLM (bit 0 of parameter No. 1301) is held to 1, the control unit operates as described below.

(1) +EXL1 to +EXL4 <G104>
Stroke limit 1 (+ side) is checked using parameter No. 1326 instead of No. 1320.

(2) –EXL1 to –EXL4 <G105>
Stroke limit 1 (– side) is checked using parameter No. 1327 instead of No. 1321.

NOTE
If the bit 0 (DLM) of parameter No.1301 is set to 1, the stored stroke limit switch signal EXLM<G007#6> is disabled.

Stroke check external setting signals +LM1 to +LM4 <G110> and –LM1 to –LM4 <G112> (M series)

[Classification] Input signal

[Function] Change the values of the parameters governing the stroke check (1320 and 1321).

[Operation] When these signals are set to 1, the control unit operates as follows:
– Change the stored checks, set with parameter Nos. 1320 and 1321, to the machine coordinates when the signals are input.

Stroke check release signal RLSOT <G007#7> (M series)

[Classification] Input signal

[Function] Selects whether the stored stroke check 1 limits are checked or not.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Does not check the stored stroke check 1 limits.

Stroke limit reached signals +OT1 to +OT4 <F124> –OT1 to –OT4 <F126> (M series)

[Classification] Output signal

[Function] Notify that the tool is about to enter the forbidden area of stored stroke check 1. Each direction of each controlled axis has one stroke limit reached signal. The algebraic sign +/– in the signal name corresponds to the direction of each controlled axis, and the number at the end of the signal name represents the related controlled–axis number.

[Operation] If the tool has gone beyond the border of stored stroke check 1, the signal corresponding to the related axis direction becomes 1. Moving the tool in the opposite direction to put it back within the border (movable range) turns the signal to 0. When the tool is within the border (movable range), a reset turns the signal to 0.

CAUTION

1 The stroke limit reached signal is output when the NAL (bit 1 of No. 1300) parameter is 1. It is not output when the parameter is 0.

2 If the BFA (bit 7 of No. 1300) is 1, the tool does not go beyond the border even if a movement command that attempts to drive the tool beyond the border is issued. Instead, the tool stops rather inside the border (or, if the OTF (bit 5 of No. 1301) parameter is 1, on the border). Also in this case, the stroke limit reached signal becomes 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007	RLSOT	EXLM						
G104					+EXL4	+EXL3	+EXL2	+EXL1
G105					–EXL4	–EXL3	–EXL2	–EXL1
G110					+LM4	+LM3	+LM2	+LM1
G112					–LM4	–LM3	–LM2	–LM1
F124					+OT4	+OT3	+OT2	+OT1
F126					–OT4	–OT3	–OT2	–OT1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR				LMS	NAL	

[Data type] Bit

NAL Specifies whether to issue an alarm related to stored stroke check 1, as follows:

0 : To issue an alarm.
1 : Not to issue an alarm; the stroke limit reached signal F124 or F126 is output (for a manual operation).

LMS The EXLM signal for switching stored stroke check 1

0: Disabled
1: Enabled

LZR Checking of stored stroke check 1 during interval between power–on and setting the manual position reference return

0: The stroke check 1 is checked.
1: The stroke check 1 is not checked

NOTE
When the absolute–position detector is being used, and the reference position is already set at power–on, the stored stroke check is checked immediately after the power is turned on, regardless of the setting of this bit.

BFA When a command is issued where the resulting motion would exceed the value of stored stroke check 1 or 3

0: An alarm is generated after the stroke check 1, 3 is exceeded.
1: An alarm is generated before the stroke check 1, 3 is exceeded.

	#7	#6	#5	#4	#3	#2	#1	#0
1301			OTF	OF1				DLM

[Data type] Bit

DLM Axis direction dependent stored stroke limit switch signals +EXL1 to +EXL4 and –EXL1 to –EXL4 <G104 and G105> are
0: Disabled.
1: Enabled

NOTE
If the bit 0 (DLM) of parameter No. 1301 is set to 1, the stored stroke limit switch signal EXLM <G007#6> is disabled.

OF1 If the tool is moved into the range allowed on the axis after an alarm is generated by stored stroke check 1,
0: The alarm is not canceled before a reset is made.
1: The OT alarm is immediately canceled.

CAUTION

In the cases below, the automatic release function is disabled. To release an alarm, a reset operation is required.

1 When a setting is made to issue an alarm before a stored stroke limit is exceeded (bit 7 (BFA) of parameter No. 1300)
2 When an another overtravel alarm (such as stored stroke check 2, stored stroke check 3, and interference check) is already issued

OTF Specifies whether to change the specification of the stored stroke check, as follows:

0 : Not to change.
1 : To change as stated below.

- If the BFA (bit 7 of No. 1300) parameter is 1, stored stroke check 1 for a manual operation causes the axis to stop on the border and an alarm to be issued.
- If the BFA (bit 7 of No. 1300) parameter is 1, stored stroke check 2 causes an alarm to be issued just before the stroke check is passed through.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Define the coordinate values of stored stroke checks 1 in the positive and negative directions for each axis in the machine coordinate system. For each axis, travel beyond the defined limits is prohibited.

WARNING

1 For axes with diameter specification, a diameter value must be set.
2 When the parameters are set as follows, the stroke becomes infinite:
   parameter 1320 < parameter 1321
   For movement along the axis for which infinite stroke is set, only incremental commands are available. If an absolute command is issued for this axis, the absolute register may overflow, and normal movement will not result.

1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Define the coordinate values of stored stroke checks 1 in the positive and negative directions for each axis in the machine coordinate system.

When stroke check switching signal EXLM is ON, stroke checks are checked with parameters 1326 and 1327, not with parameters 1320 and 1321. For each axis, travel beyond the defined limits (parameter Nos. 1326 and 1327) is prhibited.

NOTE
The EXLM signal is enabled only when LMS, #2 of parameter 1300, is set to 1.

Alarm and message

Number	Message	Description
500	OVER TRAVEL : +n	Tool has moved beyond overtravel limit of n–th axis (n: 1 to 4) in positive direction stored stroke check 1
501	OVER TRAVEL : –n	Tool has moved beyond overtravel limit of n–th axis (n: 1 to 4) in negative direction stored stroke check 1

Caution

CAUTION
By setting the same value for both check limits of a given axis, the entire axis become restricted.

Note

NOTE

1 Parameter LZR (bit 6 of No. 1300) selects whether each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on.

2 Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.6.3	Stored stroke check
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.6.3	Stored stroke check
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.6.3	Stored stroke check
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.6.3	Stored stroke check

2.3.3 Stored Stroke Check 2, 3

General

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2,and stored stroke check 3.

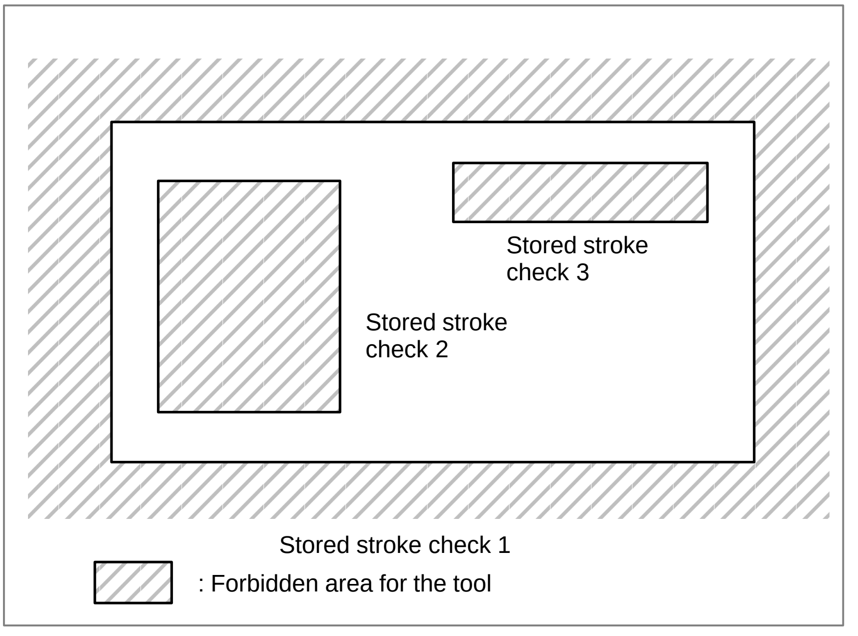


Fig. 2.3.3 (a) Stroke check (T series)

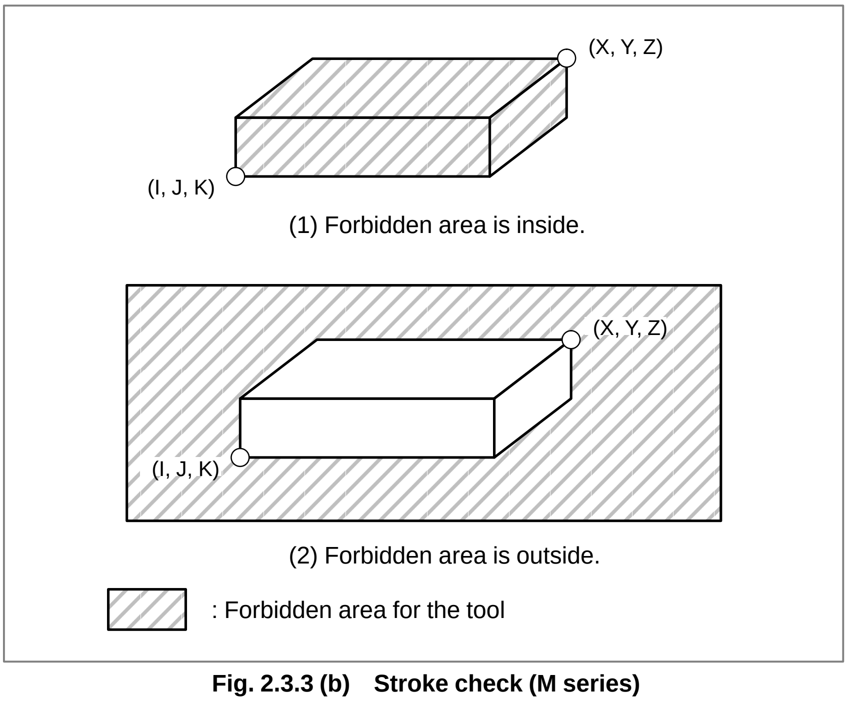


Fig. 2.3.3 (b) Stroke check (M series)

When the tool tries to move beyond a stored stroke check limit, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool may only be moved in the direction from which the tool came.

Stored stroke check 2

The stored stroke check 2 values are set either by parameters (Nos. 1322, 1323) or by command. The foribidden area may be defined as the area external to the limits, or internal to the limits. This is determinal by the value in parameter OUT (No. 1300#0). A G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. G22 and G23 should be commanded independently of any other commands in a block.
Setting both the BFA (bit 7 of No. 1300) and OTF (bit 5 of No. 1301) parameters to 1 can issue an alarm just before the forbidden area is entered (M series only).
The command below creates or changes the forbidden area:

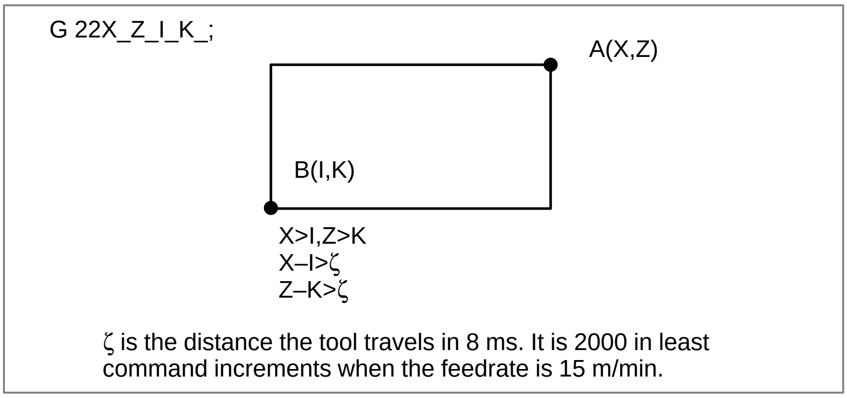


Fig. 2.3.3 (c) Creating or changing the forbidden area using a program (T series)

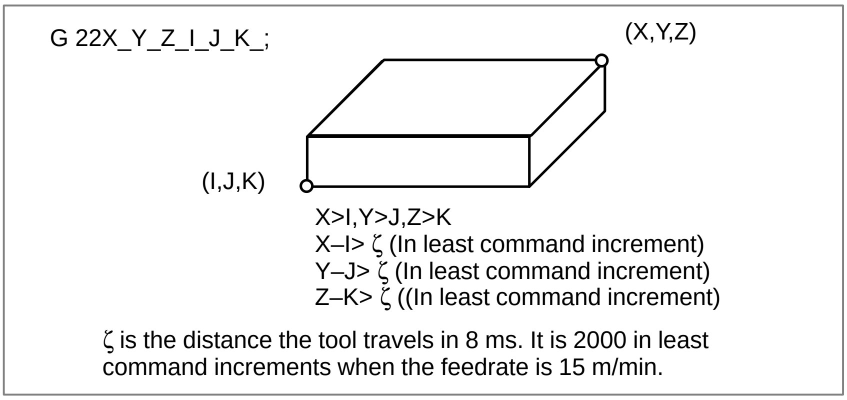


Fig. 2.3.3 (d) Creating or changing the forbidden area using a program (M series)

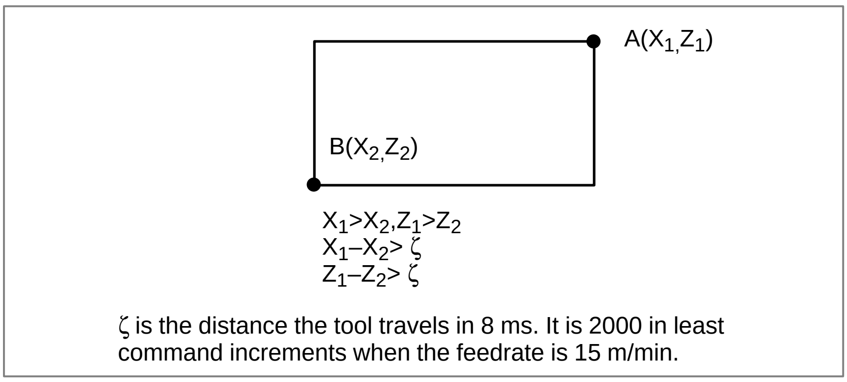


Fig. 2.3.3 (e) Creating or changing the forbidden area using a parameters (T series)

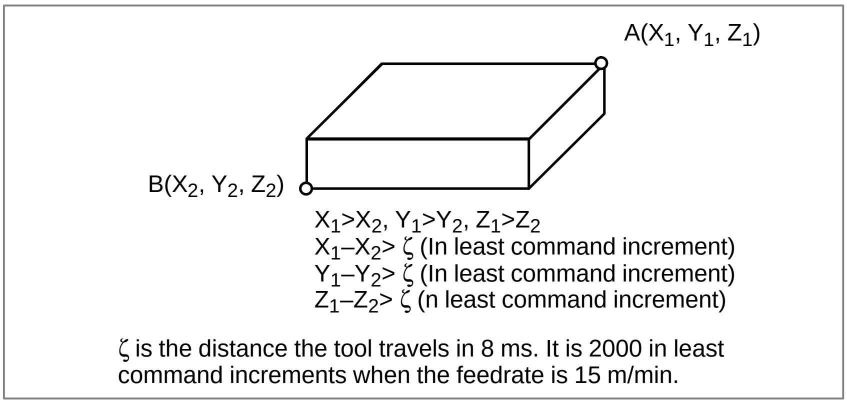


Fig. 2.3.3 (f) Creating or changing the forbidden area using parameters (M series)

When setting the forbidden area X_1, Y_1, Z_1, X_2, Y_2 and Z_2 by parameters (Nos. 1322 and 1323), specify the data as the distance from the reference position in units of the least command increment (output increment).
When setting the forbidden area X, Y, Z, I, J, K (X, Z, I, K, on T series) by a G22 command, specify the data as the distance from the reference position in units of the least input increment (input increment). The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

- **Stored stroke check 3**

Define the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area.

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.
If point C (The tip of the tool) is checked in Fig. 2.3.3 (g), the distance “c” should be set as the data for the stored stroke check function. If point D (The tool chuck) is checked, the distance “d” must be set.

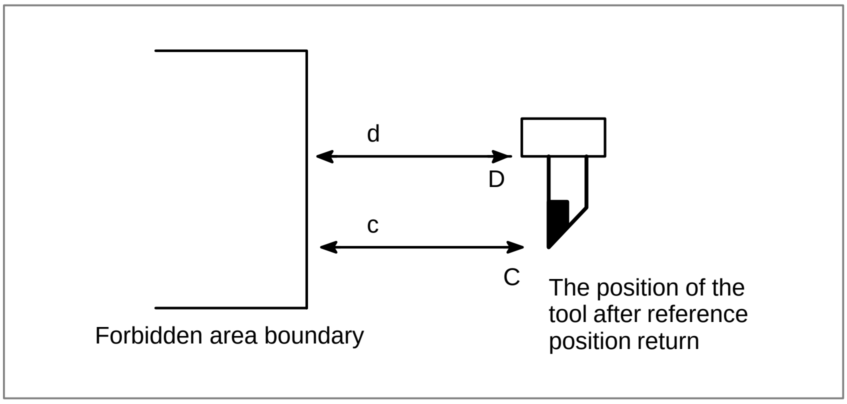


Fig. 2.3.3 (g) Setting the forbidden area (T series)

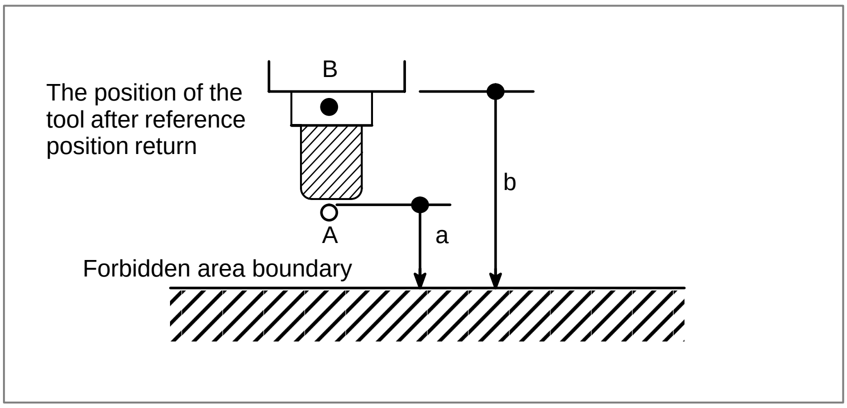


Fig. 2.3.3 (h) Setting the forbidden area (M series)

- **Forbidden area over–lapping**

Forbidden areas can be set to overlap.

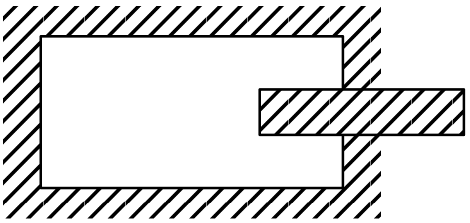
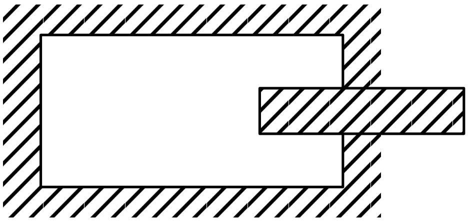


Fig. 2.3.3 (i) Setting the forbidden area overlapping (T series)

Unnecessary checks should be set beyond the machine stroke.



Setting the forbidden area overlapping

Fig. 2.3.3 (j) Setting the forbidden area overlapping (M series)

- **Effective time for a forbidden area**

Parameter LZR (bit 6 of No. 1300) selects whether each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on.
After the power is turned on, if the reference position is in the forbidden area of each check, an alarm is generated immediately (Only in G22 mode for stored stroke check 2).

- **Releasing the alarms**

When the tool enters and forbidden area and an alarm is generated, the tool may only be moved in the direction from which the tool came. First move the tool out of the forbidden area, then clear the alarm by reset. If successfully cleared, the tool may be moved in both paths.

- **Change from G23 to G22 in a forbidden area**

When G23 is switched to G22 while the tool is in a forbidden area, the following results.

(1) When the forbidden area is internal to the limits, an alarm is generated in the next move.

(2) When the forbidden area is external to the limits, an alarm is generated immediately.

Signal

Stroke check 3 release signal RLSOT3 <G007#4>

[Classification] Input signal

[Function] Selects whether stored stroke check 3 is checked.

[Use] When this signal is set to 1, the control unit operates as follows:

· The control unit does not check stored stroke check 3.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007				RLSOT3				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR	RL3					OUT

[Data type] Bit

OUT The area inside or outside of the stored stroke check 2 is set as a restricted area.

0: Inside
1: Outside

RL3 Stroke check 3 release signal RLSOT3

0: The signal is disabled.
1: The signal is enabled.

LZR Checking of stored stroke check 1 during the time from power–on to the manual position reference return

0: The stroke check 1 is checked.
1: The stroke check 1 is not checked

NOTE
When the absolute–position detector is being used, and the reference position is already set at power–on, the stored stroke check is checked immediately after the power is turned on, regardless of the setting of this bit.

BFA When a command is issued where the resulting motion would exceed the value of a stored stroke check 1, 3

0: An alarm is generated after the stroke check 1, 3 is exceeded.
1: An alarm is generated before the stroke check 1, 3 is exceeded.

	#7	#6	#5	#4	#3	#2	#1	#0
1301			OTF					

[Data type] Bit

OTF Defines whether to change the specification of the stored stroke check, as follows:

0 : Not to change.
1 : To change as stated below.

- If the BFA (bit 7 of No. 1300) parameter is 1, stored stroke check 1 for a manual operation causes the axis to stop on the border and an alarm to be issued.
- If the BFA (bit 7 of No. 1300) parameter is 1, stored stroke check 2 causes an alarm to be issued just before the stroke check is passed through.

	#7	#6	#5	#4	#3	#2	#1	#0
1310							OT3x	OT2x

[Data type] Bit axis

OT2x Defines whether stored stroke check 2 is checked for each axis is set.

0: Stored stroke check 2 is not checked.
1: Stored stroke check 2 is checked.

OT3x Defines whether stored stroke check 3 is checked for each axis is set.

0: Stored stroke check 3 is not checked.
1: Stored stroke check 3 is checked.

1322	Coordinate value of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value of stored stroke check 2 in the negative direction on each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Defines the coordinate values of stored stroke checks 2 in the positive and negative directions for each axis in the machine coordinate system. OUT, #0 of parameter 1300, sets either the area outside or the area inside specified by two checks as the inhibition area.

WARNING
For axes with diameter specification, a diameter value must be set.

1324	Coordinate value of stored stored check 3 in the positive direction on each axis
1325	Coordinate value of stored stroke check 3 in the negatice direction on each axis

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Defines the coordinate values of stored stroke checks 3 in the positive and negative directions for each axis in the machine coordinate system. For each axis, travel within the area defined by the check limits is prohibited.

Alarm and message

Number	Message	Description
502	OVER TRAVEL : +n	Tool has moved beyond overtravel limit of n–th axis in positive direction stored stroke check 2. (Parameter No.1322)
503	OVER TRAVEL : –n	Tool has moved beyond overtravel limit of n–th axis in negative direction stored stroke check 2. (Parameter No.1323)
504	OVER TRAVEL : +n	Tool has moved beyond overtravel limit of n–th axis in positive direction stored stroke check 3. (Parameter No.1324)
505	OVER TRAVEL : –n	Tool has moved beyond overtravel limit of n–th axis in negative direction stored stroke check 3. (Parameter No.1325)

Warning

WARNING

1 Whenever the two check limits are set to the same value, the following results are seen.
   (1) In the case of stored stroke check 1, all areas are prohibited.
   (2) In the case of stored stroke check 2 or 3, no areas are prohibited.

2 Whenever the value of the negative limit is greather than the value of the positive limit, the following results are seen.
   (1) In the case of stored stroke check 1, no areas are prohibited.
   (2) In the case of stored stroke check 2 or 3, the prohibited area will consists of a quadrangle formed with the two points acting vertexes.

Note

NOTE

Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area. (check 1, 3 only)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.6.3	Stored stroke check
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.6.3	Stored stroke check
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.6.3	Stored stroke check
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.6.3	Stored stroke check

2.3.4 Chuck/Tailstock Barrier (T series)

General

The chuck/tailstock barrier function prevents damage to the machine by checking whether the tool tip interferes with either the chuck or tailstock. Specify an area into which the tool may not enter (entry–prohibition area). This is done using the special setting screen, according to the shapes of the chuck and tailstock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.
The tool can be removed from the prohibited area only by retracting it in the direction from which the tool entered the area.

Signal

Tailstock barrier select signal *TSB <G060#7>

[Classification] Input signal

[Function] Enables or disables the tailstock barrier.

[Operation] When this signal is set to 1, the control unit operates as follows:

– Disables the tailstock barrier, even when the G22 command (stored stroke check on) is specified in the program.

G code	*TSB	Tailstock barrier	Chuck barrier
G22	0	Enabled	Enabled
	1	Disabled	Enabled
G23	0	Disabled	Disabled
	1	Disabled	Disabled

When the G23 command (stored stroke check off) is specified, the tailstock barrier is disabled regardless of the *TSB signal. When the G22 command (stored stroke check on) is specified, the tailstock can be disabled by setting the signal to 1.

This signal is used to select whether the tailstock area is a prohibited area. It is used whenever M commands are applied, resulting in the tailstock being attached to the workpiece or detached from the wokpiece while the workpiece is being machined.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G060	*TSB							

Parameter

● Profile of a chuck

1330	Profile TY of a chuck

[Data type] Byte

[Valid data range] 0 or 1

0 : Chuck which holds a workpiece on the inner surface
1 : Chuck which holds a workpiece on the outer surface

(*) See Fig. 2.3.4 (a) for the figures.

1331	Dimensions of the claw of a chuck (L)
1332	Dimensions of the claw of a chuck (W)
1333	Dimensions of the part of a claw at which a workpiece is held (L1)
1334	Dimensions of the part of a claw at which a workpiece is held (W1)

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

1335	X coordinate of a chuck (CX)
1336	Z coordinate of a chuck (CZ)

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

Specify the profile of a chuck.

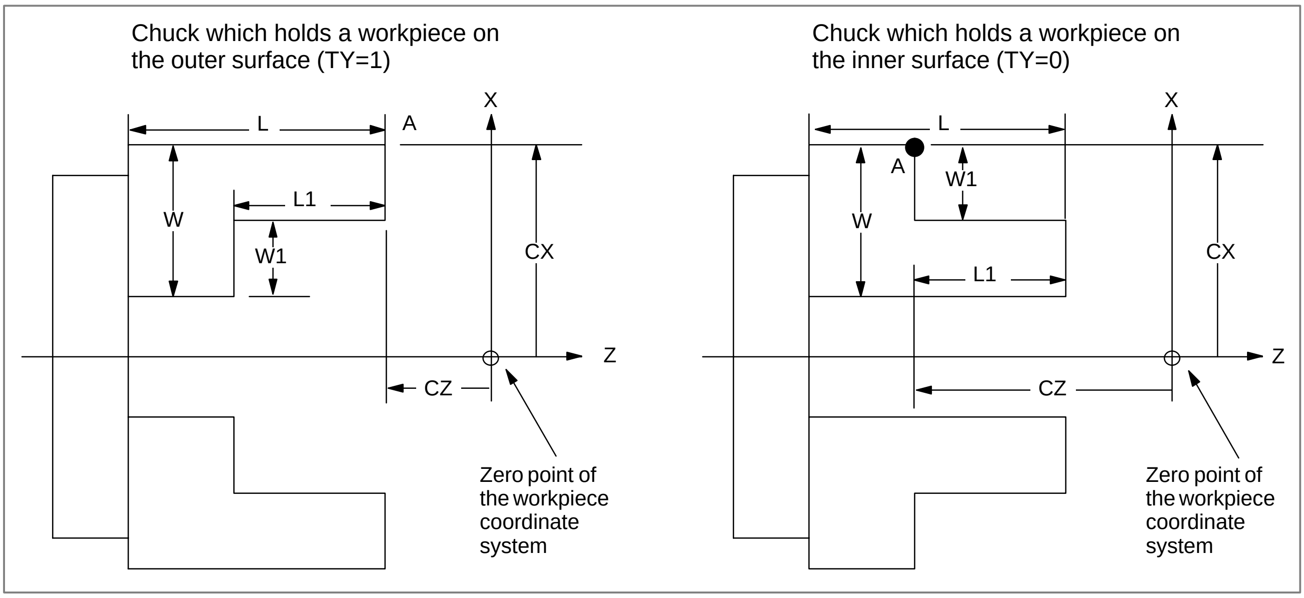


Fig. 2.3.4 (a)

Symbol	Description
TY	Profile of a chuck (0: Chuck which holds a workpiece on the inner surface, 1: Chuck which holds a workpiece on the outer surface)
CX	X coordinate of a chuck
CZ	Z coordinate of a chuck
L	Dimensions of the claw of a chuck
W	Dimensions of the claw of a chuck (radius input)
L1	Dimensions of the part of a claw at which a workpiece is held
W1	Dimensions of the part of a claw at which a workpiece is held (radius input)

TY: Specifies the profile of a chuck. When TY is set to 0, the chuck holding a workpiece on the inner surface is specified. When TY is set to 1, the chuck holding a workpiece on the outer surface is specified. The profile of the chuck is assumed to be symmetrical with respect to the z–axis.

CX, and CZ: Specify the position (point A) of a chuck with the coordinates of the workpiece coordinate system. In this case, do not use the coordinates of the machine coordinate system.

WARNING
Specifying the coordinates with a diameter or radius depends on whether the corresponding axis conforms to diameter or radius specification . When the axis conforms to diameter specification, the coordinates are specified with a diameter.

L, L1, W, and W1: Define the profile of a chuck.

WARNING
Always specify W and W1 with radii. L and L1 are also specified with radii whenever the Z–axis conforms to a radius specification.

● Profile of a tailstock

1341	Length of a tailstock (L)
1342	Diameter of a tailstock (D)
1343	Length of a tailstock (L1)
1344	Diameter of a tailstock (D1)
1345	Length of a tailstock (L2)
1346	Diameter of a tailstock (D2)
1347	Diameter of the hole of a tailstock (D3)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

1348	Z coordinate of a tailstock (TZ)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Specify the profile of a tailstock.

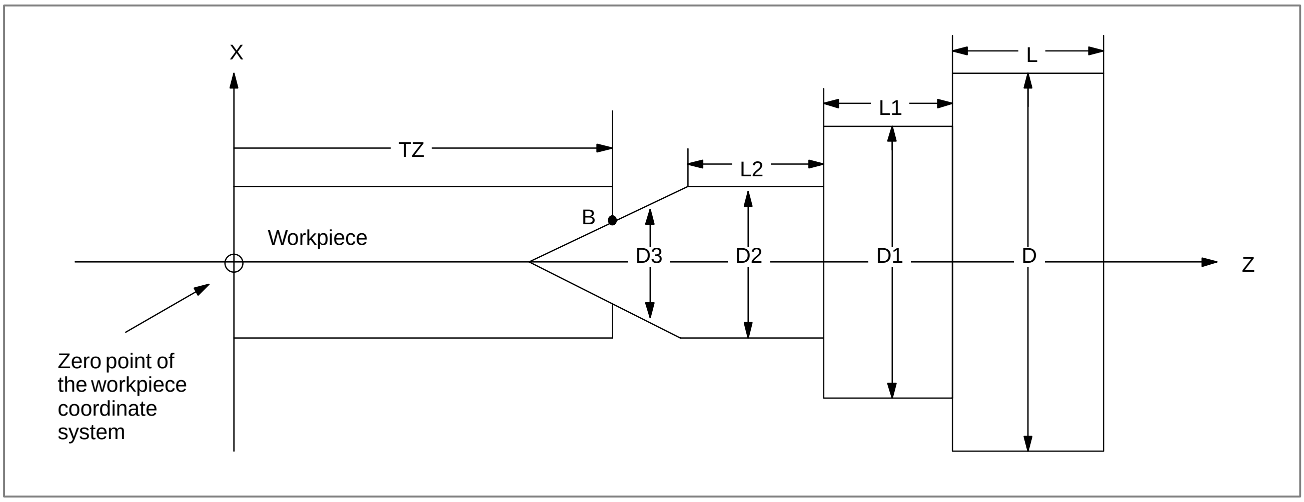


Fig. 2.3.4 (b)

Symbol	Description
TZ	Z–axis coordinate of a tailstock
L	Length of a tailstock
D	Diameter of a tailstock (diameter input)
L1	Length of a tailstock (1)
D1	Diameter of a tailstock (1) (diameter input)
L2	Length of a tailstock (2)
D2	Diameter of a tailstock (2) (diameter input)
D3	Diameter of the hole of a tailstock (diameter input)

TZ: Specifies the position (point B) of a tailstock with the Z–axis coordinate of the workpiece coordinate system. In this case, do not use the coordinate of the machine coordinate system. The profile of a tailstock is assumed to be symmetrical with respect to the Z–axis.

WARNING
Specifying the position of a tailstock with a radius or diameter depends on whether the Z–axis conforms to radius or diameter specification.

L, L1, L2, D, D1, D2, and D3:
Define the profile of a tailstock.

WARNING
D, D1, D2, and D3 are always specified with diameters. L, L1, and L2 are also specified with radii whenever the Z–axis conforms to radius specification.

Alarm and message

Number	Message	Description
502	OVER TRAVEL : +X	The tool has entered the forbidden area when moving in the positive direction along the X–axis.
	OVER TRAVEL : +Z	The tool has entered the forbidden area when moving in the positive direction along the Z–axis.
503	OVER TRAVEL : –X	The tool has entered the forbidden area when moving in the negative direction along the X–axis.
	OVER TRAVEL : –Z	The tool has entered the forbidden area when moving in the negative direction along the Z–axis.

Warning

WARNING

1 Invalid settings will result in the absence of a prohibited area, as follows:
   1) In the setting of the chuck shape, if the jaw length (parameter No. 1331) is less than the grasp length (parameter No. 1333) or if the jaw width (parameter No. 1332) is less than the grasp width (parameter No. 1334).
   2) In the setting of the tailstock shape, if the tailstock diameter (parameter No. 1346) is less than the hole diameter (parameter No. 1347).
   3) If the position of a chuck overlaps the position of a tailstock.

2 When the options for stored stroke check 2 or 3 and chuck/tailstock barrier are used at the same time, the chuck/tailstock barrier is valid but stored stroke check 2 or 3 is ignored.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	III.6.4	Chuck and Tailstock Barriers
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	III.6.4	Chuck and Tailstock Barriers

2.3.5 Stroke Limit Check Before Move

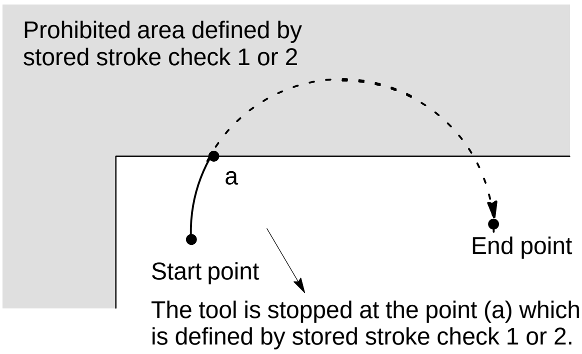
General

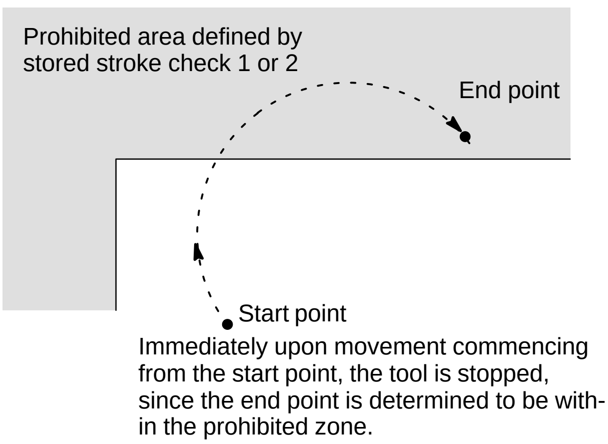
In automatic operation, before executing the move command by a given block, the position of the end point is determined. It is calculated from the current position of the machine and from the specified amount of travel. It is also determined whether or not the tool will enter the prohibited area defined by stored stroke check 1, 2 or 3. It it is determined that the tool will enter the prohibited area defined by a stored stroke check, the tool is stopped immediately once this block starts execution, and an alarm is displayed.

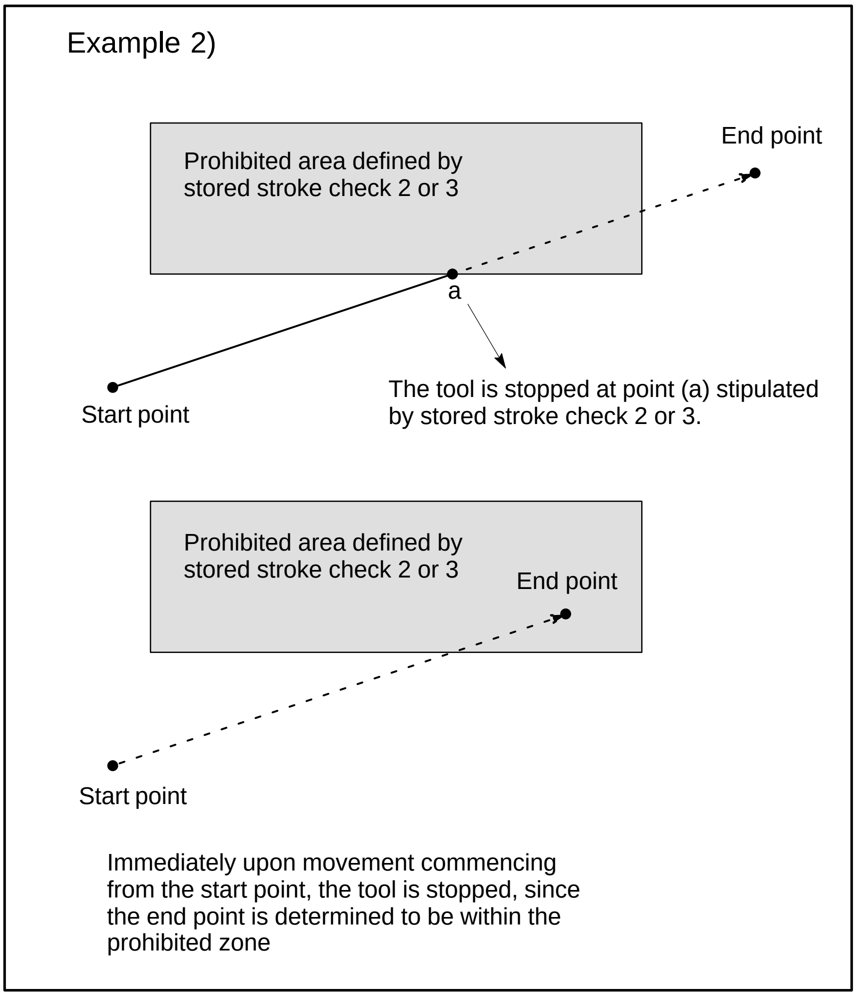
WARNING

Only the coordinates of the end point, reached as a result of traversing the distance specified in each block, are checked against the prohibited area. The coordinates along the path are not checked. However, if the tool enters the prohibited area defined by stored stroke check 1, 2, or 3, an alarm is issued at that point along the path. (See the examples below.)

Example 1)







Explanations

When a stroke limit check before move is performed, NPC (parameter No. 1301#2) is used to determine whether to check the moves performed by G31 (skip) or G37 (automatic tool length measurment).

Limitations

- **Machine lock**

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- **G23**

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the prohibited area defined by stored stroke check 2.

- **Program restart**

When a program is restarted, an alarm is issued if the restart position is within a prohibited area.

- **Manual intervention following a feed hold stop**

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even though the end point following a manual intervention is within a prohibited area.

- **A block consisting of multiple operations**

If a block consisting of multiple operations (such as a canned cycle) is executed, an alarm is issued at the start point of any operation whose end point falls within a prohibited area.

- **Cyrindrical interpolation mode** — In cylindrical interpolation mode, no check is made.
- **Polar coordinate interpolation mode** — In polar coordinate interpolation mode, no check is made.
- **Angular axis control** — When the angular axis control option is selected, no check is made.
- **Simple synchronous control** — In simple synchronous control, only the master axis is checked; no slave axes are checked.
- **Drawing** — While drawing in dynamic graphic display mode (only drawing is performed), no check is made.
- **PMC axis control** — No check is made for a movement based on PMC axis control.
- **Chuck/tailstock barrier** — The chuck/tailstock barrier area is not checked. (T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1301	PLC					NPC		

[Data type] Bit

NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement (for M series) or automatic tool compensation (for T series)) blocks is:

0: Checked
1: Not checked

PLC Stroke limit check before movement is:

0: Not performed
1: Performed

Alarm and message

Number	Message	Contents
510	OVER TRAVEL : +n	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit prohibited area in the positive direction of the n–axis. Correct the program or redefine the prohibited area.
511	OVER TRAVEL : –n	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit prohibited area in the negative direction of the n–axis. Correct the program or redefine the prohibited area.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.6.4	Stroke Limit Check Prior to Performing Movement
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.6.5	Stroke Limit Check Prior to Performing Movement
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.6.4	Stroke Limit Check Prior to Performing Movement
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.6.5	Stroke Limit Check Prior to Performing Movement

2.4 ALARM SIGNAL

General

When an alarm is triggered in the CNC, the alarm is displayed on the screen, and the alarm signal is set to 1.

If the voltage level of the memory backup battery falls to below a specified level while the CNC is turned off, the battery alarm signal is set to 1.

Signal

Alarm signal AL<F001#0>

[Classification] Output signal

[Function] The alarm signal reports that the CNC is in an alarm state.

The following are the alarms that may be issued:

(a) TH alarm
(b) TV alarm
(c) P/S alarm
(d) Overtravel alarm
(e) Overheat alarm
(f) Servo alarm

[Output condition] The alarm signal is set to 1 when:
– The CNC is placed in the alarm state.
The alarm signal is set to 0 when:
– The alarm has been released by resetting the CNC.

Battery alarm signal BAL<F001#2>

[Classification] Output signal

[Function] The battery alarm signal indicates that the voltage of the battery for the memory has fallen to below a specified level while the CNC is off. In general, this signal is used to turn on an LED to notify the operator.

[Output condition] The signal is set to 1 when:
– The battery voltage has fallen to below the specified level.
The signal is set to 0 when:
– The battery voltage has risen to the specified level or higher.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F001						BAL		AL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

[Data type] Bit

NPA Action taken when an alarm is generated or when an operator message is entered

0 : The display shifts to the alarm or message screen.
1 : The display does not shift to the alarm or message screen.

2.5 START LOCK/ INTERLOCK

General

These signals disable machine movement along axes. When any of these signals is activated during movement, tool movement along the affected axis (or axes) is decelerated, then stopped.

Signal

Start lock signal STLK<G007#1>(T series)

[Classification] Input signal

[Function] This signal disables machine movement along all axes subject to automatic operation (memory or MDI operation).

[Operation] When the STLK signal turns to “1”, the axis movement is decelerated and stopped.
In automatic operation, blocks containing M, S, T, or B commands or 2nd auxiliary function are executed consecutively until a block containing an axis move command is encountered; the movement then stops and the CNC is placed in automatic operation mode (STL is “1”, SPL is “0”). When the STLK signal turns to “0”, operation restarts. (Figs. 2.5 (a), (b)).

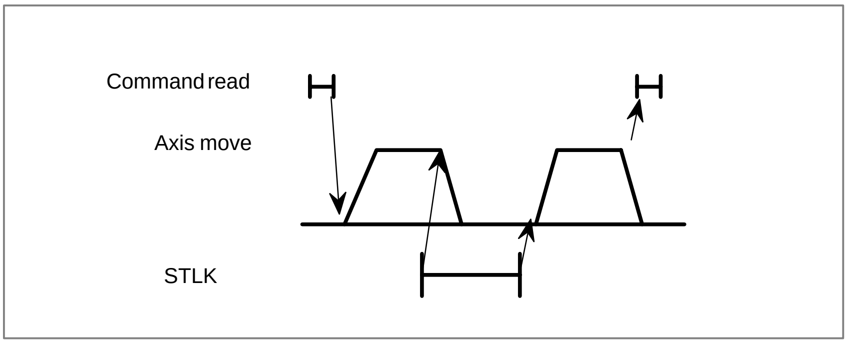


Fig. 2.5 (a) Block containing axis move commands only

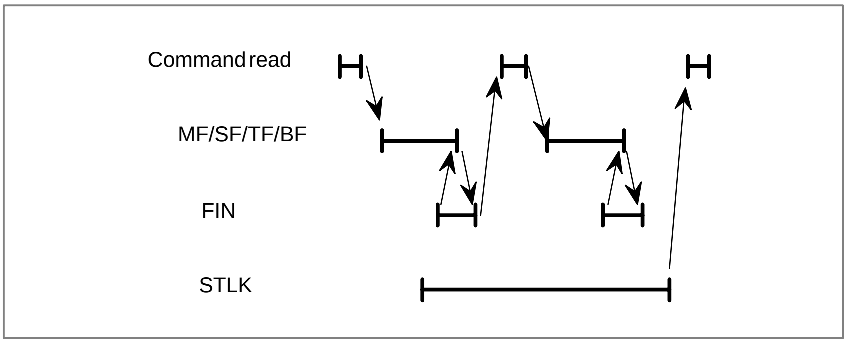


Fig. 2.5 (b) Block containing only auxiliary functions only

All axes Interlock signal *IT<G008#0>

[Classification] Input signal

[Function] This signal is used to inhibit the machine from moving, and is effective regardless of the selected mode.

[Operation] When the *IT signal is “0”, the axis movement is decelerated and stopped. In automatic operation, blocks containing M, S, T or B (2nd auxiliary) mode commands are executed consecutively until a block containing an axis move command is encountered; the system then stops and is placed into the automatic operation mode (cycle start lamp signal STL is “1”, feed hold lamp signal SPL is “0”). When the *IT signal turns to “1”, operation resumes (Figs. 2.5(c), (d)).

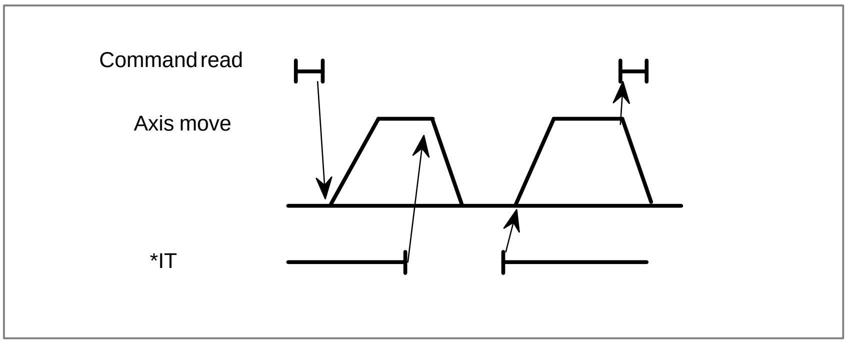


Fig. 2.5 (c) Block containing axis move commands only (manual and automatic operation)

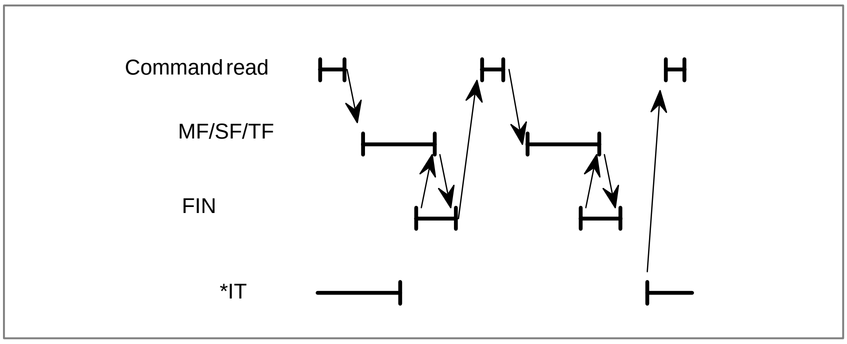


Fig. 2.5 (d) Block containing auxiliary functions only (automatic operation)

NOTE

The overtravel distance of the motor after turning *IT to “0” is represented by the following formula.

$$Q_{max} = F_m \cdot \frac{1}{60} \times \left(\frac{T_c}{1000} + \frac{T_s}{1000} + \frac{A}{1000} \right)$$

Where

Q_{max} : Overtravel quantity (mm or inch)
F_m : Feedrate (mm/min or inch/min)
T_c : Cutting time constant (ms)
T_s : Servo time constant (Ts = 33ms normally)
A : Processing time of CNC
A = 50ms

Interlock signal for each axis *IT1 to *IT4<G130>

[Classification] Input signal

[Function] These signals disable feed along axes on an axis–by–axis basis.
A separate interlock signal is provided for each controlled axis. The number at the end of each signal name denotes the number of the corresponding controlled axis.

```
*IT 1
    1 ..... Interlock for the first axis
    2 ..... Interlock for the second axis
    3 ..... Interlock for the third axis
    :    :
    :    :
```

[Operation] a) In manual operation

The movement of an interlocked axis is inhibited, but the other axes are movable. If an axis is interlocked during movement, it stops after being decelerated, and it starts moving again when it is released from interlock.

b) In automatic operation (MEM RMT or MDI mode)

If an axis is interlocked while its movement is being commanded (the move amount is not 0, inclusive of the tool offset), movement in all axes is prevented.

If a moving axis is interlocked, all axes stop moving after being decelerated, and they start moving again when interlock is released.

This function is also effective during dry run.

Interlock signal for each axis and direction +MIT1, –MIT1, +MIT2, –MIT2, +MIT3, –MIT3, +MIT4, –MIT4, <G132#0 to #3, G134#0 to #3> (M series) +MIT1, –MIT1, +MIT2, –MIT2 <X004#2~X004#5>(T series)

[Classification] Input signal

[Function] This function allows a directional interlock for each axis.

[Operation] When the axis/directional interlock signal becomes “1”, CNC applies interlock only in the corresponding axial direction. However, during automatic operation, all axes will stop.

NOTE
In the T series, when bit 4 (DAU) of parameter No. 3003 is 0, a directional interlock for each axis is applied only during manual operation. To allow a directional interlock for each axis also during automatic operation, set bit 4 (DAU) of parameter No. 3003 to 1.

Block start interlock signal *BSL<G008#3>

[Classification] Input signal

[Function] This signal disables the start of the next block during automatic operation

[Operation] While this signal is 0, the execution of the next block during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to 1.

NOTE
When blocks for cycle operation are internally created by a canned cycle and so on, only the first block is generally interlocked by this signal. The intermediate blocks are executed continuously even if this signal is set to 0.

Cutting block start interlock signal *CSL<G008#1>

[Classification] Input signal

[Function] This signal disables the start of blocks specifying move commands other than positioning during automatic operation.

[Operation] While this signal is 0, the execution of blocks specifying movement other than positioning during automatic operation is not started. This signal does not affect a block that has already started, and that block is continuously executed until its end. This signal does not halt automatic operation. The command in the next block is ready for execution as a valid command, so execution restarts as soon as the signal is set to 1.

[Usage] When the spindle has been specified, or when the spindle speed has been changed, this signal can be held 0 until a target spindle speed is achieved. Then, the next cutting block can be executed at the target spindle speed.

> **NOTE**
> This signal is effective for any blocks including blocks for cycle operation internally created by a canned cycle and so on.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007							STLK	
G008					*BSL		*CSL	*IT
G130					*IT4	*IT3	*IT2	*IT1
G132					+MIT4	+MIT3	+MIT2	+MIT1
G134					–MIT4	–MIT3	–MIT2	–MIT1
X004			–MIT2	+MIT2	–MIT1	+MIT1		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003				DAU	DIT	ITX		ITL
					DIT	ITX		ITL

[Data type] Bit

ITL Interlock signal
0 : Enabled
1 : Disabled

ITX Interlock signals for each axis
0 : Enabled
1 : Disabled

DIT Interlock for each axis direction
0 : Enabled
1 : Disabled

DAU If bit 3 (DIT) of parameter No. 3003 is set to 0, the interlock signal of each axial direction is:
0 : Enabled only in manual operation and disabled in automatic operation.
1 : Enabled in both manual operation and automatic operation.

	#7	#6	#5	#4	#3	#2	#1	#0
3004							BCY	BSL

[Data type] Bit

BSL The block start interlock signal *BSL and cutting block start interlock signal *CSL are:
0 : Disabled.
1 : Enabled.

BCY When more than one operation is performed by one block command such as a canned cycle, the block start interlock signal *BSL is:
0 : Checked only at the beginning of the first cycle.
1 : Checked at the beginning of every cycle.

Note

> **NOTE**
> The interlock signal for each axis and direction (T series) is supported regardless of whether the direct input of tool offset value measurement B is provided.

2.6 MODE SELECTION

General

The mode select signal is a code signal consisting of the three bits MD1, MD2, and MD4.

The following seven modes can be selected.

- Memory edit (EDIT)
- Memory operation (MEM)
- Manual data input (MDI)
- Manual handle/incremental feed (HANDLE/INC)
- Manual continuous feed (JOG)
- TEACH IN JOG
- TEACH IN HANDLE

And in addition, DNC operation mode can be selected by combining the (MEM) mode setting and the DNCI signal. Manual reference position return mode can be selected by combining the manual continuous feed (JOG) mode setting and the ZRN signal.
The currently selected operation mode can be posted by outputting the operation mode check signal.

Signal

Mode selection signal MDI, MD2, MD4 <G043#0 to #2> DNCI <G043#5> ZRN <G043#7>

[Classification] Input signal

[Operation] As shown in the following table, the mode select signal is a grey code (a code in which only one bit is different from that of the adjacent mode). To prevent faulty mode switching, use an overcrossing contact rotary switch so that only one bit changes from that of the adjacent mode.
An example of ”Faulty mode switching” would be:

When the mode is switched to the EDIT mode during memory operation, the CNC enters the single block state and the operation stops at the end of the executing block.

For this example mode switching, only MD2 should change from 0 to 1. However if a transient signal status change were to occur in a signal other than MD2 during mode switching, another mode (manual continuous feed mode, for example) would be set between automatic operation mode and memory edit mode. When manual continuous feed mode is set while the CNC is in automatic status, the CNC immediately stops memory operation. As a result, although the operator intended to switch the mode to the memory edit mode, the transient signal caused the CNC to be placed in feed hold state instead.

	Mode	Signal status				
		MD4	MD2	MD1	DNCI	ZRN
1	Memory edit (EDIT)	0	1	1	0	0
2	Memory operation (MEN)	0	0	1	0	0
3	Manual data input (MDI)	0	0	0	0	0
4	Manual handle/incremental feed (HANDLE/INC)	1	0	0	0	0
5	Manual continuous feed (JOG)	1	0	1	0	0
6	TEACH IN HANDLE (THND)	1	1	1	0	0
7	TEACH IN JOG (TJOG)	1	1	0	0	0
8	DNC operation (RMT)	0	0	1	1	0
9	Manual reference position return (REF)	1	0	1	0	1

Operation mode check signal MMDI, MMEM , MRMT, MEDT, MH, MINC, MJ, MREF, MTCHIN <F003, F004#6>

[Classification] Output signal

[Function] The currently selected operation mode is output.

[Operation] The following lists the relationship between the mode selection signals and check signals:

Mode		Input signal MD4	MD2	MD1	DNCI	ZRN	Output signal
Automatic operation	Manual data input (MDI) (MDI operation)	0	0	0	0	0	MMDI<F003#3>
	Memory operation (MEM)	0	0	1	0	0	MMEM<F003#5>
	DNC operation(RMT)	0	0	1	1	0	MRMT<F003#4>
Memory edition (EDIT)		0	1	1	0	0	MEDT<F003#6>
Manual operation	Manual handle feed / Incremental feed (HANDLE/INC)	1	0	0	0	0	MH<F003#1> MINC<F003#0>
	Manual continuous feed(JOG)	1	0	1	0	0	MJ<F003#2>
	Manual reference position return (REF)	1	0	1	0	1	MREF<F004#5>
	TEACH IN JOG (TJOG)	1	1	0	0	0	MTCHIN<F003#7>, MJ<F003#2>
	TEACH IN HANDLE (THND)	1	1	1	0	0	MTCHIN<F003#7>, MH<F003#1>

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN		DNCI			MD4	MD2	MD1

	#7	#6	#5	#4	#3	#2	#1	#0
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF					

Note

Precautions on modes and mode switching

NOTE

1 In MDI mode, the STL signal turns to “0” and the CNC stops as soon as the commands entered via the MDI have been executed. But the SPL signal does not turn to “1”. Therefore, another command can be entered from the manual data input unit under this condition.

2 Manual operation in jog feed mode
   - a) When bit 0 (JHD) of parameter No. 7100 is set to 0
     Only jog feed is possible.
   - b) When bit 0 (JHD) of parameter No. 7100 is set to 1
     Both jog feed and manual handle feed are possible, provided the manual handle feed is installed. Jog feed and manual handle feed cannot, however, be performed simultaneously. Manual handle feed can be performed when the tool is not being moved by means of jog feed.

3 Manual operation in manual handle/incremental feed and TEACH IN HANDLE mode.
   - a) Incremental feed is possible, provided the manual handle feed is not installed.
   - b) Incremental feed is possible, provided the manual handle feed is installed.
     - i) When bit 0 (JHD) of parameter No. 7100 is set to 0
       Only manual handle feed is possible.
     - ii) When bit 0 (JHD) of parameter No. 7100 is set to 1
       Manual handle feed and incremental feed is possible. Manual handle feed and incremental feed cannot, however, be performed simultaneously. Manual handle feed can be performed when the tool is not being moved by means of incremental feed.

NOTE

4 Manual operation in TEACH IN JOG mode
   a) When bit 1 (THD) of parameter No. 7100 is set to 0
      Only jog feed is possible.
   b) When bit 1 (THD) of parameter No. 7100 is set to 1
      Both jog feed and manual handle feed are possible, provided the manual handle feed option is installed. Jog feed and manual handle feed cannot, however, be performed simultaneously. Manual handle feed can be performed when the tool is not being moved by means of jog feed.

5 When switching to manual data input mode is made during operation in memory operation mode, the CNC enters the automatic operation stop state after executing the command in the current block. Signal STL is then set to 0. In this case, signal SPL is not set to 1 ((a) in Fig. 2.6). When switching to memory operation mode is made during operation in manual data input mode, the CNC enters memory operation mode after executing the currently executed command ((b) in Fig. 2.6).

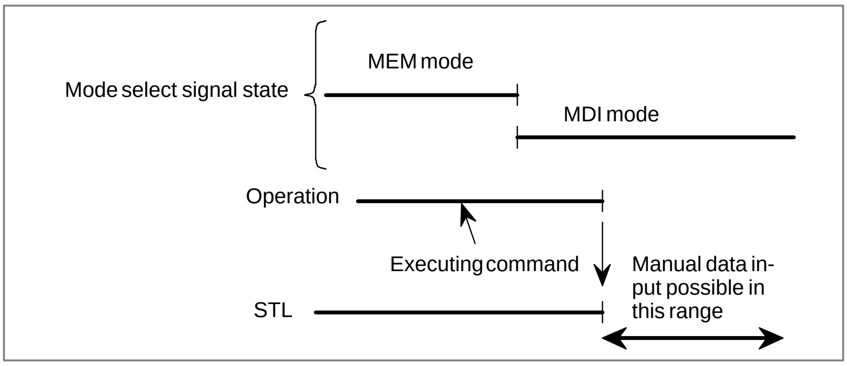


Fig. 2.6 (a)

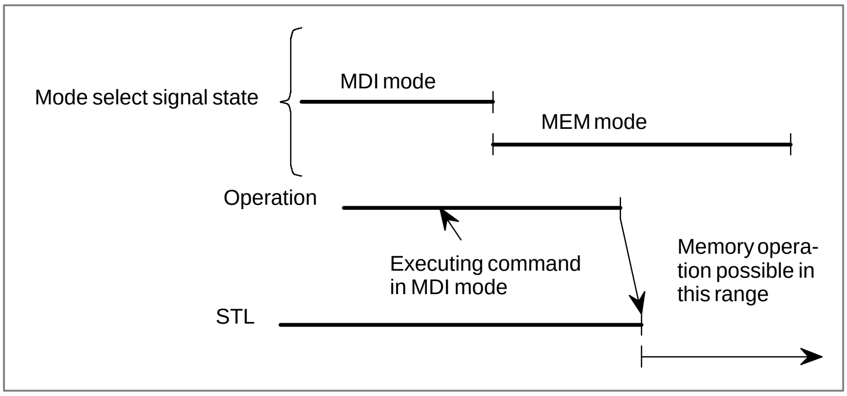


Fig. 2.6 (b)

NOTE

6 When the HANDLE/INC or TEACH IN HANDLE mode is selected while the CNC is operating in the MEM or MDI mode, the automatic or MDI operation stops, the STL signal turns to "0", the SPL signal simultaneously turns to "1", and the CNC enters the HANDLE/INC or TEACH IN HANDLE mode. Under these conditions, manual handle feed or incremental feed by axis direction select signal is permitted. Since the MEM mode or MDI mode commands are held, operation can be restarted by the cycle start signal and by selecting the MDI or MEM mode. However, if operation was stopped by switching to the HANDLE/INC or TEACH IN HANDLE mode during manual data input or during automatic operation, it can be restarted only by reactivating the mode in use before the operation was stopped (Fig. 2.6 (c)).

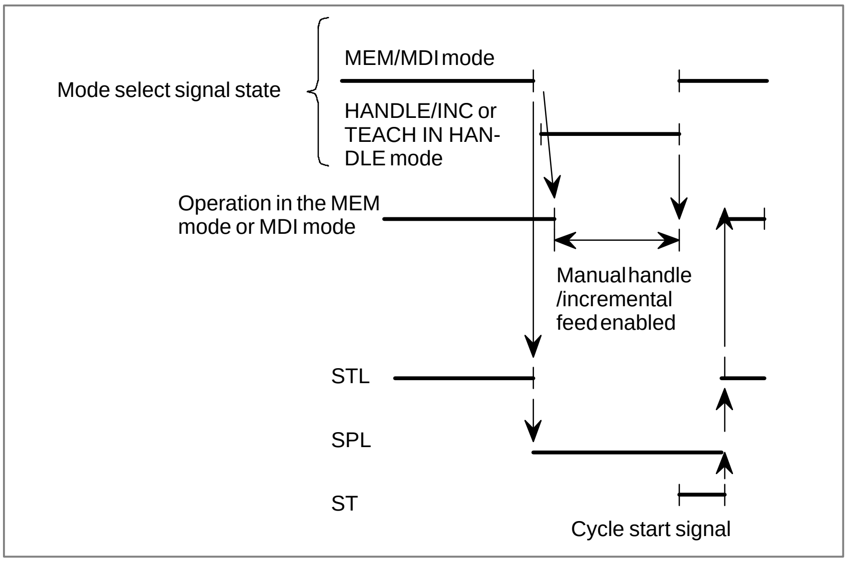


Fig. 2.6 (c)

NOTE

7 When the JOG or TEACH IN JOG mode is selected during RMT, MEM or MDI mode operation, operation stops, the STL signal turns to “0”, the SPL signal simultaneously turns to “1”, and the CNC enters the JOG or TEACH IN JOG mode. Under these conditions, manual feed by feed axis direction select signal is permitted. Operation can be restarted by returning to the original mode, as described for HANDLE/STEP or TEACH IN HANDLE mode (Fig. 2.6 (d)).
When the mode is switched to the JOG or TEACH IN JOG mode during manual handle feed or during step feed operation, the CNC ignores the manual handle feed or step feed command and manual jog feed becomes effective. If a feed axis direction select signal turns to “1” before the JOG or TEACH IN JOG mode is selected, that signal is ignored. The feed axis select signal is selected by turning the necessary feed axis direction signal to “1” after turning all the feed axis direction select signals to “0” (Fig. 2.6 (e)).
It is possible to perform handle feed in TEACH IN JOG mode by activating parameter THD no.7100#1. For details, refer to item (2), (4).

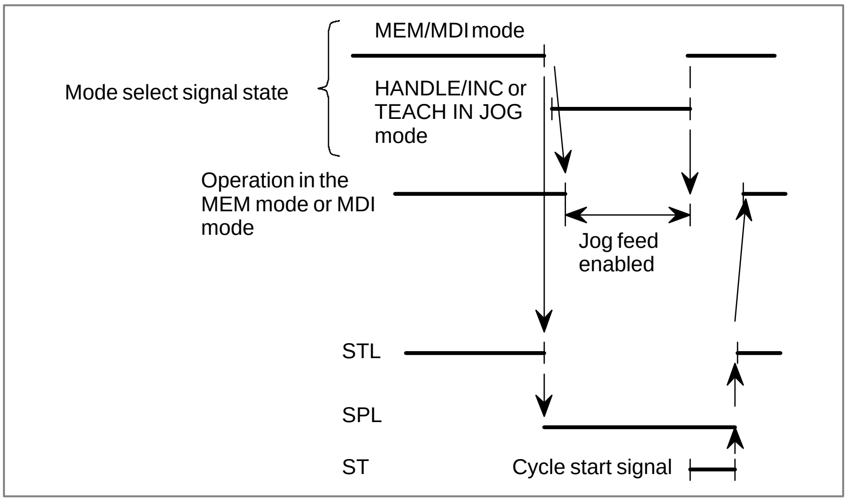


Fig. 2.6 (d)

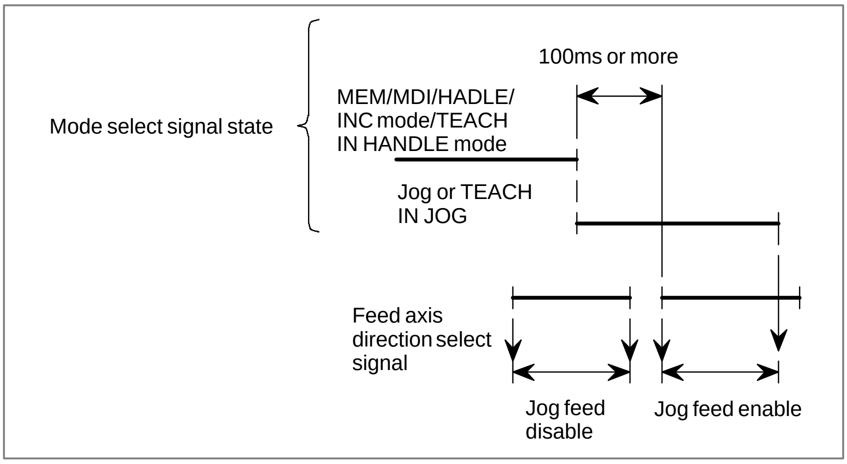


Fig. 2.6 (e)

> **NOTE**
> 8 The mode switching operation is summarized in the time chart below (Fig. 2.6 (f)).

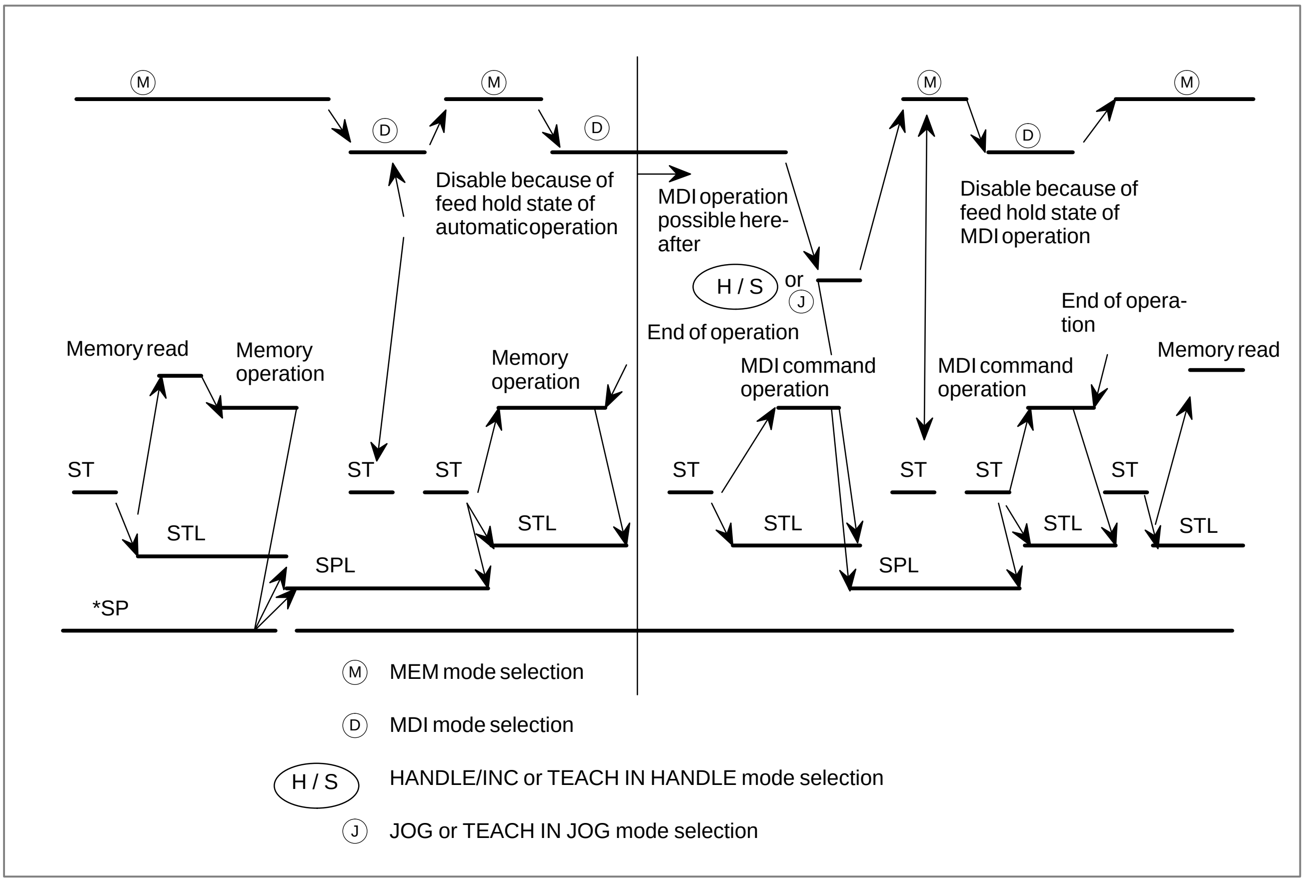


Fig. 2.6 (f) Mode signal time chart

Reference item

CONNECTION MANUAL (This manual)	4.1	Manual Reference Position Return

2.7 STATUS OUTPUT SIGNAL

General

The table below lists the status output signals. They indicate the state of the CNC. See the sections listed in the table for details of each signal.

Signal name	Symbol	Reference section
Alarm signal	AL	2.4
Battery alarm signal	BAL	2.4
Reset signal	RST	5.2
Rewinding signal	RWD	5.2
Tapping signal	TAP	11.6
Moving signal	MV1 – MV4	1.2.5
Moving direction signals	MVD1 – MVD4	1.2.5
In–position signals	INP1 – INP4	7.2.5.1
Rapid traversing signal	RPDO	2.7 (the section you are reading)
Cutting feed signal	CUT	2.7 (the section you are reading)
Thread cutting signal	THRD	6.4.1
Constant surface speed signal	CSS	9.4
Inch input signal	INCH	11.4

Signal

Rapid traversing signal RPDO <F002#1>

[Classification] Output signal

[Function] This signal indicates that a move command is being executed at rapid traverse.

[Output condition] “1” indicates an axis starts moving after rapid traverse has been selected; “0” indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

NOTE
1 Rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. Rapid traverse in manual operation also includes rapid traverse in reference position return.
2 Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement has been started.

Cutting feed signal CUT <F002#6>

[Classification] Output signal

[Function] Signals that cutting feed is being performed by automatic operation.

[Output condition] This signal is set to 1 in the following case:

– When cutting feed is being performed by automatic operation (cutting feed for linear interpolation, circular interpolation, helical interpolation, thread cutting, skip cutting, or cutting in canned cycle)

CAUTION
This signal is not set to "1" in the feed hold state.

NOTE
This signal is set to "1" even when the feedrate override is 0%, and even during interlock.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002		CUT					RPDO	

2.8 VRDY OFF ALARM IGNORE SIGNAL

General

The German VDE safety standard requires that the motor be deactivated when the safety guard is opened. By using the VRDY OFF Alarm Ignore signal, however, the CNC can be restarted without resetting, even if the safety guard has been opened.

Signal

All–Axis VRDY OFF Alarm Ignore Signal IGNVRY<G066#0>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, VRDY OFF, for all axes.

[Operation] When this signal is set to logical 1, the control unit operates as follows:

· The control unit does not issue servo alarm No. 401, VRDY OFF, even when the servo amplifier ready signal goes off. The control unit, however, sets servo ready signal SA to 0. The SA signal can remain set to 1, depending on the setting of SAK, bit 6 of parameter No. 1804.

Each–Axis VRDY OFF Alarm Ignore Signal IGVRY1 – IGVRY4 <G192#0–#3>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, VRDY OFF, for the corresponding axis. These signals correspond to the controlled axes. The suffixed number of each signal corresponds to the number of the controlled axis.

[Operation] When this signal is set to logical 1, the control unit operates as follows:

· The control unit does not issue servo alarm No. 401, VRDY OFF, even when the servo amplifier ready signal for the corresponding axis goes off. The servo ready signal SA, however, is set to 0. The SA signal, however, can remain set to 1 depending on the setting of SAK, bit 6 of parameter No. 1804.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066								IGNVRY
G192					IGVRY4	IGVRY3	IGVRY2	IGVRY1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1804		SAK						

[Data type] Bit

SAK When the VRDY OFF Alarm Ignore signal IGNVRY is 1, or when any of the VRDY OFF Alarm Ignore signals IGVRY1 to IGVRY4 are 1:

0 : Servo ready signal SA is set to 0.
1 : Servo ready signal SA remains set to 1.

Alarm and Message

Number	Message	Description
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–4) servo amplifier READY signal (DRDY) went off.

Caution

CAUTION

1. When the control enters NOT READY status due to emergency stop or a servo alarm and then the control is reset, reset processing is not terminated until the VRDY OFF alarm ignore signal is set to 0.
2. When the VRDY OFF alarm ignore signal is set to 1 and the servo amplifier ready signal is set to off, the motor is freed from the drive, but follow up is not performed. To perform follow up, set the servo off signal to 1.

Note

NOTE

While the VRDY OFF alarm ignore signal is set to 1, and a servo alarm other than alarm No. 401 occurs, the control unit detects the alarm.

2.9 ABNORMAL LOAD DETECTION

General

Machine collision, defective, and damaged cutters cause a large load torque on the servo and spindle motors, compared with normal rapid traverse or cutting feed. This function detects the load torque on the motors and sends this value as an estimated load torque to the PMC. If the detected load torque value is abnormally great compared with the value specified in the parameter, the function stops the servo motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize possible damage to the machine. (The function to reverse motors is effective only for servo motors.)

The abnormal load detection function is further divided as follows:

(1) Estimated load torque output function

The CNC is always calculating the estimated load torque for the motor (excluding acceleration/deceleration torque). The estimated load torque output function enables the PMC to read the calculated torque using the window function.

(2) Abnormal load detection alarm function

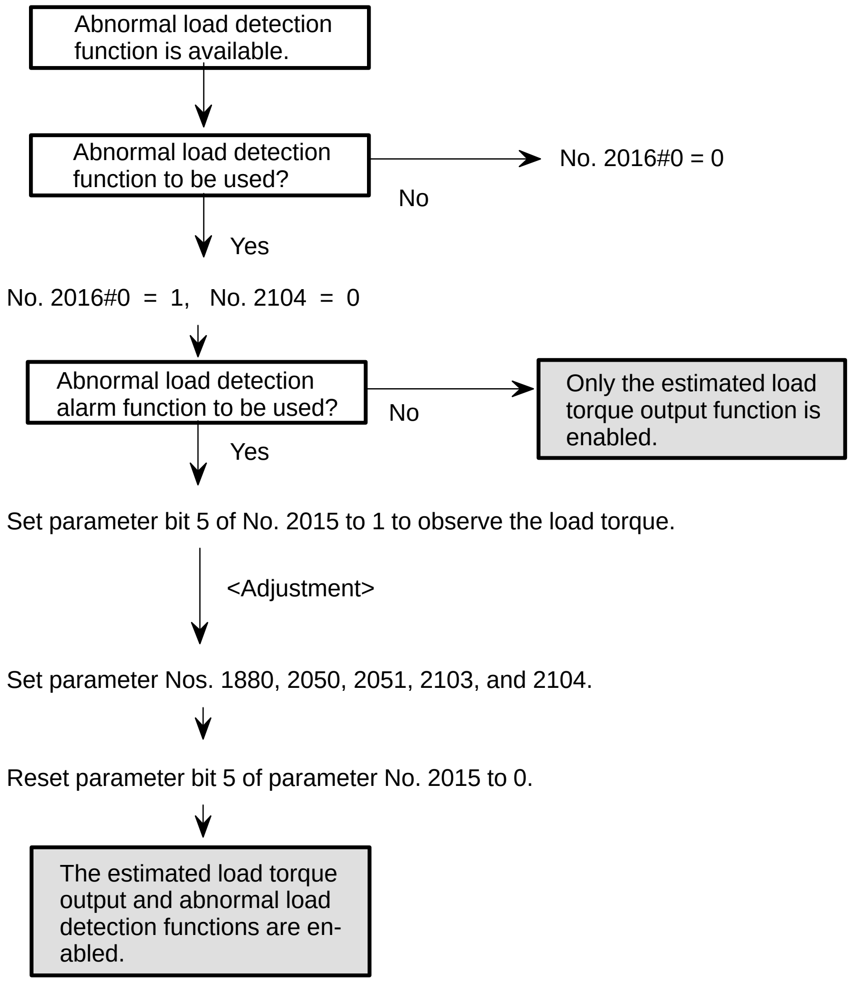
This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, whenever the load torque obtained by the estimated load torque output function is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)

Abnormal load detection can also be disabled only for specific axes by using bit 5 (ABDSW) of parameter No. 2215 for the abnormal load detection function and abnormal load detection ignore signals IUDD1 to IUDD4 <G0125>. (This function is effective only for servo motors.)

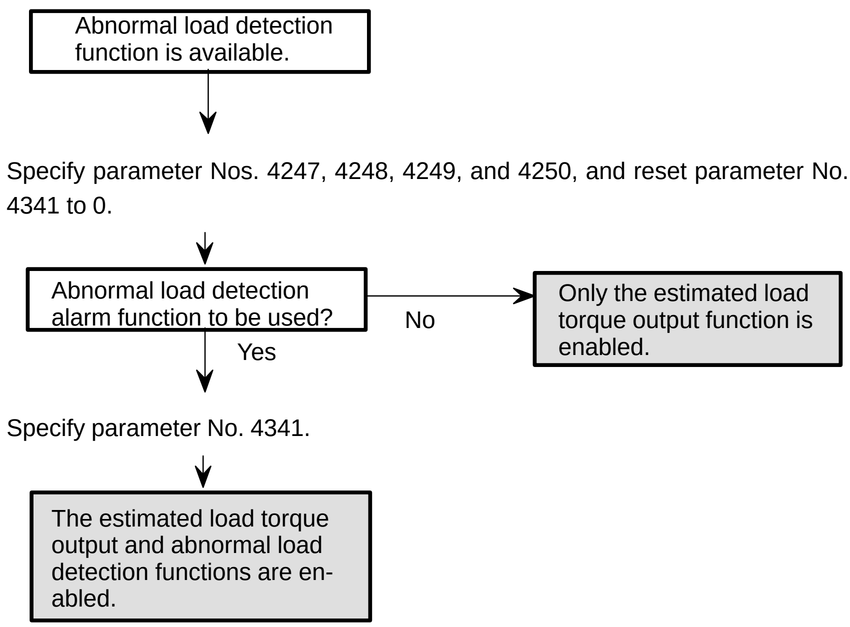
- **Parameter setting**

The following flowcharts explain how to specify parameters for the abnormal load detection function.

(1) Servo axis



(2) Spindle



Signal

Servo axis abnormal load detected signal ABTQSV <F090#0>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on a servo axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for a servo axis, Cs axis, spindle positioning axis, or spindle axis during rigid tapping.

First–spindle abnormal load detected signal ABTSP1 <F090#1>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the first spindle axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for the first spindle under speed control.

Second–spindle abnormal load detected signal ABTSP2 <F090#2>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the second spindle axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for the second spindle under speed control.

The following list summarizes the alarms and signals output by each function.

	Signal output		Alarm	
	ABTQSV	ABTSP1/ ABTSP2	409	754/764
Servo axis	○	—	○	—
Cs contour control	○	—	○	—
Spindle positioning axis	○	—	○	—
Rigid tapping	○	—	—	○
Spindle axis for speed control	—	○	—	○

Abnormal load detection ignore signal IUDD1 to IUDD4 <G0125#0–#3>

[Classification] Input signal

[Function] These signals disable the abnormal load detection function for corresponding axes. These signals correspond to the controlled axes. The suffixed number of each signal corresponds to the number of a controlled axis.

[Output condition] When the signal is set to 1, abnormal load detection is not performed for the corresponding axis.
When the signal is used, bit 5 (ABDSW) of parameter No. 2215 for the abnormal load detection function must be set to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0090						ABTSP2	ABTSP1	ABTQSV

	#7	#6	#5	#4	#3	#2	#1	#0
G0125					IUDD4	IUDD3	IUDD2	IUDD1

Parameter

(1) Parameter common to servo axes and spindles

1880	Timer for abnormal load detection alarm

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 32767
(If 0 is set, 200 ms is assumed.)

This parameter specifies the interval between the detection of an abnormal load and the issuance of a servo alarm. When the set value is not a multiple of eight, it is rounded up to the nearest multiple of eight.

[Example] When 30 is set, the system assumes 32 ms.

(2) Servo axis parameters

	#7	#6	#5	#4	#3	#2	#1	#0
2015			TDOUT					

[Data type] Bit axis

TDOUT Select output to the check board (for each axis)

0 : Output the torque command to the check board.
1 : Output the estimated load torque to the check board.

	#7	#6	#5	#4	#3	#2	#1	#0
2016								ABNTDT

[Data type] Bit axis

ABNTDT Output of the estimated load torque (for each axis)

0 : Disabled
1 : Enabled

This parameter must be specified when using the estimated load torque output function or the abnormal load detection alarm function.

2050	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3559

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 956 in this parameter.

2051	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3329

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 510 in this parameter.

2103	Retraction distance upon the detection of an abnormal load

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies the distance by which the tool is to be retracted, by reversing the motor, if an abnormal load is detected. When the motor is rotating at low speed, however, the tool may be retracted too far. To prevent this, the motor is stopped, instead of being reversed, while the specified feedrate is less than the value listed in the table below.

When this parameter is set to value A, the detection of an abnormal load causes the tool to be retracted in the reverse direction by the distance A, then stopped, if the specified feedrate is equal to or greater than the value listed below.

Detection unit	Feedrate
1μ	A / 8 mm/ min
0.1μ	A / 80 mm/ min

When this parameter is set to 0, the motor stops immediately upon the detection of abnormal load.

2104	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] Torque command unit (Refer to the digital servo operator's manual for details.)

[Valid data range] 0 to 7282
(The maximum motor torque is 7282, regardless of the motor type.)

This parameter specifies the threshold load torque at which an abnormal load detection alarm is issued.

Monitor the load torque by setting bit 5 of parameter No. 2015 to 1 then, for this parameter (No. 2104), set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
2215			ABDSW					

[Data type] Bit axis

ABDSW The abnormal load detection function for a specified axis:
0 : Cannot be disabled.
1 : Can be disabled.

(3) Spindle parameters

	#7	#6	#5	#4	#3	#2	#1	#0
4015							SPLDMT	

[Data type] Bit axis

SPLDMT Spindle load torque monitor function
0 : The spindle load torque monitor function is disabled.
1 : The spindle load torque monitor function is enabled.

4247	Magnetic flux compensation time constant for spindle load torque monitor

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 8192

[Standard setting] Depends of the motor model.

This parameter is used to compensate the delay in the generation of magnetic flux in the spindle motor relative to the specified value. When 0 is set, it is assumed that the generation of magnetic flux is not delayed.

4248	Spindle load torque monitor constant

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] Depends of the motor model.

This constant is determined by the maximum output torque and inertia of the motor. It is used for observer processing.

4249	Observer gain 1 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4250	Observer gain 2 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4341	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] 0.01 %

[Valid data range] 0 to 10000

This parameter specifies the threshold load torque at which an abnormal load detection alarm is issued for the spindle. Set a percentage (in units of 0.01 %) for the maximum output torque of the motor. When 0 is set, no abnormal load detection alarm is issued for the spindle.

Alarm and message

(1) Servo axis

Number	Message	Description
409	Servo alarm: Abnormal load detected on axis n	An abnormal load was detected on a servo motor, or on a spindle motor during Cs mode. To release the alarm, use RESET.

(2) Spindle

Number	Message	Description
754	Abnormal load detected on the first spindle	An abnormal load was detected on the first spindle motor. To release the alarm, use RESET.
764	Abnormal load detected on the second spindle	An abnormal load was detected on the second spindle motor. To release the alarm, use RESET.

PMC window function

- **Reading the load torque data**

The load torque data can be read at the PMC using its window function.

(1) Servo axis

[Input data structure]

Address	Content
Top address +0	(Function code) 211
2	(Completion code) — (Not to be set)
4	(Data length) — (Not to be set)
6	(Data number) 0
8	(Data attribute) M (M = 1 to n or –1)
10	(Data area) — (Not to be set)
42	

(M = 1 to n: Specifies separately for each axis whether data for the axis is to be read; n is an axis number.

M = –1: Specifies that data for all axes are to be read.

[Completion code]

0: The load torque data was read normally.

4: Invalid data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than the number of controllable axes was specified.

[Output data structure]

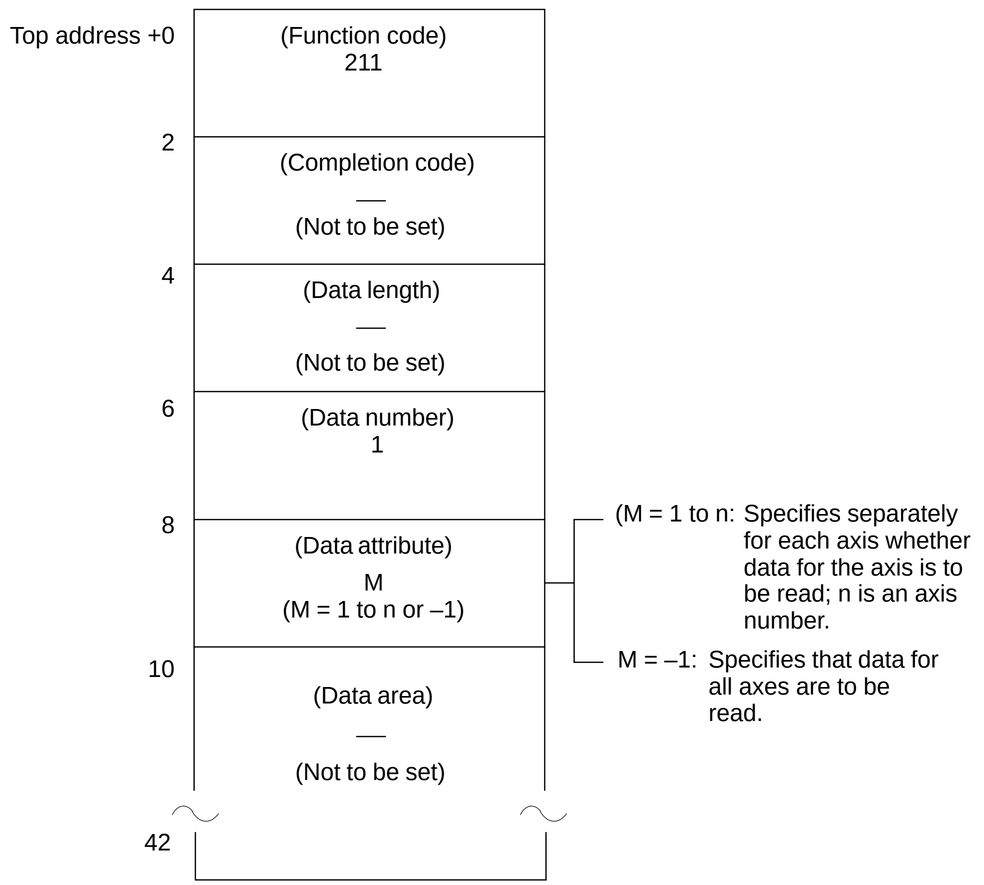
Top address + 0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 0	
8	(Data attribute) M (M: Data at input)	Description of value
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are four controllable axes

		Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	
14	Load torque for the third axis (2 bytes)	
16	Load torque for the fourth axis (2 bytes)	

(2) Spindle

[Input data structure]



[Completion code]

0: The load torque data was read normally.

4: Invalid data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than controllable axes was specified.

[Output data structure]

Offset	Content	Description of value
Top address + 0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 1	
8	(Data attribute) M (M: Data at input)	
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are two controllable axes

Offset	Content	Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	

2.10 SERVO SPEED CHECK

General

While the servo speed check enable signal (G0349) is set to 1, the CNC checks the servo speed and compares the servo speed with the value set in parameter No. 12291. If the servo speed is lower than the value set in the parameter, the servo under–speed alarm signal (F0349) is output on an axis–by–axis basis. At this time, the servo alarm (616) is issued if bit 1 (SSA) of parameter No. 12290 is set to 1.

Notes

- This function is usable with a servo axis.
- If this function is used during simple synchronous control, the speed of the master axis is compared with the speed set for the master axis, and the speed of the slave axis is compared with the speed set for the slave axis. So, when this function is used during simple synchronous control, the speed (No. 12291) and the servo speed check enable signal (G0349) need to be set in the same way.
- While the servo speed check enable signal (G0349) is input, this function makes a speed check at all times. So, even if the speed decreases below the allowable level due to deceleration, for example, at the end of a block, the servo under–speed alarm signal (F0349) is output (together with a servo alarm, depending on parameter setting). To prevent this, measures for disabling servo speed checking at a joint of blocks need to be taken.

Signal

Servo speed check enable signal SVSCK1 to SVSCK8 <G0349>

[Classification] Input signal (G)

[Function] Enables the servo speed check function for each axis.

When this signal is set to 1, the servo speed is monitored.
When this signal is set to 0, the servo speed is not monitored.

Servo under–speed alarm signal TSA1 to TSA8 <F0349>

[Classification] Output signal (F)

[Function] This signal is set to 1 when the servo speed is lower than the value set in parameter No. 12291 and the servo speed check enable signal (G0349) is set to 1.

This signal is set to 0 when the servo speed is higher than the value set in parameter No. 12291 or when the servo speed is lower than the set value but the servo speed check enable signal is not set to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0349	SVSCK8	SVSCK7	SVSCK6	SVSCK5	SVSCK4	SVSCK3	SVSCK2	SVSCK1

	#7	#6	#5	#4	#3	#2	#1	#0
F0349	TSA8	TSA7	TSA6	TSA5	TSA4	TSA3	TSA2	TSA1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
12290							SSA	SSC

[Data type]

SSC Servo speed checking is:

0 : Disabled.
1 : Enabled.

SSA When the actual servo speed is lower than the speed set in parameter No. 12061:

0 : No alarm is issued.
1 : An alarm is issued.

12291	Reference speed for servo speed checking

[Data type] Word axis

[Unit of data] rpm

[Valid data range] 0 to 8000

NOTE
This parameter is valid when bit 0 (SSC) of parameter No. 12290 is set to 1.

Alarm and Message

Number	Message	Description
SV616	SERVO SPEED TOO SMALL	A servo speed check finds that the actual speed has not reached the value set in parameter No. 12291. (This alarm is issued only when bit 1 of parameter No. 12290 is set to 1.)

3 MANUAL OPERATION

3.1 JOG FEED/ INCREMENTAL FEED

General

- **Jog feed**

In jog mode, setting a feed axis and direction selection bit to “1” on the machine operator’s panel moves the tool along the selected axis in the selected direction.
Manual operation is allowed one axis at a time. 3 axes can be selected at a time by setting parameter JAX (No.1002#0).

- **Incremental feed**

In incremental feed mode, setting a feed axis and direction selection bit to “1” on the machine operator’s panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved, is the least input increment. The step can be 10, 100, or 1000 times the least input increment.

The jog feedrate is specified in parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
With rapid traverse selected, the tool can be moved at the rapid traverse rate regardless of the jog feedrate override signal.

Signal

The following signals determine that way the jog feed or incremental feed is executed.

<table>
<tr><th>Selection</th><th>Jog feed</th><th>Incremental feed</th></tr>
<tr><td>Mode selection</td><td>MD1, MD2, MD4, MJ</td><td>MD1, MD2, MD4, MINC</td></tr>
<tr><td>Selection of the axis to move</td><td colspan="2" rowspan="2">+J1, –J1, +J2, –J2, +J3, –J3, ...</td></tr>
<tr><td>Selection of the direction to move the axis</td></tr>
<tr><td>Selection of the move amount</td><td></td><td>MP1, MP2</td></tr>
<tr><td>Selection of feedrate</td><td colspan="2">*JV0 – *JV15, RT, ROV1, ROV2</td></tr>
</table>

The only difference between jog feed and incremental feed is the method of selecting the feed distance. In jog feed, the tool continues to be fed while the following signals selecting the feed axis and direction are “1”: +J1, –J1, +J2, –J2, +J3, –J3, etc. In incremental feed, the tool is fed by one step.
The distance of the step is selected by the manual handle feed move distance select signals MP1 and MP2.
For the signals selecting the mode, see Section 2.6, “Mode Selection Signals.” For the manual handle feed selection signals, MP1 and MP2 select the move amount, see 3.2 “Manual handle feed.” For rapid traverse override signals ROV1 and ROV2, see Section 7.1.6.1, (Feedrate Override Signals).
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 – +J4<G100#0–#3> –J1 – –J4<G102#0–#3>

[Classification] Input signal

[Function] Selects a desired feed axis and direction in jog feed or incremental feed. The sign (+ or –) in the signal name indicates the feed direction. The number following J indicates the number of the control axis.

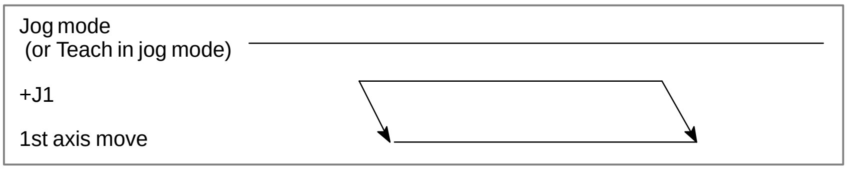
+ J 1

1 First axis is fed
2 Second axis is fed
3 Third axis is fed
4 Fourth axis is fed

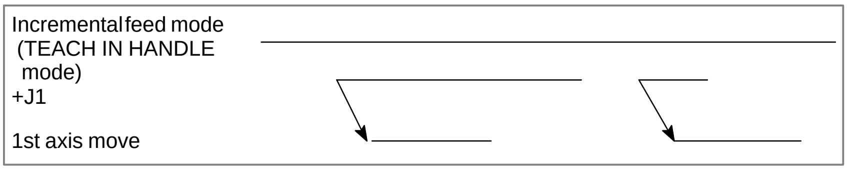
\+ Feed in positive direction
– Feed in negative direction

[Operation] When the jog bit is “1”, the control unit operates as described below.

- When jog feed or incremental feed is allowed, the control unit moves the specified axis in the specified direction.
- In jog feed, the control unit continues to feed the axis while the bit is “1”.

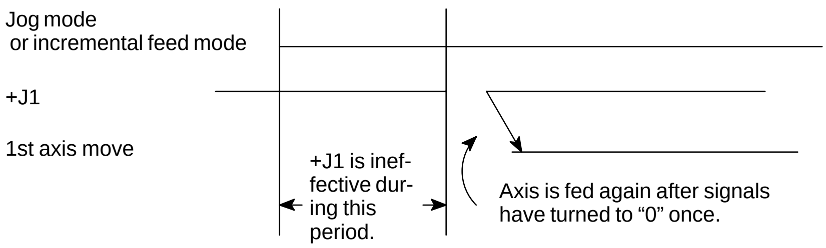


- In incremental feed, the control unit feeds the requested axis by the step distance which is specified by the manual handle feed move distance selection signals MP1, MP2, then the axis stops. Even if the signal is set to “0” while the axis is being fed, the control unit does not stop moving.
  To feed the axis again, set the signal to “0”, then to “1” again.

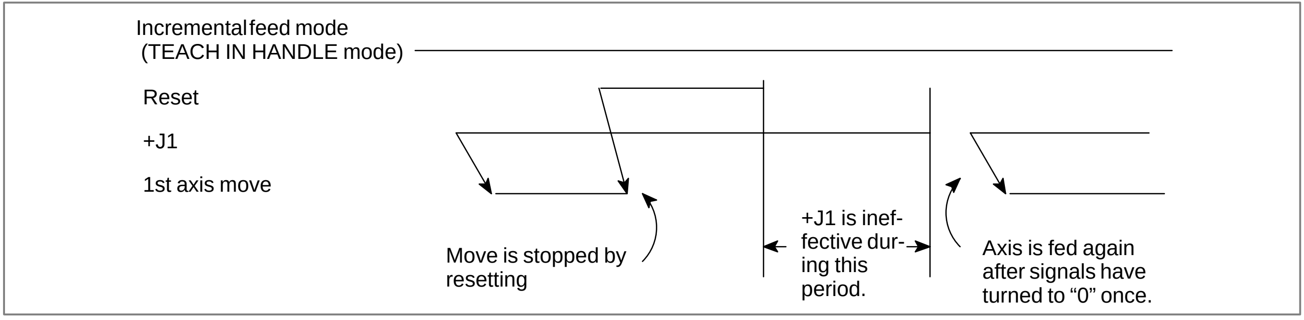


NOTE

1. If both the positive direction and negative direction signals of the same axis are simultaneously set to “1”, neither the positive direction nor the negative direction is selected. The control unit assumes that both these signals are set to “0”.
2. If the feed axis and direction selection signals are set to “1” before the jog feed mode or incremental feed mode is selected, these signals are invalidated. After the jog feed mode or incremental feed mode is selected, set these signal to “0”, then set them to “1” again.



3. If the control unit is reset while the feed axis and direction selection signals are set to “1” or if a feed axis and direction signal turns to “1” while the control unit is in the reset state, the signal will be ignored even after releasing reset. After the reset state is released, set these signals to “0”, then set them to “1” again.



Manual Feedrate Override Signal *JV0 – *JV15 <G010, G011>

[Classification] Input signal

[Function] Selects a feedrate in jog feed or incremental feed. These signals are in sixteen bit binary code, which corresponds to the override values as follows:

$$\text{Override value (\%)} = 0.01\% \times \sum_{i=0}^{15} |\, 2^{i} \times Vi \,|$$

where
Vi = 0 when the *JVi signal is “1”
Vi = 1 when the *JVi signal is “0”

The override value is assumed to be zero when all of the signals, (*JV0 to *JV15) are set to “1” or “0”. When this occurs, the feed is stopped. The override value can be specified in the range of 0% to 655.34% in units of 0.01%. Some examples are listed below.

*JV0 – *JV15	12 … 8	… 4	… 0	Override value (%)
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 0	0.01
1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.10
1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.00
1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.00
1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.00
0 1 1 0	0 0 1 1	1 0 1 1	1 1 1 1	400.00
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 1	655.34
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 0	0

[Operation] If rapid traverse selection signal RT is “0” during jog feed or incremental feed, the manual feedrate specified by parameter (No. 1423) is overridden by the value specified by the JVi signal.

NOTE
The JVi signals also serve as the override signals during dry run in automatic operation mode.

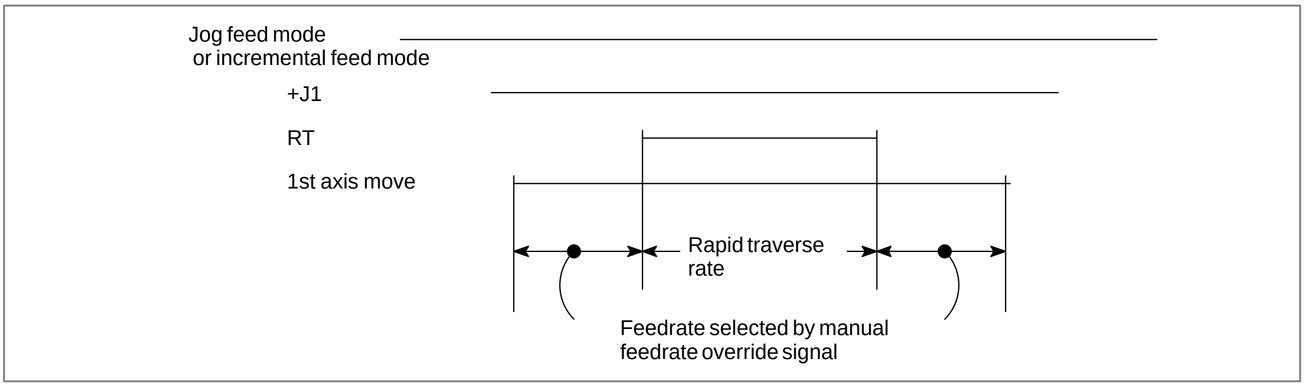
Manual rapid traverse selection signal RT<G019#7>

[Classification] Input signal

[Function] Selects a rapid traverse rate for jog feed or incremental feed.

[Operation] When the signal turns to “1”, the control unit operates as described below:

- The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is validated.
- When the signal is switched from “1” to “0” or vice versa during jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept “1”.



WARNING
After the power is turned on, the stroke limit function does not work until the reference position return is completed. During this period, the control unit ignores the RT signal, if it is set to “1”, and keeps moving the tool at a feedrate selected by the manual feedrate override signal. The parameter RPD (No. 1401#0) can be set so the rapid traverse is validated before the reference position return is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G019	RT							
G100					+J4	+J3	+J2	+J1
G102					–J4	–J3	–J2	–J1

Parameter

1002	#7	#6	#5	#4	#3	#2	#1	#0
								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

1401	#7	#6	#5	#4	#3	#2	#1	#0
								RPD

[Data type] Bit

RPD Manual rapid traverse during the period from power–on time to the completion of the reference position return.

0 : Disabled (Jog feed is performed.)
1 : Enabled

1402	#7	#6	#5	#4	#3	#2	#1	#0
				JRV				

[Data type] Bit

JRV Manual continuous feed (jog feed)

0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

> **NOTE**
> Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Data type] Word axis

(1) In M series, or in T series when JRV, bit 4 of parameter No. 1402, is set to 0 (feed per minute), specify a jog feedrate at feed per minute with an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	6 – 32767
Inch machine	0.1 inch/min	
Rotation axis	1 deg/min	

(2) When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a jog feedrate (feed per revolution) under an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 – 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

NOTE
If 0 is set, the rate set in parameter 1420 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				

[Data type] Bit axis

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1624	Time constant of exponential acceleration/deceleration or linear acceleration/ deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 4000 (for exponential acceleration/deceleration)
0 to 512 (for linear acceleration/deceleration after interporation or bell–shaped acceleration/deceleration)

Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation in jog feed for each axis.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1901				RFD				

[Data type] Bit

RFD In jog feed mode, the fine acceleration/deceleration function and feed–forward function are:

0: Disabled.
1: Enabled.

> **NOTE**
> 1 The axis operating under PMC axis control are not affected by this parameter. For such an axis, the settings for PMC axis control are followed. To enable the fine acceleration/deceleration function and feed–forward function in PMC axis control, advanced preview control for the PMC–controlled axis must be enabled. (See the descriptions of bit 3 (G8C) of parameter No. 8004 and bit 4 (G8R) of parameter No. 8004.)
> 2 Note that when the abnormal load detection function for cutting and rapid traverse is used, setting this parameter changes the threshold value
> (0: Threshold value for rapid traverse,
> 1: Threshold value for cutting feed).

Warning

> **WARNING**
> For incremental feeding along an axis under diameter programming, the tool moves in units of the diameter.

Note

> **NOTE**
> 1 Time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.
> 2 If a manual pulse generator is provided, the manual handle feed mode is enabled instead of incremental feed mode. However, using parameter JHD (bit 0 of parameter No. 7100) enables both manual handle and incremental feed in the manual handle feed mode.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED

3.2 MANUAL HANDLE FEED

General

In manual handle feed mode, the tool can be incrementally moved by rotating the manual pulse generator. Select the axis along which the tool is to be moved with the handle feed axis selection signal.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 7113 and 7114).
Arbitrary magnifications can be set for each axis by using the parameters (No. 12350 and No. 12351).
The handle magnifications can be selected by the manual handle feed move distance selection signal.

The number of manual pulse generators is as follows.

(M series)
– Up to three generators

(T series)
– Up to two generators

• Availability of manual handle feed in Jog mode

Parameter JHD (bit 0 of No. 7100) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

• Availability of manual handle feed in TEACH IN JOG mode

Parameter THD (bit 1 of No. 7100) enables or disables the manual handle feed generator in the TEACH IN JOG mode.

• A command to the MPG exceeding rapid traverse rate

Parameter (No. 7117) specifies as follows:

SET VALUE 0: The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

Other than 0: The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

• Movement direction of an axis to the rotation of MPG

Parameter HNGx (No. 7102#0) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

• Setting of magnifications for each axis

Magnifications m and n for manual handle feed can be set for each axis by setting an arbitrary magnification in parameter Nos. 12350 and 12351.
If no value is set in parameter No. 12350, parameter No. 7113 is used.
If no value is set in parameter No. 12351, parameter No. 7114 is used.
For manual handle interrupts as well, the parameters above are valid.

Signal

Manual Handle Feed Axis Selection Signals

- **(M series)**
  HS1A – HS1D <G018#0 – #3>
  HS2A – HS2D <G018#4 – #7>
  HS3A – HS3D <G019#0 – #3>

- **(T series)**
  HS1A – HS1D <G018#0 – #3>
  HS2A – HS2D <G018#4 – #7>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of four code signals, A, B, C, and D is provided for each manual pulse generator. (Up to three generators can be used.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS 1 A

1 First manual pulse generator
2 Second manual pulse generator
3 Third manual pulse generator (M series)

Code signals A, B, C, and D correspond to the feed axes as listed in the following table:

Manual handle feed axis selection				Feed axis
HSnD	HSnC	HSnB	HSnA	
0	0	0	0	No selection (None of axis is fed)
0	0	0	1	1st axis
0	0	1	0	2nd axis
0	0	1	1	3rd axis
0	1	0	0	4th axis

Manual Handle Feed Amount Selection Signal MP1, MP2<G019#4, 5> (Incremental Feed Signal)

[Classification] Input signal

[Function] This signal selects the distance traveled per pulse from the manual pulse generator during the manual handle feed or manual handle interrupt. It also selects the distance traveled per incremental feed step.
The table below lists the signal–to–distance correspondence.

Travel distance select signal for manual handle feed		Distance traveled		
MP2	MP1	Manual handle feed	Manual handle interrupt	Incremental feed
0	0	Least input increment × 1	Least command increment × 1	Least input increment × 1
0	1	Least input increment × 10	Least command increment × 10	Least input increment × 10
1	0	Least input increment × m*1	Least command increment × m*1	Least input increment × 100
1	1	Least input increment × n*1	Least command increment × n*1	Least input increment × 1000

*1 Scale factors m and n are specified using parameter Nos. 7113, 7114, 12350, and 12351.

WARNING

1 Because the least input increment is used as the units for manual handle and incremental feed, the same value represents a different distance depending on whether the metric or inch input system is used.
2 For an axis under diameter programming, the tool moves by the diameter value.
3 For the function indicated below, magnifications for manual handle feed cannot be set on an axis–by–axis basis. The magnifications, m and n, common to all axes set in parameter Nos. 7113 and 7114 are applied.
   • Handle pulse output function

NOTE

See Section 3.3, “Manual Handle Interrupt” for manual handle interrupts, and Section 3.1, “Jog Feed/Incremental Feed” for incremental feed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019			MP2	MP1	HS3D	HS3C	HS3B	HS3A

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF			THD	JHD

[Data type] Bit

JHD Manual handle feed in JOG mode or incremental feed in the manual handle feed

0: Invalid
1: Valid

THD Manual pulse generator in TEACH IN JOG mode

0: Invalid
1: Valid

HPF When a manual handle feed exceeding the rapid traverse rate is issued,

0: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are ignored. (The graduations of the manual pulse generator may not agree with the distance the machine has traveled.)
1: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are not ignored, but stored in the CNC. (If the rotation of the manual pulse generator is stopped, the machine moves by the distance corresponding to the pulses preserved in the CNC, then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Data type] Bit axis

HNGx Axis movement direction for rotation direction of manual pulse generator

0: Same in direction
1: Reverse in direction

7110	Number of manual pulse generators used

[Data type] Byte

[Valid data range] 1, 2, or 3

This parameter sets the number of manual pulse generators.
Varid data range is below:
1, 2 (T series)
1, 2, 3 (M series)

7113	Manual handle feed magnification m

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 127

This parameter sets the magnification when manual handle feed movement selection signal MP2 is on.

7114	Manual handle feed magnification n

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 1000

This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are “1”.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Data type] 2–word

[Unit of data] Pulses

[Valid data range] 0 to 99999999

If manual handle feed is specified such that the rapid traverse rate will be momentarily exceeded, those pulses received from the manual pulse generator that exceed the rapid traverse rate are accumulated rather than canceled. This parameter sets the maximum number of pulses which can be accumulated in such a case.

12350	Manual handle feed magnification m (in each axis)

[Data type] Word axis

[Unit of data] 1

[Valid data range] 1 to 127

This parameter sets the magnification to be used for each axis when manual handle feed movement amount select signal MP1 <G019#4> is set to 0, and MP2 <G019#5> is set to 1.
If this parameter is set to 0 for a target axis for movement, the setting of parameter No. 7113 applies.

12351	Manual handle feed magnification n (in each axis)

[Data type] Word axis

[Unit of data] 1

[Valid data range] 1 to 1000

This parameter sets the magnification to be used for each axis when manual handle feed movement amount select signal MP1 <G019#4> is set to 1, and MP2 <G019#5> is set to 1.
If this parameter is set to 0 for a target axis for movement, the setting of parameter No. 7114 applies.

Movement amount select signal		Movement amount (manual handle feed or manual handle interrupt)
MP2	MP1	
0	0	Least input increment × 1
0	1	Least input increment × 10
1	0	Least input increment × m (*1)
1	1	Least input increment × n (*2)

*1 When 0 is set in parameter No. 12350 for setting a movement target axis magnification, parameter No. 7113 is used.

*2 When 0 is set in parameter No. 12351 for setting a movement target axis magnification, parameter No. 7114 is used.

Warning

WARNING
Rotating the handle quickly with a large magnification such as x100 moves the tool too fast or the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator. The feedrate is clamped at the rapid traverse rate.

Caution

CAUTION
Rotate the manual pulse generator at a rate of five rotations per second or lower.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.3.4	MANUAL HANDLE FEED
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.3.4	MANUAL HANDLE FEED
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.3.4	MANUAL HANDLE FEED
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.3.4	MANUAL HANDLE FEED

3.3 MANUAL HANDLE INTERRUPTION

General

Rotating the manual pulse generator during automatic operation can increase the distance traveled by the amount corresponding to the handle feed. The axis to which the handle interrupt is applied is selected using the manual handle interrupt axis select signal.
The minimum travel distance per graduation is the least command increment. The minimum travel distance can be increased by tenfold or by two scale factors (parameter Nos. 7113 and 7114). Arbitrary magnifications can be set for each axis by using the parameters (No. 12350 and No. 12351). Each scale factor can be selected using the manual handle travel distance select signal (Section 3.2, ”Manual Handle Feed”).

Signal

Manual Handle Interrupt Axis Selection Signal

- **(M series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>
  HS3IA – HS3ID
  <G042#0 – #3>

- **(T series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied. There are three sets of signals, each corresponding to a manual pulse generator (up to three). Each set consists of four code signals A, B, C, and D. The number in each signal name corresponds to the number (position) of the manual pulse generator.

HS 1 IA

1 Selects the axis for which manual pulse generator No. 1 is used
2 Selects the axis for which manual pulse generator No. 2 is used
3 Selects the axis for which manual pulse generator No. 3 is used (M series)

The correspondence between the code signals and the selected feed axis is similar to the correspondence of the manual handle feed axis select signals. See Section 3.2, “Manual Handle Feed.”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042					HS3ID	HS3IC	HS3IB	HS3IA

Warning

WARNING
The distance travelled by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN).
Since the movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.

Note

NOTE
1 No handle interrupt can be used in manual handle feed mode.
2 Handle interruption is disabled when the machine is locked or interlocked.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.4.7	MANUAL HANDLE INTERRUPTION
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.4.7	MANUAL HANDLE INTERRUPTION
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.4.7	MANUAL HANDLE INTERRUPTION
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.4.7	MANUAL HANDLE INTERRUPTION

4 REFERENCE POSITION ESTABLISHMENT

4.1 MANUAL REFERENCE POSITION RETURN

General

The tool is moved in the direction specified by parameter ZMI (bit 5 of No. 1006) setting the feed axis and direction select signal to “1” during manual reference position return mode. Movement will continue until the reference position is obtained.

Manual reference position return is performed by using a grid method. The reference position is based on an electrical grid, using on one–rotation signals received from the position detector.

The following signals relate with the manual reference position return:

	Manual Reference Position Return
Mode selection	MD1, MD2, MD4
Selection of reference position return	ZRN, MREF
Selection of axis to be moved	+J1, –J1, +J2, –J2, +J3, –J3, ...
Selection of direction to be moved	
Selection of speed to be moved	ROV1, ROV2
Deceleration signal for reference position return	*DEC1, *DEC2, *DEC3, ...
Completion signal for reference position return	ZP1, ZP2, ZP3, ...
Reference position establishment signal	ZRF1, ZRF2, ZRF3, ...

Basic Procedure for Manual Reference Position Return

(1) Select JOG mode or TEACH IN JOG mode, and the manual reference position return selection signal ZRN to “1”.

(2) Feed a target axis toward the reference position by setting an appropriate feed axis and direction selection signal (+J1, –J1, +J2, –J2,…) “1”.

(3) While the feed axis and direction selection signal is “1”, rapid traverse takes place along that axis. Although the rapid traverse override signals (ROV1, ROV2) are valid, the override is generally set to 100%.

(4) When the reference position is approached, a limit switch installed on the machine is activated, making the deceleration signal (*DEC1, *DEC2, *DEC3,…) for reference position “0”. Consequently, the feedrate is decelerated to 0, then the tool is fed at a constant low speed (reference position return FL feedrate specified by parameter (No. 1425) setting).

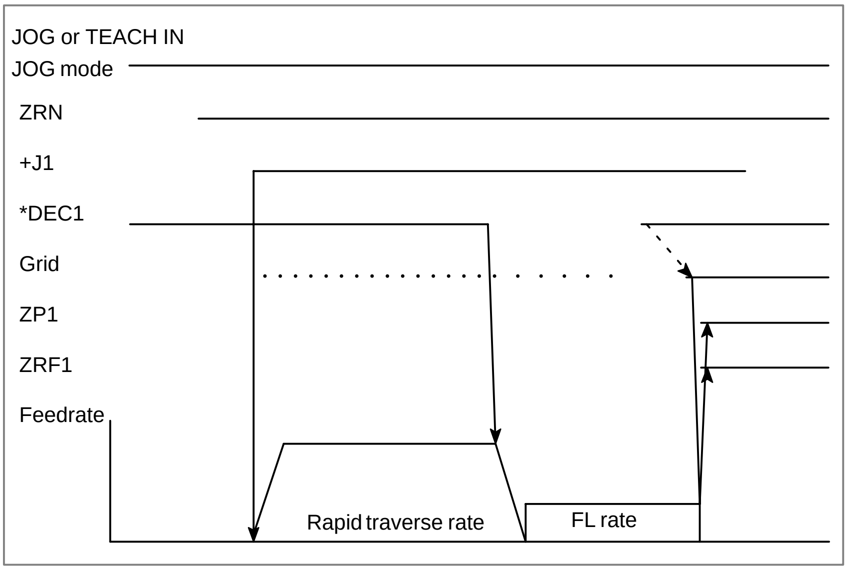
(5) When the deceleration signal returns to “1” again after the limit switch is passed, the tool is continues to feed, until the tool stops at the first grid point (electric grid point).

(6) Upon confirmation that the current position is at the in–position area, the reference position return end signal (ZP1, ZP2, ZP3,…) and the reference position establishment signal (ZRF1, ZRF2, ZRF3,…) turn to “1”.

These steps are repeated for each axis. The number of simultaneously controlled axes is usually one, but it becomes three by setting parameter JAX (No. 1002#0).

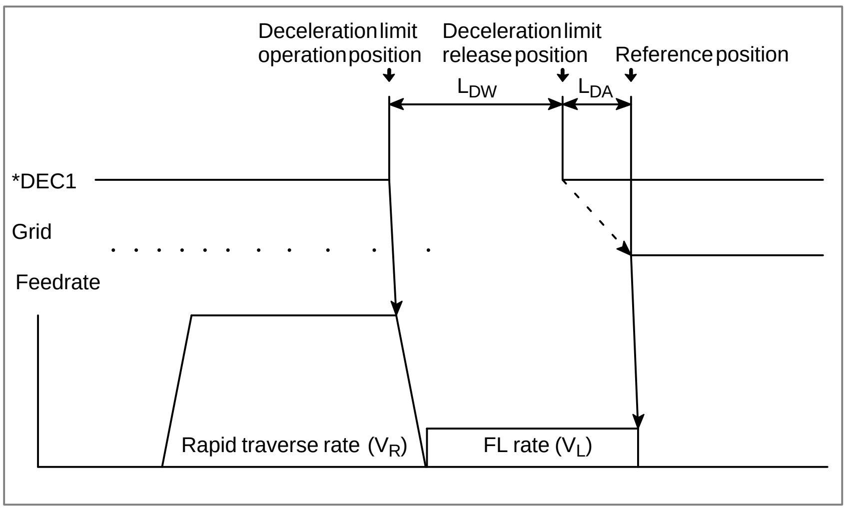
If the feed axis direction selection signal (+J1, –J1, +J2, –J2,…) turns to “0” between step (2) and (4) , the tool is stopped at once, and reference position return is canceled. If the signal turn to “1” again, operation resumes from step (3) (rapid traverse).

The timing charts for the basic procedures are given below.



Installation conditions for deceleration limit switch

When installing the deceleration limit switch for manual reference position return, ensure that following conditions are satisfied:



- L_{DW}: Deceleration dog width (mm or inch)

  $$L_{DW} > \frac{V_R(\frac{T_R}{2} + 30 + T_S) + 4V_L \times T_S}{60 \times 1000}$$

  V_R: Rapid traverse (mm/min or inch/min)
  T_R: Rapid traverse time constant (ms)
  T_S: Servo time constant (ms)
  V_L: FL speed for reference position return (mm/min or inch /min)
- L_{DA}: Distance between deceleration limit switch released position and reference position

  L_{DA}: Move amount of 1/2 revolution of motor

  Since the above conditions do not include the limit switch operation variations, this point must also be considered at installation.

Servo position error and one–rotation signal

To perform the first manual reference position return after power on, the tool must be fed in manual reference position return mode, in the reference position return direction at a speed so that the servo position error exceeds the value set in parameter No. 1836. At that time, the tool must cross the grid line corresponding to a one–rotation signal from the position detector.

The servo position error is calculated from the following formula:

$$\text{Servo position error amount} = \frac{F \times 1000}{60} \times \frac{1}{G} \times \frac{1}{U}$$

F: Feedrate
G: Servo loop gain [s^{-1}]
U: Detection unit [μm]

(Example)

When the tool is fed at a feedrate F of 6000 mm/min with a servo loop gain G of 30 s^{-1} and a detection unit U of 1 μm, the servo position error is calculated as follows:

$$\text{Servo position error} = \frac{6000 \times 1000}{60} \times \frac{1}{30} \times \frac{1}{1}$$
$$= 3,333$$

By reversing the formula above, the following formula gives the feedrate F needed to obtain a servo position error of 128, when the servo loop gain G is 30 s^{-1} and the detection unit U is 1 μm:

$$F = \frac{128 \times 60}{1000} \times 30$$
$$= 230 \text{ [mm/min]}$$

Therefore, when the servo loop gain is 30 s^{-1}, the detection unit is 1 μm, and parameter No. 1836 is set to 128, the tool must be fed in the reference position return direction at a speed of at least 230 mm/min before completing manual reference position return.

Grid shift

The electronic grid can be shifted by the distance set in parameter 1850, thus shifting the reference position. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821) (grid interval).

Signal

Manual reference position return selection signal ZRN <G043#7>

[Classification] Input signal

[Function] This signal selects manual reference position return. Manual reference position return is a kind of jog feed. Therefor, to select manual reference position return, it is required that the jog mode be selected and that the manual reference position return selection signal be set to “1”.

[Operation] When the manual reference position return selection signal is set to “1”, the control unit performs as described below.

- If jog feed mode is not selected, the control unit ignores the manual reference position return selection signal.
- If jog mode is selected, manual reference position return is enabled. In this case, the manual reference position return selection check signal MREF turns to “1”.

> **NOTE**
> If the ZRN status changes state during jog feed, the feedrate is decelerated to 0. Then, to restart reference position return or jog feed, turn feed axis and direction selection signal to “0” then set it to “1”.

Manual reference position return selection check signal MREF <F004#5>

[Classification] Output signal

[Function] This signal indicates that manual reference position return has been selected.

[Output condition] This signal turns to “1” when:

· Manual reference position return has been selected.

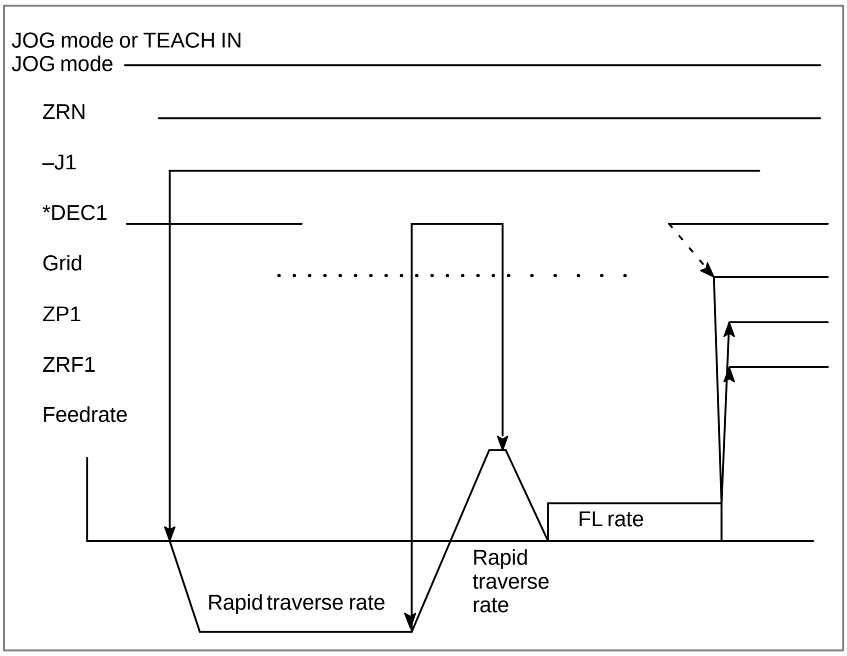
The signal turns to “0” when:

· The selection of manual reference position return has terminated.

Feed Axis and Direction Selection Signal

For details about this signal, see 3.1, "Feed Axis and Direction Selection Signal". Only notes on use of reference position return are given, here.

NOTE
The direction of reference position return is set for each axis by parameter ZMI (No. 1006#5). If the tool is fed opposite to the preset direction in manual reference position return, while the deceleration signal is "0", the tool feeds until the signal returns to "1". The reference position return is then performed automatically in the predetermined direction.



NOTE
When reference position return is selected, an axis who has already completed referencing movement along that axis is disabled while the reference position return selection signal (ZRN) is "1". To perform movement again, ZRN must be set "0", and the feed axis and direction selection signal must be reset to "0" and then returned to "1" again.

Reference position return deceleration signals *DEC1 to *DEC4 <X009#0 to #3>

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference position return to a low feedrate (FL).
Deceleration signals are provided for each axis. The number appended to a deceleration signal represents a controlled axis number.

*DEC 1
- 1 : Reference position return deceleration signal for the first axis
- 2 : Reference position return deceleration signal for the second axis
- 3 : Reference position return deceleration signal for the third axis
- :
- :

[Operation] For a description of the control unit response to the deceleration signal, see the basic procedure for manual reference position return.

Reference position return end signals ZP1 to ZP4 <F094#0 to #3>

[Classification] Output signal

[Function] These signals report that the tool is at the refernce position on a controlled axis.
These siganls are provided for each axis.
The number appended to a signal represents a controlled axis number.

ZP 1
- 1 : Reference position return end signal for the first axis
- 2 : Reference position return end signal for the second axis
- 3 : Reference position return end signal for the third axis
- :
- :

[Output condition] These signals are set to “1” when:

- Manual reference position returns is completed, and the axis position is in the in–position area.
- Automatic reference position return (G28) is completed, and the axis position is in the in–position area.
- Reference position return check (G27) is completed, and the axis position is in the in–position area.

These signals turn to “0”when:

- The tool has moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Reference position establishment signal ZRF1 to ZRF4 <F120#0 to #3>

[Classification] Output signal

[Function] Notify the system that the reference position has been established.

A reference position establishment signal is provided for each axis. The number appended to each signal indicates the number of the controlled axis.

ZRF 1

1 : 1st–axis reference position establishment signal
2 : 2nd–axis reference position establishment signal
3 : 3rd–axis reference position establishment signal
:
:

[Output condition] The signals are set to 1 in the following case:

· When the reference position is established after manual reference position return

· When the reference position is established using the absolute–position detector at initial power–on

The signals are set to 0 in the following case:

· When the reference position is lost

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X009					*DEC4	*DEC3	*DEC2	*DEC1

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0
F004			MREF					
F094					ZP4	ZP3	ZP2	ZP1
F120					ZRF4	ZRF3	ZRF2	ZRF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in JOG feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement (except for G28) is issued in automatic operation (MEM, RMT, or MDI) before referencing is completed (after power on)

0 : An alarm is generated. (P/S alarm 224).
1 : No alarm is generated.

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

NOTE
When this parameter is changed, you must power down the control before continuing.

[Data type] Bit axis

ZMIx The direction of reference position return

0 : Positive direction
1 : Negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		

[Data type] Bit

ZCL Local coordinate system when the manual reference position return is performed

0 : The local coordinate system is not canceled.
1 : The local coordinate system is canceled.

1240	Coordinate value of the reference position on each axis in the machine coordinate system

NOTE
After setting this parameter, turn the power off, then on again so that the setting will take effect.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual reference position return

0 : The stroke limit 1 is checked.

1 : The stroke limit 1 is not checked

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		

[Data type] Bit

JZR The manual reference position return at JOG feedrate

0 : Not performed

1 : Performed

1425	FL rate of the reference position return for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in the halt state during automatic operation (feed hold stop state) under any of the conditions listed below:

0 : Manual reference position return is not performed, with P/S alarm No. 091.

1 : Manual reference position return is performed without an alarm occurring.

< Conditions >

- When there is a remaining distance to travel.
- When an auxiliary function (miscellaneous function, spindle–speed function, tool function) is being executed.
- When a dwell or cycle such as a canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

To set the size of the reference counter, specify the grid interval for the reference position return in the grid method.

$$\text{Size of the reference counter} = \frac{\text{grid interval}}{\text{detection unit}}$$

Grid interval = the amount of travel per rotation of the pulse coder

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

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets the servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

WARNING
When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.

Example: When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.

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

	#7	#6	#5	#4	#3	#2	#1	#0
3003			DEC					

[Data type] Bit

DEC Deceleration signal (*DEC1 to *DEC4) for manual reference position return

0 : Deceleration is applied when the signal is 0.
1 : Deceleration is applied when the signal is 1.

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. During reference position return with the absolute–position detector, if this alarm occurs even though condition 1 is satisfied, do the following: After turning the servo motor for the axis at least one turn, turn the power off and then on again. Then perform reference position return.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed in the feed hold state. Perform a manual reference position return in the automatic operation stop state or reset state.
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start. (Only when parameter ZRNx (No. 1005#0). Do reference position return.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.3.1	MANUAL REFERENCE POSITION RETURN
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.3.1	MANUAL REFERENCE POSITION RETURN
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.3.1	MANUAL REFERENCE POSITION RETURN
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.3.1	MANUAL REFERENCE POSITION RETURN

4.2 SETTING THE REFERENCE POSITION WITHOUT DOGS

General

This function moves each axis in the manual continuous feed mode near the reference position. It then sets the reference position in the reference position return mode without the deceleration signal for reference position return. This is done by setting the feed axis and direction select signal to "1". With this function, the machine reference position can be set at a given position without installing the limit switches for reference position return.

If the absolute-position detector is provided, the set reference position is retained after the power is turned off. In this case, when the power is turned on again, there is no need for setting the reference position again.

Basic Procedure for Setting the Reference Position Without Dogs

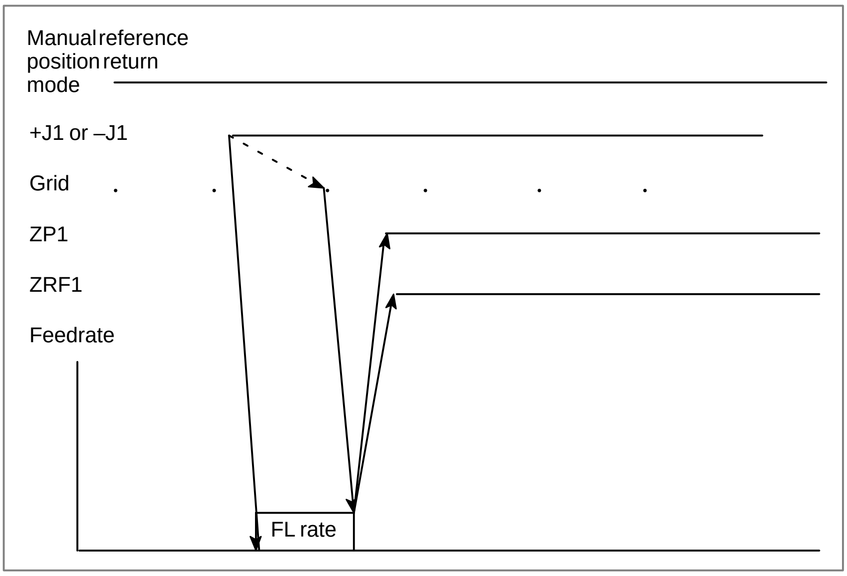
(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.

(2) Enter manual reference position return mode, then set 1 for the feed axis direction selection signal (for the positive or negative direction) for the axis.

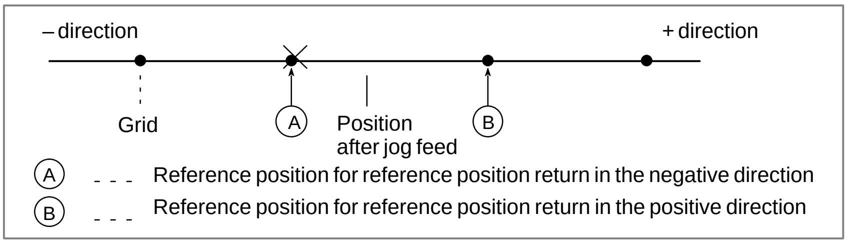
(3) The CNC positions the tool to the nearest grid line (based on one–rotation signals from the position detector) in the reference position return direction specified with bit 5 (ZMIx) of parameter No. 1006. The point at which the tool is positioned becomes the reference position.

(4) The CNC checks that the tool is positioned to within the in–position area, then sets the completion signal for reference position return and the reference position establishment signal to 1.

The timing chart for the basic elements constituting steps (2) to (4) is shown below.



The following figure shows the positional relation between the reference position and the point to which the tool is positioned by manual continuous feed.



Servo position error and one–rotation signal

To set the reference position without dogs, when the reference position has not yet been established. The tool must be fed, in manual continuous feed mode, in the reference position return direction at such a speed that the servo position error exceeds the value set in parameter No. 1836. The tool must cross the grid line corresponding to a one–rotation signal from the position detector.

Section 4.1 explains how to calculate the servo position error.

Grid shift

To shift the reference position, the grid can be shifted by the distance set in parameter No. 1850. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821).

Reference position return

When the feed axis and direction selection signal is set to 1 in manual reference position return mode after the reference position has been established, the tool is positioned to the reference position regardless of the direction specified by the feed axis and direction selection signal. The completion signal for reference position return is then set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002							DLZ	JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in manual continuous feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

DLZ Function for setting the reference position without dog (all axes)

0 : Disabled
1 : Enabled

> **NOTE**
> Bit 1 of parameter No.1002 (DLZ) is used to make common settings for all axes.
> This function can be specified for each axis by DLZx, bit 1 of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
1005							DLZx	ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement (except for G28) is issued in automatic operation (MEM, RMT, or MDI) and when a return to the reference position has not been performed since the power has been turned on

0 : An alarm is generated (P/S alarm 224).
1 : An alarm is not generated.

DLZx Function for setting the reference position without dogs (each axis)

0 : Disabled
1 : Enabled

> **NOTE**
> Bit 1 of parameter No.1005 (DLZx) is used to make settings for each axis.
> When DLZ of parameter No. 1002#1 is 0, DLZx is enabled.
> When DLZ of parameter No. 1002#1 is 1, DLZx is disabled, and the function for setting the reference position without dogs is enabled for all axes.

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

> **NOTE**
> When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction
1 : Negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL		

[Data type] Bit

ZCL Local coordinate system when the manual reference position return is performed

0 : The local coordinate system is not canceled.
1 : The local coordinate system is canceled.

1240	Coordinate value of the reference position on each axis in the machine coordinate system

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual position reference return

0: The stroke limit 1 is checked.

1: The stroke limit 1 is not checked

1425	FL rate of the reference position return for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in feed hold during automatic operation under any of the conditions listed below:

0: Manual reference position return is not performed, with P/S alarm No. 091.

1: Manual reference position return is performed without an alarm occurring.

< Conditions >

- When there is a remaining distance to travel.
- When a auxiliary function (miscellaneous function, spindle–speed function, tool function, B function) is being executed.
- When a dwell or cycle such as a canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

To set the size of the reference counter, specify the grid interval for the reference position return in the grid method.

$$\text{Size of the reference counter} = \frac{\text{grid interval}}{\text{detection unit}}$$

Grid interval = the amount of travel per rotation of the pulse coder

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

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets the servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

> **WARNING**
> When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.

Example: When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the maximum value counted by the reference counter can be specified as the grid shift.

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

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	1. The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. 2. During reference position return with the absolute–position detector, if this alarm occurs even though condition 1 is satisfied, do the following: After turning the servo motor for the axis at least one turn, turn the power off and then on again. Then perform reference position return.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed in the feed hold state. Perform a manual reference position return in the automatic operation stop state or reset state.
224	RETURN TO REFERENCE POINT	Not returned to reference position before cycle start. (Only when parameter ZRNx (No. 1005#0) =0). Do reference position return.

Note

NOTE

P/S alarm No. 090 is issued when G28 is specified and the reference position has not yet be established.

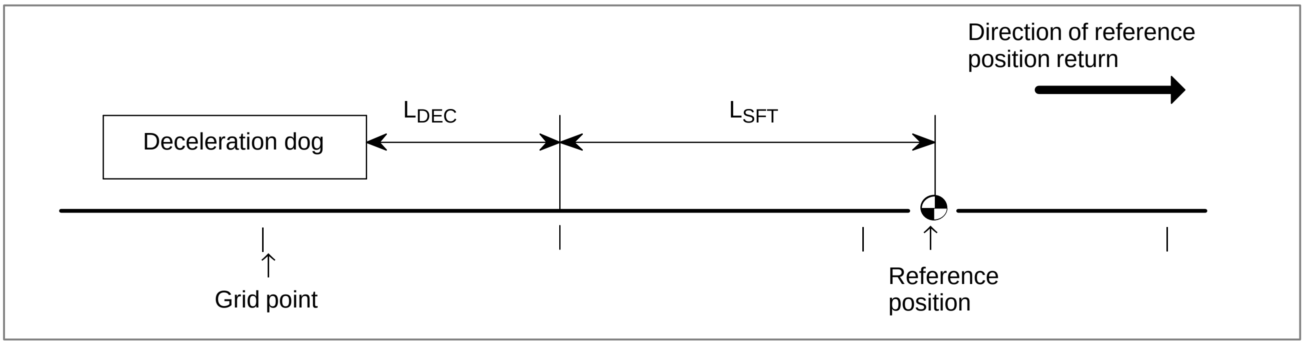
4.3 REFERENCE POSITION SHIFT

General

When reference position return is performed using the grid method, the reference position can be shifted by a parameter–set distance without having to move the deceleration dog.

This function is enabled by setting bit 2 of parameter No. 1002 (SFD) to 1. When distance L_{SFT}, (shown below,) is set in parameter No. 1850, the reference position can be shifted.

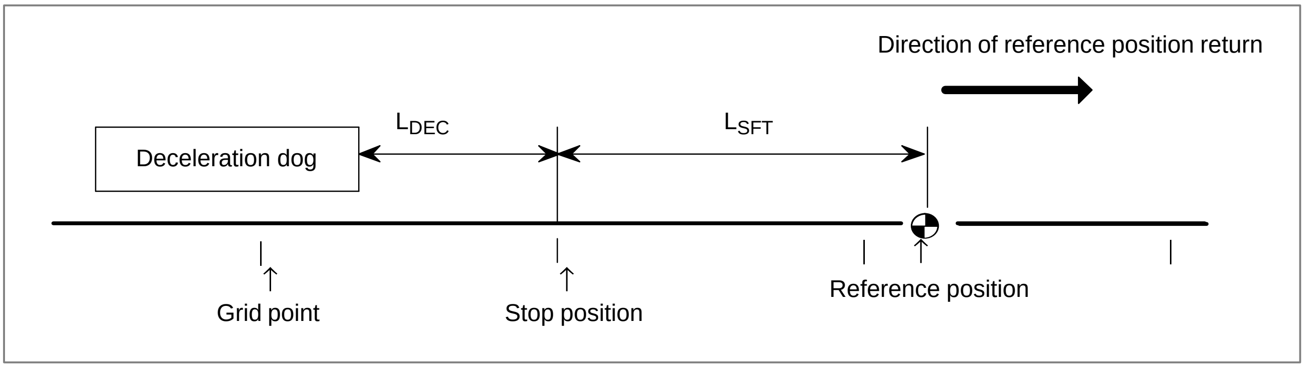
Distance L_{DEC}', (shown below,) for the axis which reference position return was last performed is indicated on the diagnostic screen (No. 0302).



L_{SFT}: Reference position shift amount
L_{DEC}: Distance from the position where the deceleration dog is turned off to the first grid point (grid point when the shift amount is 0)

- **How to adjust the reference position**

(1) Set the SFD bit (bit 2 of parameter No. 1002) to 1, and set the reference position shift amount to 0. Then, perform reference position return.

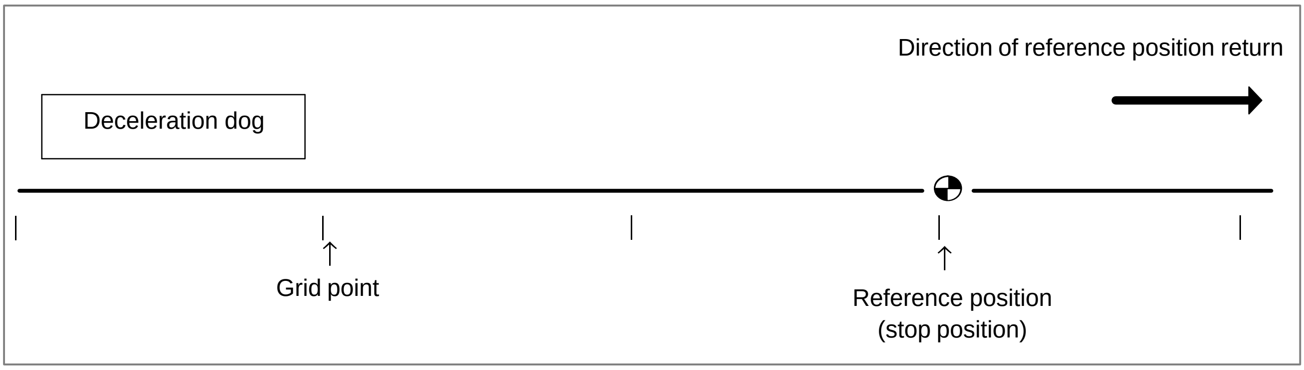


After the deceleration dog is turned off, the tool stops when the first grid point is reached. Distance L_{DEC} is indicated on the diagnostic screen (No. 0302).

(2) Determine the distance L_{SFT} (reference position shift amount) from the stop position to the reference position, and set it in parameter No. 1850.

This completes the adjustment of the reference position.

(3) Perform reference position return again. The tool stops when it reaches the reference position.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002						SFD		

[Data type] Bit

SFD The function for shifting the reference position is
0 : Not used
1 : Used

1850	Reference position shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A reference position shift is set for each axis.

> **CAUTION**
> When bit 2 of parameter No. 1002, SFD, is set to 0, this parameter is used for reference position shift.

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

Alarm and message

- **Diagnostic display**

0302	Distance from the position where the deceleration dog is turned off to the first grid point

[Data type] Two–word axis

[Unit of data] 0.001 mm (metric output), 0.0001 inch (inch output)

[Valid data range] –99999999 to 99999999

Note

NOTE

1 The reference position can be shifted only in the direction of reference position return.

2 When the SFD bit (bit 2 of parameter No. 1002) is 0, only the distance from the position where the deceleration dog is turned off to the first grid point (the grid point after grid shift) is indicated.

4.4 REFERENCE POSITION RETURN

General

The G28 command positions the tool to the reference position, via the specified intermediate point, then sets the completion signal for reference position return (see Section 4.1) to 1.

The reference position must be set in parameter No. 1240 (with the coordinates specified in the machine coordinate system,) before issuing the G28 command.

The tool moves to the intermediate point or reference position at the rapid traverse rate.

Parameter

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

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction
1 : Negative direction

1240	Coordinate value of the reference position on each axis in the machine coordinate system

NOTE
After setting this parameter, turn the power off then on again so that the setting will take effect.

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the reference positions in the machine coordinate system.

Alarm and message

Number	Message	Description
405	SERVO ALARM: (WRONG ZRN)	Position control system fault. Due to an CNC or servo system fault in the reference position return, there is a possibility that reference position return could not be executed correctly. Try the manual reference position return again.

Caution

CAUTION

1 The tool is moved from the intermediate point in a sequence similar to manual reference position return, if the G28 command is issued in the following cases:
   - When the reference position has not yet been established
   - When the input increment (millimeter/inch) is changed at a position other than the reference position

   In these cases, the tool leaves the intermediate point in the reference position return direction specified with bit 5 (ZMIx) of parameter No. 1006. The intermediate point must be specified at a position from which reference position return is possible.

2 If the G28 command is issued in the machine lock status, the completion signal for reference position return is not set to 1.

3 If millimeter input is selected for an inch–system machine, the completion signal for reference position return may be set to 1, even when the programmed tool position deviates from the reference position by the least input increment. This is because the least input increment is smaller than the least command increment for the machine.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.6	REFERENCE POSITION
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.6	REFERENCE POSITION

4.5 2ND REFERENCE POSITION RETURN/3RD, 4TH REFERENCE POSITION RETURN

General

The G30 command positions the tool to the 2nd, 3rd, or 4th reference position, via the specified intermediate point. It then sets the completion signal for 2nd, 3rd, or 4th reference position return to 1.

Before issuing the G30 command, The 2nd, 3rd, or 4th reference position must be set in parameter No. 1241, 1242, or 1243 with coordinates in the machine coordinate system.

The tool moves to the intermediate point or 2nd, 3rd, or 4th reference position at the rapid traverse rate.

Return to the 2nd, 3rd, or 4th reference position can be performed only after the reference position has been established.

Signal

Second reference position return end signals ZP21 to ZP24 <F096#0 to #3> Third reference position return end signals ZP31 to ZP34 <F098#0 to #3> Fourth reference position return end signals ZP41 to ZP44 <F100#0 to #3>

[Classification] Output signal

[Function] The second, third, and fourth reference position end signals report the tool is at the second, third, and fourth reference positions on a controlled axis. These signals are provided for axes in a one–to –one correspondence. A numeric character appended to the end of a signal represents a controlled axis number, and a numeric character immediately following ZP represents a reference position number.

ZP 2 1

1 : Return end signal for the first axis
2 : Return end signal for the second axis
3 : Return end signal for the third axis
:
:

2 : Second reference position return
3 : Third reference position return
4 : Fourth reference position return

[Output condition] These signals turn to “1” when:

- The second, third, or fourth reference position return (G30) is completed, and the current position is in the in–position area.

These signals turn to “0” when:

- The tool moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F096					ZP24	ZP23	ZP22	ZP21
F098					ZP34	ZP33	ZP32	ZP31
F100					ZP44	ZP43	ZP42	ZP41

Parameter

1241	Coordinate value of the second reference position on each axis in the machine coordinate system
1242	Coordinate value of the third reference position on each axis in the machine coordinate system
1243	Coordinate value of the fourth reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to 99999999

Set the coordinate values of the second–fourth reference positions in the machine coordinate system.

Alarm and message

Number	Message	Description
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command. Correct program.

Caution

CAUTION

1 If the G30 command is issued in machine lock status, the completion signal for 2nd, 3rd, or 4th reference position return is not set to 1.

2 If millimeter input is selected for an inch–system machine, the completion signal for 2nd, 3rd, or 4th reference position return may be set to 1, even when the programmed tool position deviates from the 2nd, 3rd, or 4th reference position by the least input increment. This is because the least input increment is smaller than the least command increment for the machine.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.6	REFERENCE POSITION
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.6	REFERENCE POSITION

4.6 BUTT–TYPE REFERENCE POSITION SETTING

General

This function automates the procedure of butting the tool against a mechanical stopper on an axis to set a reference position. The purpose of this function is to eliminate the variations in reference position setting that arise depending on the operator, and to minimize work required to make fine adjustments after reference position setting.

Select the axis for which the reference position is to be set, then perform cycle start. Then, the following operations are performed automatically:

1. The torque (force) of the selected axis is reduced to make the butting feedrate constant, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter–set distance from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter–set distance from the mechanical stopper.
3. The withdrawal point on the axis is set as the reference position.

Basic procedure for butt–type reference position setting

(1) First, set the parameters required for butt–type reference position setting.

ZMIx, bit 5 of parameter (No. 1006):	Direction of reference position setting
Parameter No. 7181:	Withdrawal distance
Parameter No. 7182:	Reference position setting distance
Parameter No. 7183:	Butting feedrate 1
Parameter No. 7184:	Butting feedrate 2
Parameter No. 7185:	Withdrawal feedrate in reference position setting
Parameter No. 7186:	Torque limit

(2) Select manual reference position return mode.

(3) By using a manual handle feed axis select signal, select the axis on which the reference position is to be set.

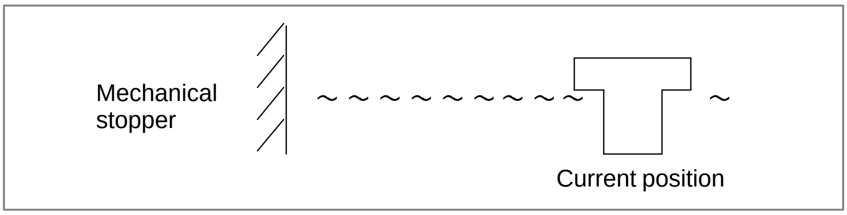
(4) Perform cycle start.

This starts the cycle operation for reference position setting.

(5) During the cycle operation, the automatic operation start signal OP is 1.

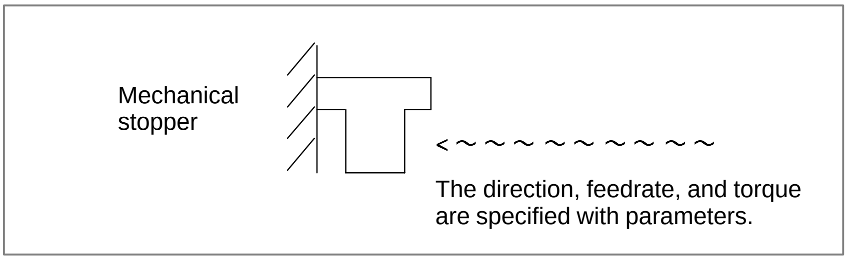
Cycle operation

When no reference position has been set (APZx, bit 4 of parameter No. 1815, is 0), operations (A) to (E), below, are performed automatically to set a reference position.



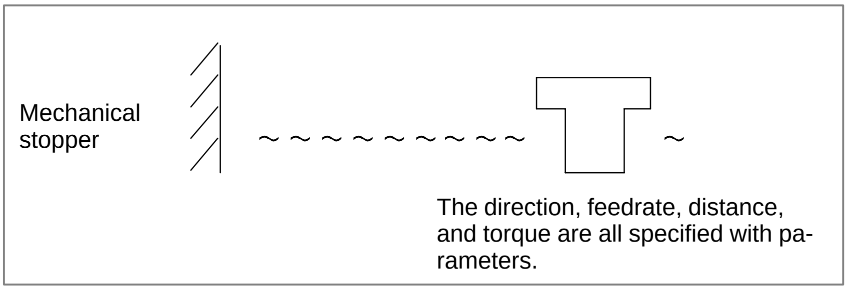
(A)The tool is moved along a specified axis with a limited torque until it butts against the mechanical stopper.

The tool is moved in the direction specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7183, at the torque specified with parameter No. 7186 (until the tool strikes the mechanical stopper).



(B)After the tool strikes the mechanical stopper, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter–set distance.

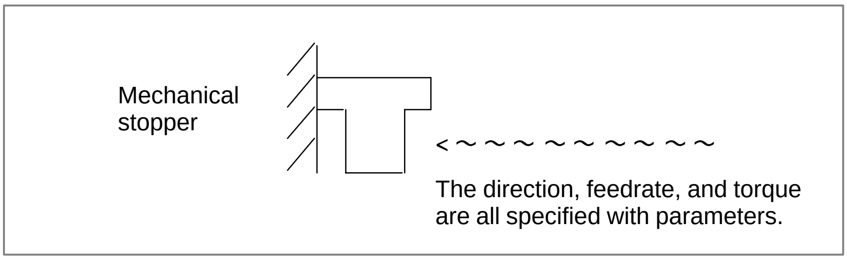
The tool is moved in the direction opposite to that specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7181.



(C)Operations (D) and (E) are performed from the withdrawal point, such that the tool is butted against the mechanical stopper at a constant feedrate in reference position setting.

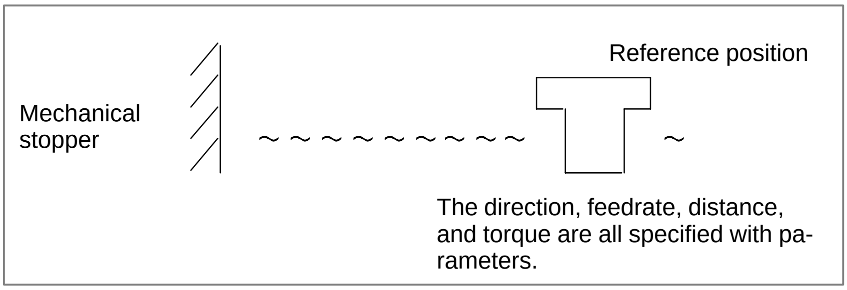
(D)The tool moves along the specified axis at a specified torque until it butts against the mechanical stopper.

The tool moves in the direction specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7184, at the torque specified with parameter No. 7186 (until the tool strikes the mechanical stopper).



(E)After the tool strikes the mechanical stopper end on the axis, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter–set distance.

The tool is moved in the direction opposite to that specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7182.



For parameter Nos. 7183 and 7184, set the feedrates at which the tool is moved toward the mechanical stopper with a limited torque, considering the machine accuracy.

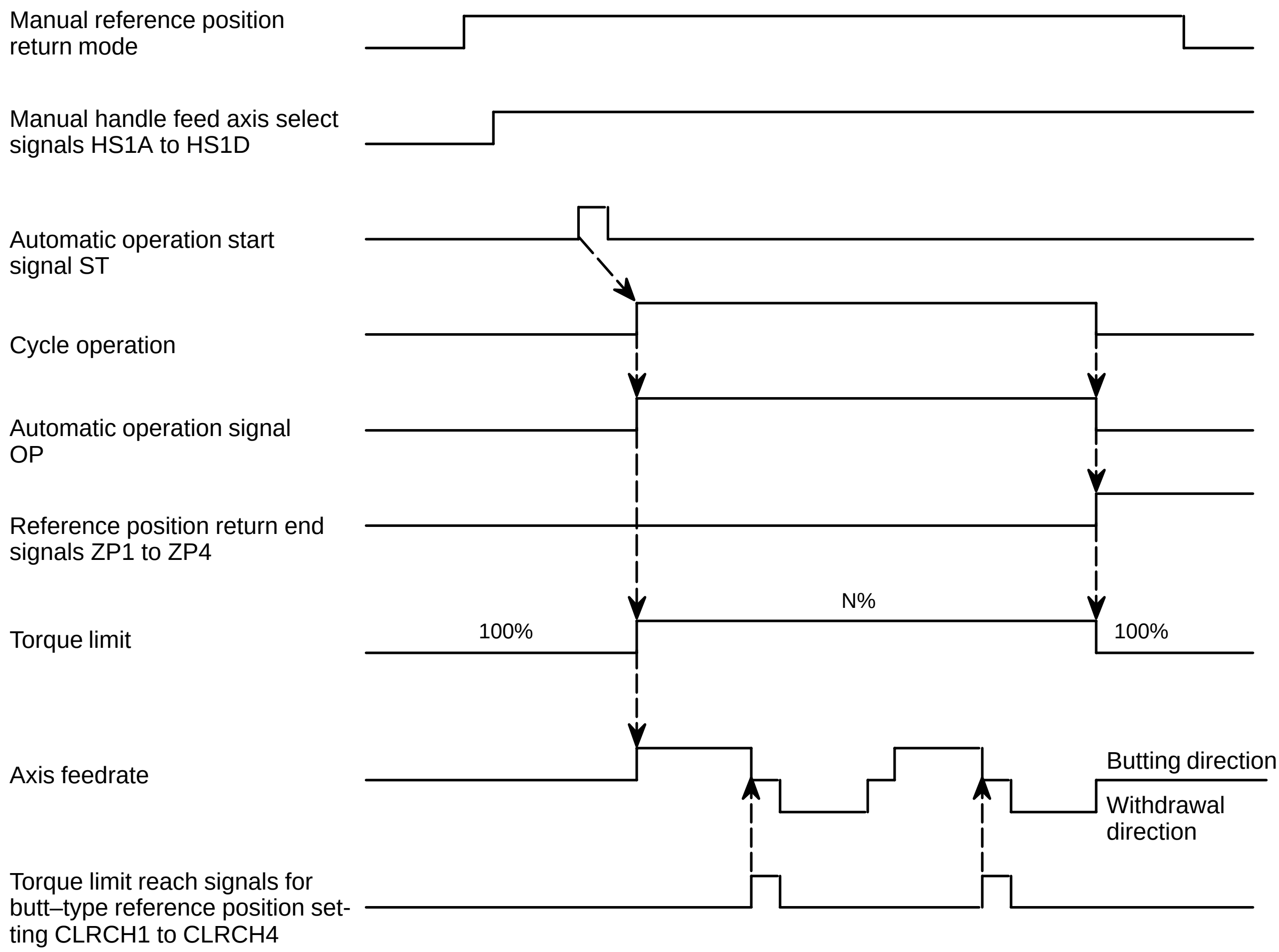
After the tool strikes the mechanical stopper, and the tool is withdrawn the distance specified with parameter No. 7182, the withdrawal point is set as the reference position on the specified axis. Then, the reference position return end signal and reference position establishment signal are set to 1.

After reference position return has been completed, alarm PS000 is issued. Turn the power off then back on before continuing operation.

After the reference position is set

When the reference position has already been set (when APZx, bit 4 of parameter No. 1815, is 1), performing butt–type reference position setting causes the tool to be positioned to the reference position at the rapid traverse rate without the cycle operation. Upon the completion of positioning, the reference position return end signal is set to 1.

The timing chart for the cycle operation is shown below.



N%: Torque limit (%) specified with parameter No. 7186 (%)

Signal

Torque limit reach signals for butt–type reference position setting CLRCH1 to CLRCH4 <F180#0 to #3>

[Classification] Output signal

[Function] These signals are used to post notification of the torque limit having been reached for each corresponding axis during cycle operation for butt–type reference position setting.

[Operation] Each signal is set to 1 when:

· The torque limit is reached for the corresponding axis during cycle operation for butt–type reference position setting.

Each signal is set to 0 when:

· The torque limit is not reached for the corresponding axis during cycle operation for butt–type reference position setting.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F180					CLRCH4	CLRCH3	CLRCH2	CLRCH1

Parameter

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

NOTE
When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on.

0 : Positive direction
1 : Negative direction

7181	First withdrawal distance in butt–type reference position setting

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

When the butt–type reference position setting function is used, this parameter sets a distance on an axis, along which withdrawal is performed after the mechanical stopper is hit (distance from the mechanical stopper to the withdrawal point).

> **NOTE**
> Set the same direction as the direction set for ZMIx, bit 5 of parameter No. 1006. If the opposite direction is set, the cycle operation will not start.

7182	Second withdrawal distance in butt–type reference position setting

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

When the butt–type reference position setting function is used, this parameter sets a distance on an axis, along which withdrawal is performed after the mechanical stopper is hit (distance from the mechanical stopper to the withdrawal point).

> **NOTE**
> Set the same direction as the direction set for ZMIx, bit 5 of parameter No. 1006. If the opposite direction is set, the cycle operation will not start.

7183	First butting feedrate in butt–type reference position setting

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate first used to hit the stopper on an axis.

7184	Second butting feedrate in butt–type reference position setting

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate used to hit the stopper on an axis for a second time.

7185	Withdrawal feedrate (common to the first and second butting operations) in butt–type reference position setting)

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate used for withdrawal along an axis after the mechanical stopper has been hit.

7186	Torque limit value in butt–type reference position setting

[Data type] Byte axes

[Unit of data] %

[Valid data range] 0 to 100

This parameter sets a torque limit value in butt–type reference position setting.

> **NOTE**
> When 0 is set in this parameter, 100% is assumed.

Alarm and message

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.

Note

> **NOTE**
> This function is supported only when an absolute–position detector is installed.

4.7 LINEAR SCALE I/F WITH ABSOLUTE ADDRESS REFERENCED MARK (A/B PHASE)/LINEAR SCALE WITH DISTANCE–CODED REFERENCE MARKS (SERIAL)

Outline

By using optional function ”Linear scale I/F with absolute address referenced mark”, we can use ”Linear scale I/F with absolute address referenced mark (A/B phase)”, and ”Linear scale with distance–coded reference marks (serial)”

Linear scale I/F with absolute address referenced mark (A/B phase)

The interval of each reference marks of linear scale I/F with absolute address referenced mark (A/B phase) are variable. Accordingly, if the interval is determined, the absolute position can be determined. The CNC measures the interval of reference marks by axis moving of short distance and determines the absolute position. Consequently the reference position can be established without moving to reference position.

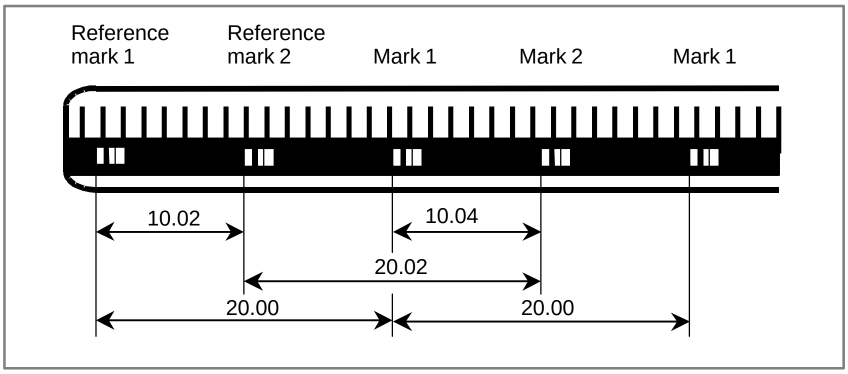
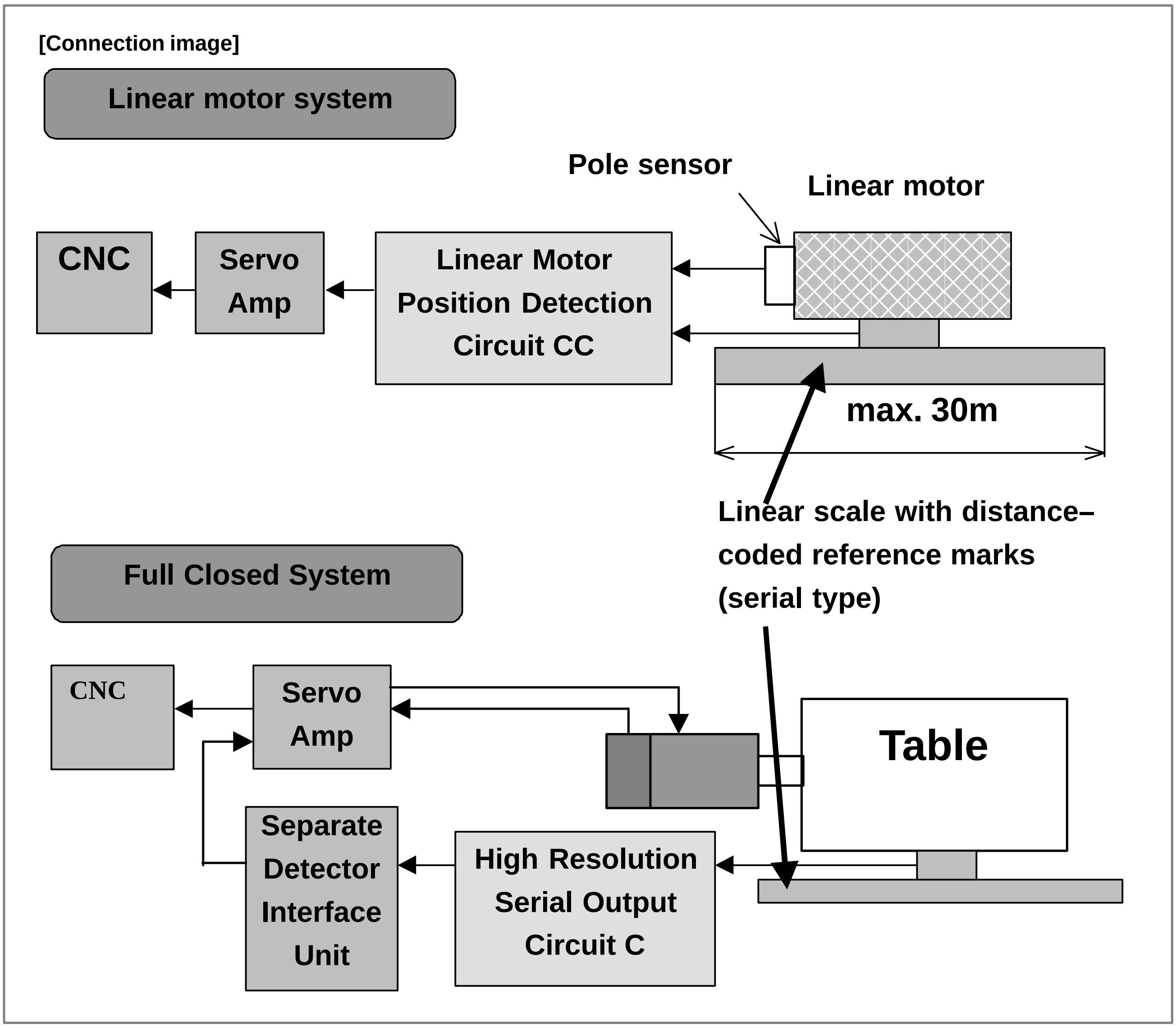


Fig. 4.7(a) Example of linear scale I/F with absolute address referenced mark (A/B phase)

Linear scale with distance–coded reference marks (serial)

The basic structure of Linear scale with distance–coded reference marks (serial) is same as A/B–phase scale (Linear scale with absolute address referenced mark). But this scale differs from A/B–phase in point of circuit. High–resolution serial output circuit is used.
This serial output signal enables high–speed high–precision detection.

Both linear motor system and full closed system are available.



Specifications (linear scale I/F with absolute address referenced mark (A/B phase))

Procedure for reference position establishment

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to ”1”.

(2) Set a direction selection signal(+J1,–J1,+J2,–J2,…) for a target axis.

(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter (No.1425) setting).

(4) When a reference mark is detected, the axis stops, then the axis is fed at a constant low speed again.

(5) Above (4) is executed repeatedly until two or three or four reference marks are detected. And absolute position is determined and reference position establishment signal (ZRF1,ZRF2,ZRF3,+ _…) turns to ”1”. The number of reference marks to be detected is defined by parameter No.1802.

The axis does not stop and reference position establishment is proceeded even if the feed axis direction selection signal (+J1,–J1,+J2,–J2,…) turns to ”0” between step (2) and (5).

The timing chart for this procedures is given below.

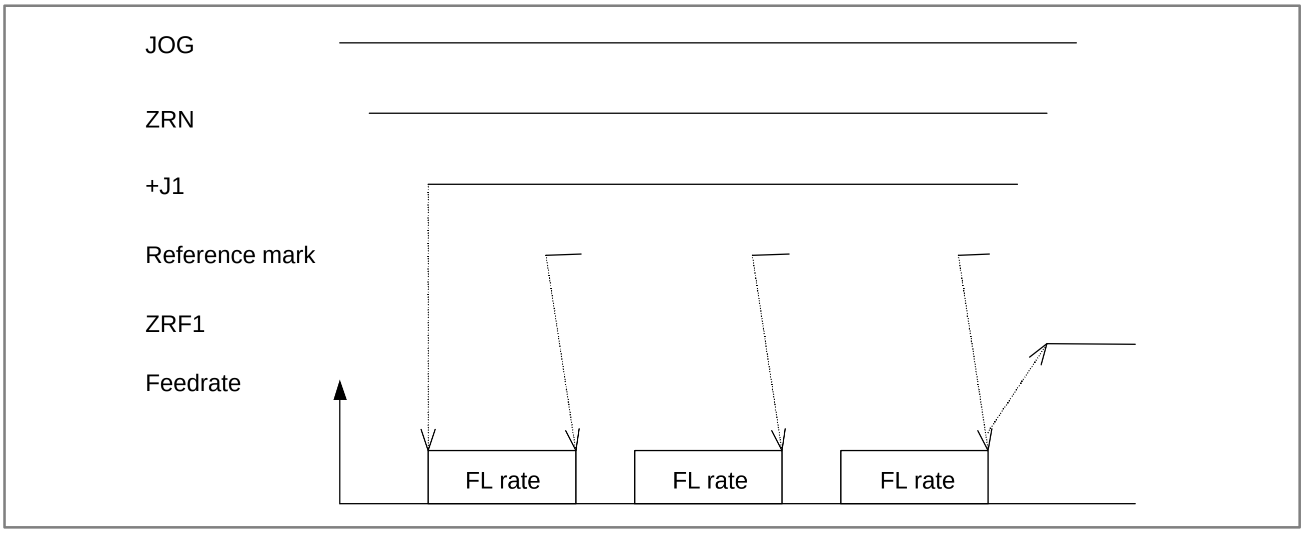


Fig. 4.7(b) Timing chart for reference position establishment

Simple synchronous axis

The function is available for only FS16i/18i/21i–MB,18i–MB5.
When the function is applied for simple synchronous axis, the following condition should be kept.

(1) Linear scale I/F with absolute address referenced mark (A/B phase) with the same reference marks at intervals should be applied for the master axis and the slave axis.

(2) The master axis scale and the slave axis scale should be installed in parallel direction. (The zero positions should be faced the same direction.)

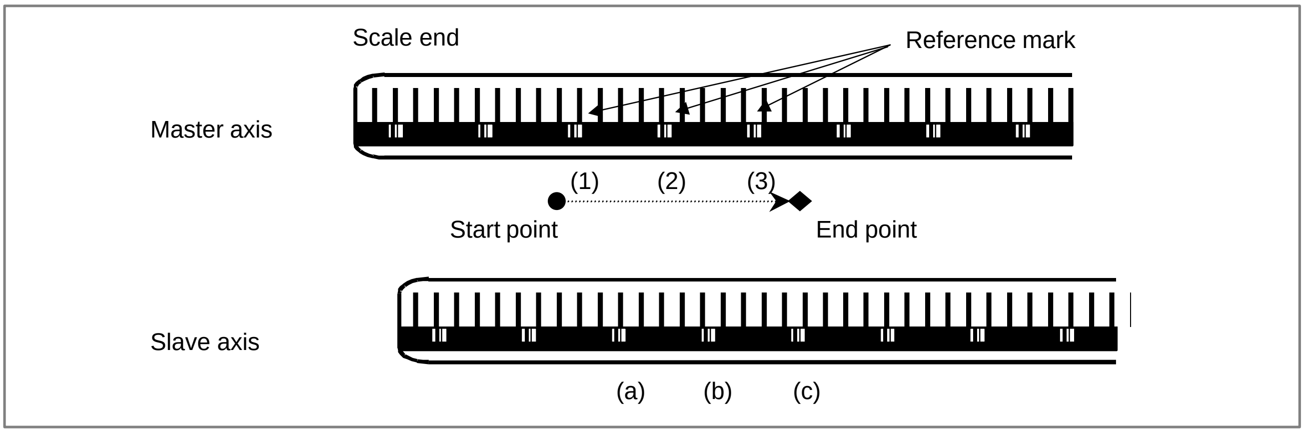
(3) To the parameters, which relates to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.

(4) During operating the establishment of reference position, the state of selecting the manual feed axis for simple synchronous control signals (SYNCJn<G0140>) should be kept.

Procedure for Reference Position Establishment by synchronous axis is as follows.

– When either reference mark of the master axis or the slave axis is detected, the both axes stop. And the both axes are fed again at a Reference Position Return FL Feedrate.
– The above mentioned operations are repeated until the master axis and the slave axis detect enough (3 or 4) reference marks.
– Absolute position of both axes are calculated and Reference Position Establishment Signal (ZRF1,ZRF2,...) turns to "1".

(Example of 3 points measurement system)



In the above example, the following sequence is executed.

a. When the reference mark (1) of the master axis is detected, both master axis and slave axis stop.
b. Both the axes begin to move again at a Reference Position Return FL feedrate.
c. When the reference mark (a) of the slave axis is detected, both axes stop again.
d. Both the axes begin to move again at FL feedrate.
e. Both axes repeat the operation until all point ((2) → (b) → (3) → (c)) are detected.
f. When the slave axis detects the third reference mark (c), both the axes end the Reference Position Establishment.

[Synchronization]

By setting bit 7 (for one simple synchronization pair) of parameter No. 8301 for synchronization or bit 7 (for multiple simple synchronization pairs) of parameter No. 8303 to 1, compensation pulses are output to the slave axis at reference position establishment time to match the machine position with the master axis. Thus, an error between the master axis and slave axis can be corrected automatically.

NOTE

1 When a synchronization error is greater than the value of parameter No. 8315 (for one simple synchronization pair) or the value of parameter No. 8325 (for multiple simple synchronization pairs), servo alarm 407 is issued. Servo alarm 407 can be reset by using the reset key. In this case, however, the slave position remains to be shifted. So, perform slave positioning.

2 When a slave alarm is reset, synchronization is not performed.

3 In the case of multiple simple synchronization pairs, set bit 7 (axis–by–axis parameter) of parameter No. 8303 for the master axis.

Reference position return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,–J1,+J2,–J2,...) to ”1” in REF mode, the reference position establishment procedure is executed.

(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,–J1,+J2,–J2,...) to ”1” in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.

(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of PRM No.1818#0(RFS).

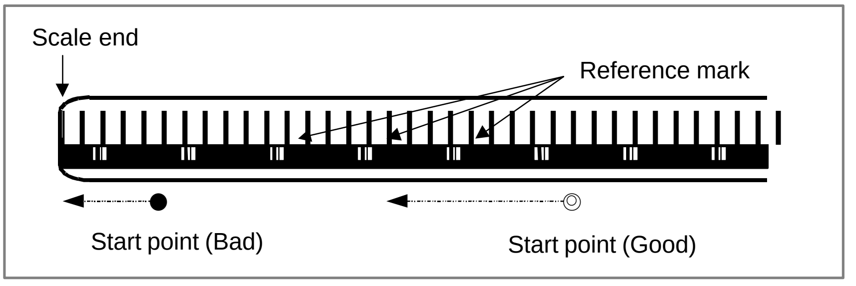
(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of PRM No.1818#1(RF2).

Note

(1) In the following case, P/S090 alarm occurs.

(a) The actual interval of reference marks is different from parameter setting value.

(2) In this procedure, the axis does not stop until three or four reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three or four reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.



(3) In the following cases, this function does not perform.

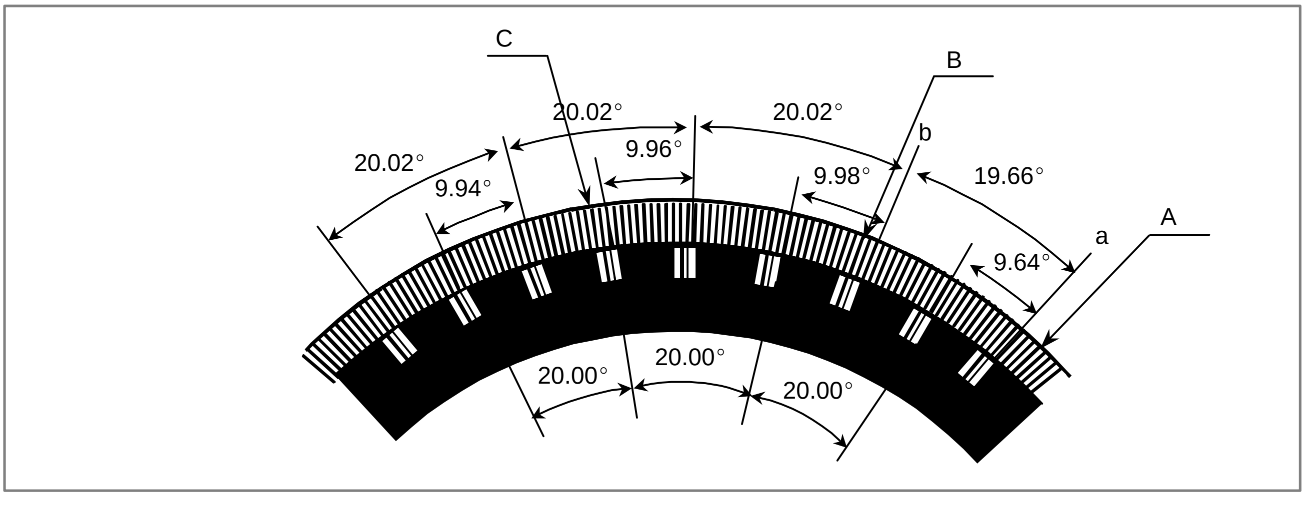
(a) Parameter No.1821(mark1 interval) or No.1882(mark2 interval) is ”0”.

(b) The setting value of parameter No.1821 and No.1882 are the same.

(c) Parameter No.1821 value ≥ No.1882 value*2
or No.1882 value ≥ No.1821 value*2

(4) Rotary encoder with absolute address referenced mark (A/B phase)

(a) When the rotary encoder with absolute address referenced mark (A/B phase) is used, please set a parameter No.1815#3 (DCR) to ”1”.
In case of rotary encoder with absolute address referenced mark (A/B phase), the marker interval may be different from parameter setting value. (a–b section of the following figure) When the reference point return is executed through this section, it is not able to establish the reference point. Therefore, in case of rotary encoder with absolute address referenced mark (A/B phase), if the reference point return is started for B point from A point of below figure, the reference point is not established yet at B point. The reference point return is re–started for C point. The reference point return procedure is finished at C point.



(b) When the reference point return procedure is executed, the coordinate value are rounded in 0 to 360 degree, even if a parameter No. 1006#1(ROS) is set to ”1” (Machine coordinate values are linear axis type).

(c) In case of rotary encoder with absolute address referenced mark (A/B phase), only the measurement by three points or four points is possible. (parameter 1802#2(DC2) is disregarded)

(5) A difference of parameter No.1821 and No.1882 must be more than 4.

Example)
When the scale, which is that mark1 interval is 20.000mm and mark2 interval is 20.004mm, is used on IS–B machine :
When the detection unit of 0.001mm is selected, parameter No.1821 and No.1882 must be set ”20000” and ”20004”, and the difference of them is ”4”.
To use such a scale, please adjust the detection unit by modification of parameter No.1820(CMR) and No.2084/2085(flexible feed gear) to make the difference of No.1821 and 1882 more than 4 as following examples.

(a) Set the detection unit=0.0001mm, and set No.1821=200000, No.1882=200040

(b) Set the detection unit=0.0005mm, and set No.1821=40000, No.1882=40008

NOTE
When the detection unit is changed, all of parameters which are concerned with the detection unit (e.g. in–position width, positioning deviation limit, etc.) must be changed.

(6) There are the following limitations when the angular axis control is used.

(a) It is necessary to use the linear scale I/F with absolute address referenced mark (A/B phase) for both the perpendicular axis and the angular axis.

(b) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the P/S090 alarm is generated.

(c) During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

(7) Measurement of two point (parameter No.1802#2(DC2)=1)

(a) Please set the direction of the scale zero point in the parameter (No.1817#4 (SCP)) correctly at two point measurement. Because an incorrect coordinate system will be established when a wrong value is set in parameter SCP. It is very dangerous. In this case, please execute reference point return again after setting the correct value in parameter SCP.

(b) Because an incorrect value is set in parameter No.1883 and 1884 when setting parameter SCP is incorrect when the automatic setting of parameter No.1883 and 1884 was executed. It is very dangerous. In this case, please execute automatic setting of parameter No.1883 and 1884 again after setting the correct value in parameter SCP.

Specifications (linear scale with distance–coded reference marks (serial))

Procedure for reference position establishment

(1) Select the JOG mode, and set the manual reference position return selection signal ZRN to ”1”.

(2) Set a direction selection signal(+J1,–J1,+J2,–J2,…) for a target axis.

(3) The axis is fed at a constant low speed (reference position return FL feedrate specified by parameter (No.1425) setting).

(4) When the absolute position of linear scale with distance–coded reference marks (serial) is detected, the axis stops. Then the absolute position of CNC is calculated and reference position establishment signal (ZRF1,ZRF2,ZRF3,+ _…) turns to ”1”.

The axis does not stop and reference position establishment is proceeded even if the feed axis direction selection signal (+J1,–J1,+J2,–J2,…) turns to ”0” between step (2) and (4).

The timing chart for this procedures is given below.

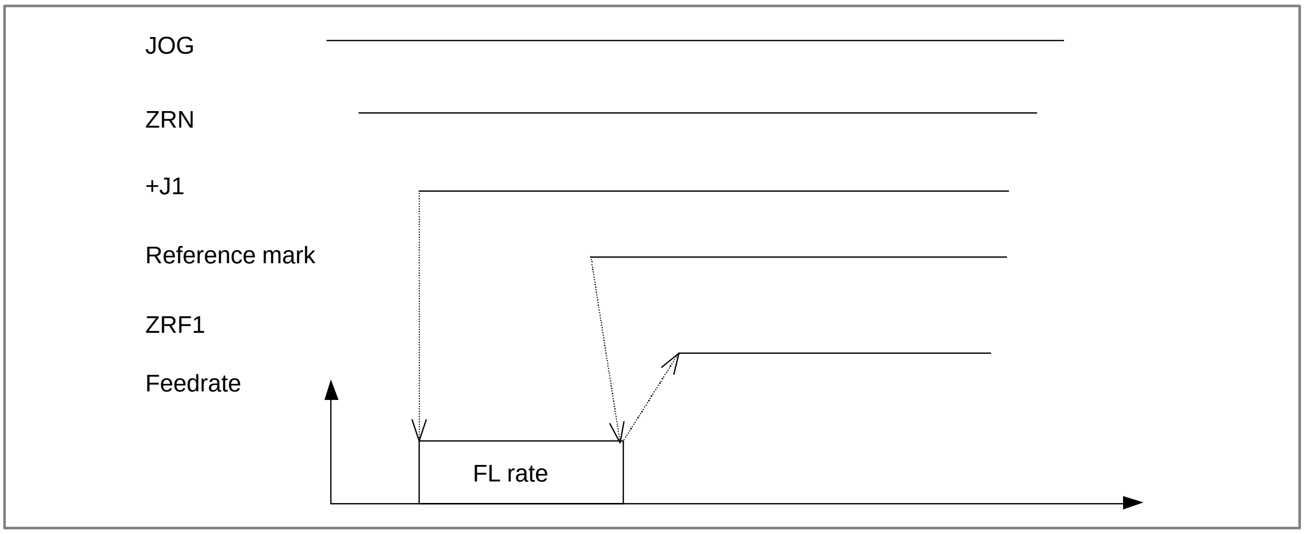


Fig. 4.7(c) Timing chart for reference position establishment

Simple synchronous axis

The function is available for only FS16*i*/18*i*/21*i*–MB,18*i*–MB5.
When the function is applied for simple synchronous axis, the following condition should be kept.

(1) Linear scale with distance–coded reference marks (serial) with the same reference marks at intervals should be applied for the master axis and the slave axis. The mixture of linear–motor system and full–closed system is unavailable between the master axis and the slave axis.
If either of the master axis or the slave axis is not the linear scale with distance–coded reference marks, P/S 5327 occurs when reference position establishment is tried with the signal of selecting the manual feed axis for simple synchronized control (SYNCJn<G0140>) to '1'.

(2) The master axis scale and the slave axis scale should be installed in parallel direction. (The zero positions should be faced the same direction.)

(3) To the parameters, which relate to this function (except No.1883, No.1884), the same value must be set for the master axis and for the slave axis.

(4) During operating the establishment of reference position, the state of selecting the manual feed axis for simple synchronized control signal(SYNCJn<G0140>) should be kept.

Procedure for Reference Position Establishment by synchronous axis is as follows.

– Both of axes (master axis and slave axis) are fed on the reference position return FL feedrate until distance coded scales of both axes detect the absolute position.
– Then absolute position of both axes are calculated and Reference Position Establishment Signals (ZRF1,ZRF2,...) turn to "1".

[Synchronization]

By setting bit 7 (for one simple synchronization pair) of parameter No. 8301 for synchronization or bit 7 (for multiple simple synchronization pairs) of parameter No. 8303 to 1, compensation pulses are output to the slave axis at reference position establishment time to match the machine position with the master axis. Thus, an error between the master axis and slave axis can be corrected automatically.

NOTE

1 When a synchronization error is greater than the value of parameter No. 8315 (for one simple synchronization pair) or the value of parameter No. 8325 (for multiple simple synchronization pairs), servo alarm 407 is issued. Servo alarm 407 can be reset by using the reset key. In this case, however, the slave position remains to be shifted. So, perform slave positioning.
2 When a slave alarm is reset, synchronization is not performed.
3 In the case of multiple simple synchronization pairs, set bit 7 (axis–by–axis parameter) of parameter No. 8303 for the master axis.

Reference position return

(1) When the reference position is not established and the axis moved by turning the feed axis direction signal (+J1,–J1,+J2,–J2,...) to ”1” in REF mode, the reference position establishment procedure is executed.

(2) When the reference position is already established and the axis is moved by turning the feed axis direction signal (+J1,–J1,+J2,–J2,...) to ”1” in REF mode, the axis is moved to the reference point without executing the reference position establishment procedure.

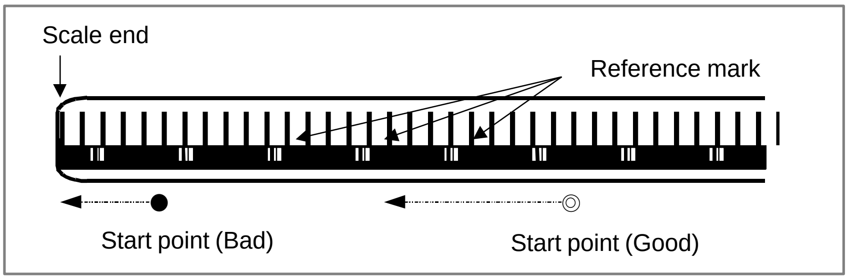
(3) When the reference position is not established and the reference position return command (G28) is executed, the reference position establishment procedure is executed. The next movement the axis depends on the setting of PRM No.1818#0(RFS).

(4) When the reference position is already established and the reference position command (G28) is executed, the movement of the axis depends on the setting of PRM No.1818#1(RF2).

Note

(1) When the Linear scale with distance–coded reference marks (serial) is used, please don’t forget to set parameter No.1818#3(NSD) to 1. And distance coded rotary encoder (serial type) is unavailable.

(2) On the Linear scale with distance–coded reference marks (serial), the axis does not stop until three reference marks are detected. If this procedure is started at the position near the scale end, CNC can not detect three reference marks and the axis does not stop until over travel alarm occurs. Please care to start at the position that has enough distance from scale end.
And if establishment of reference position is failed, the establishment is retried. Then axis does not stop until still more three reference marks are detected. So please set the maximum move amount (detection unit: Prm. No.14010) not to reach the scale end.



(3) There are the following limitations when the angular axis control is used.

(a) It is necessary to use the Linear scale with distance–coded reference marks (serial) for both the perpendicular axis and the angular axis. If not, P/S 5328 occurs when reference position establishment is tried.

(b) When the reference point establishment of angular and perpendicular axes are tried, please set parameter 8200#2(AZR) to ’0’ and input signal G063#5(NOZAGC) to ’0’. If not, P/S 5328 occurs when reference position establishment is commanded.

(c) When the reference point of the perpendicular axis is established, it is necessary to establish the reference point of the angular axis previously. When the reference point of the angular axis is not previously established, the P/S090 alarm is generated.

(d) During the reference point establishment operation of the angular axis, the command in the perpendicular axis is invalid in the manual reference point return.

(e) On angular axis control, if you use automatic setting of parameter No.1883,1884 on reference point establishment (Prm. No.1819#2 (DATx)=1), please establish reference point of perpendicular axis after reference point establishment and return of angular axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1815					DCRx	DCLx	OPTx	

[Data type] Bit axis

OPTx Position detector

0 : A separate pulse coder is not used.
1 : A separate pulse coder is used.

DCLx As a separate position detector, the linear scale with reference absolute addressing mark (A/B phase)/the Linear scale with distance–coded reference marks (serial) is:

0 : Not used.
1 : Used

CAUTION
When using the linear scale with reference absolute addressing marks (A/B phase) or the linear scale with distance–coded reference marks (serial, full closed system), also set the OPTx parameter (bit 1 of parameters No.1815) to ’1’. But when using the linear scale with distance–coded reference marks (serial, linear motor system), please set the OPTx parameter (bit 1 of parameters No.1815) to ’0’.

DCRx As a scale with absolute addressing referenced marks:

0 : The linear scale is used.
1 : The rotary encoder is used.

CAUTION
1 Please set parameter DCLx to ’1’, too.
2 The rotary encoder with distance–coded reference marks (serial) is unavailable.

	#7	#6	#5	#4	#3	#2	#1	#0
1802						DC2	DC4	

[Data type] Bit

DC4 When the reference position is established on the linear scale with reference marks:

0 : An absolute position is established by detecting three reference marks.
1 : An absolute position is established by detecting four reference marks.

> **CAUTION**
> This parameter is unavailable on the linear scale with distance–coded reference marks (serial).

DC2 The reference position on the linear scale with absolute addressing referenced mark is established:

0 : 7As determined by bit 1(DC4) of parameter No.1802.
1 : By establishing the absolute position through detection of two reference marks.

> **CAUTION**
> 1 When this parameter is set to 1, set the direction of scale zero in the parameter (No.1817#4 (SCPx)) correctly.
> 2 Even if DC2 is ”1”, the rotary axis (parameter 1815#3 (DCRx)=1) follows DC4.
> 3 This parameter is unavailable on the Linear scale with distance–coded reference marks (serial).

	#7	#6	#5	#4	#3	#2	#1	#0
1818					SDCx		RF2x	RFSx

[Data type] Bit axis

RFSx If an automatic reference position return (G28) is made before the reference position is established, the reference position is established first then,

0 : A movement to reference position is made.
1 : A movement to reference position is not made.

RF2x If an automatic reference position return (G28) is made after the reference position is established,

0 : A movement to the reference position is made.
1 : A movement to the reference position is not made, but the operation is completed.

SDCx Linear scale with distance–coded reference marks (serial) is

0 : Unavailable.
1 : Available.

> **WARNING**
> If you set parameter 1818#3 (SDCx), please don’t forget to turned off before operation is continued. This parameter doesn’t generate P/S alarm 0 (Power–off alarm).

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

[Data type] Bit axis

DATx When manual reference point return is executed, the automatic setting of paramete 1883,1884 is:

0 : Not executed.
1 : Executed.

The procedure of the automatic setting are following :

(1) Set a correct value to parameter No.1815,1821,1882.
 * On the linear scale with distance–coded reference marks (serial), parameter No.1882 is unnecessary to set.

(2) Positioning the axis to reference position by mechanical measurement by manual operation.

(3) Execute manual reference point return.
When the manual reference point return is finished, parameter No.1883,1884 are set and No.1819#2 is turn to ’0’.

1821	Reference counter size for each axis

[Data type] 2–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Sets the intervals of mark 1 of the linear scale with abssolute addressing referenced marks.

> **CAUTION**
> On the linear scale with distance–coded reference marks (A/B phase), this parameter is used on the usual purpose. (Setting reference counter size foreach axis)

1882	Space between mark–2 indications on the linear scale with absolute addressing reference marks

[Data type] 2–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Sets the intervals of mark 1 of the linear scale with abssolute addressing referenced marks.

CAUTION
This parameter is unavailable on linear scale with distance–coded reference marks (serial).

1883	Distance from the zero point of the linear scale with absolute addressing reference marks (A/B phase) / the linear scale I/F with absolute address referenced mark (serial) to the reference position

[Data type] 2–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

1884	Distance–2 from the zero point of the linear scale with absolute addressing reference marks (A/B phase) / the linear scale I/F with absolute address referenced mark (serial) to the reference position

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –20 to 20

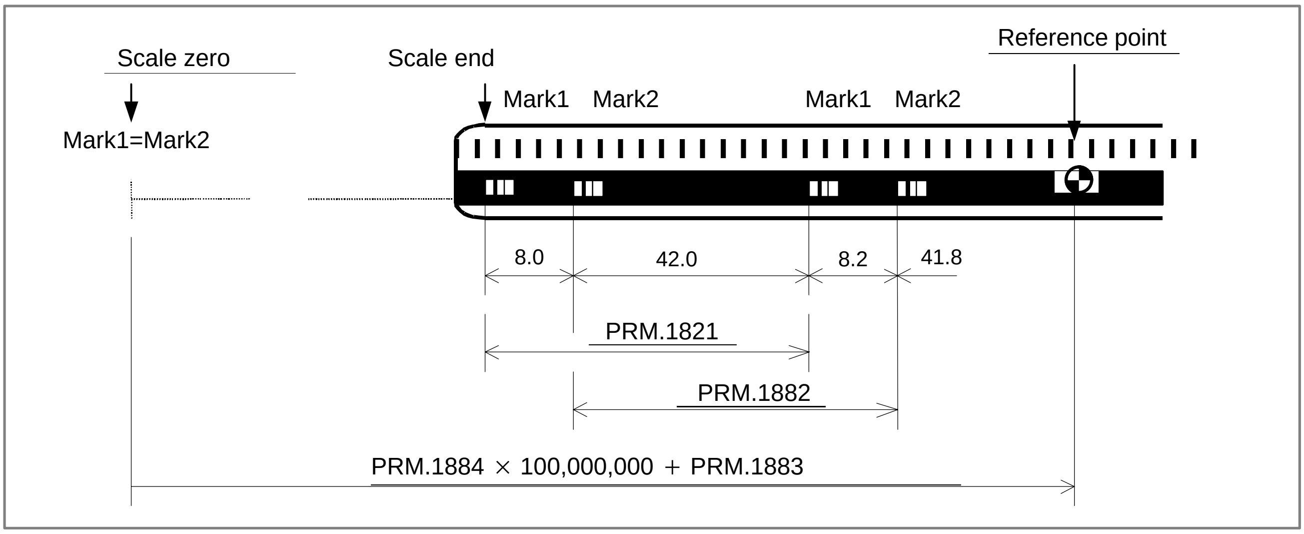
When the distance between scale zero and reference position is over the setting range of parameter 1883, please set this parameter.

CAUTION
On using the linear scale with distance–coded reference marks (serial), when the value of this parameter is out of range, P/S alarm 5325 may occur on trying to establish the reference point.

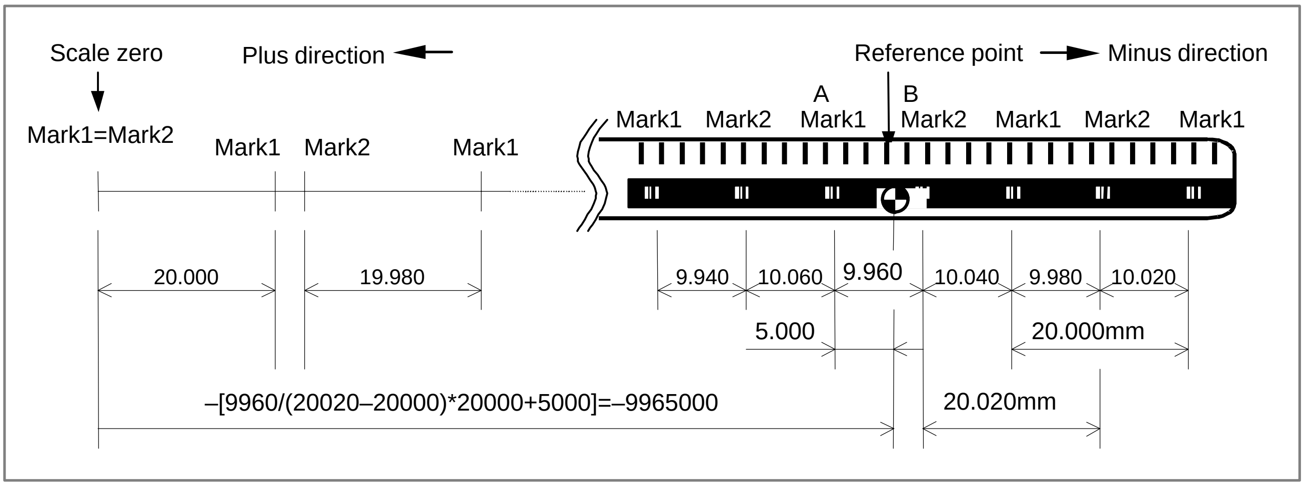
Set the distance between scale zero and reference position by parameter No.1883, 1884. The actual distance is determined by following formula:

Actual distance = Parameter 1884 * 100,000,000 + Parameter 1883

The scale zero means a point where reference mark 1 and reference mark 2 are equal. Generally, this is a virtual point that exists on the out of scale stroke.(Refer to following figure)
When a reference point is located in plus side of a scale zero, set a plus value to this parameter. And a reference point is located in minus side, set a minus value.



[Example of parameter setting]
When IS–B and millimeter machine and using a scale figured below:



Parameter
No.1821 (Mark1 interval) = ”20000”
No.1882 (Mark2 interval) = ”20020”
No.1883 (Reference position) = Position of point A + 5.000
= (Distance of A to B) / (Mark2 interval – Mark1 interval) * Mark1 interval + 5.000
= 9960 / (20020–20000) * 20000 + 5000
= 9965000
”–9965000”(Reference point is located in minus side)

[The setting method of parameter No.1883]
(Linear scale with absolute addressing reference mark (A/B phase))
When the measurement of a parameter No.1883 (Distance between scale zero and reference position) is difficult, you can determine the setting value by the following procedure.

(1) Set ”1” to parameter No.1815#2,#1 to make this function available.
Set a correct value to parameter No.1821, No.1882.
Set ”0” to parameter No.1240.
Set ”0” to parameter No.1883,1884.

(2) Establish a reference point by the procedure described in ”1.2.1 Procedure for Reference Position Establishment”. (As a result, the machine coordinate becomes the distance between scale zero and actual position.)

(3) Positioning the axis to reference position by mechanical measurement in ”HND” or ”JOG” mode.

(4) Set an actual machine coordinate value (DGN.301) to the parameter No.1883 after conversion of least command increment to detection unit. (Multiply DGN.301 and CMR)

(5) Set a parameter No.1240 if necessary.

> **NOTE**
> When the setting value is greater than 99,999,999, this method can’t be used.

[The setting method of parameter No.1883]
(Linear scale with distance–coded reference marks (serial))
You can determine the setting value by the following procedure.

(1) Set ”1” to parameter No.1815#2 to make this function available.
Set ”0” to parameter No.1240.
Set ”0” to parameter No.1883,1884.

(2) Establish a reference point by the procedure described in ”1.3.1 Procedure for Reference Position Establishment”. (As a result, the machine coordinate becomes the distance between scale zero and actual position.)

(3) Positioning the axis to reference position by mechanical measurement in ”HND” or ”JOG” mode.

(4) Set an actual machine coordinate value (DGN.301) to the parameter No.1883 after conversion of least command increment to detection unit. (Multiply DGN.301 and CMR)

(5) Set a parameter No.1240 if necessary.

> **NOTE**
> When the setting value is greater than 99,999,999, this method can not be used.

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

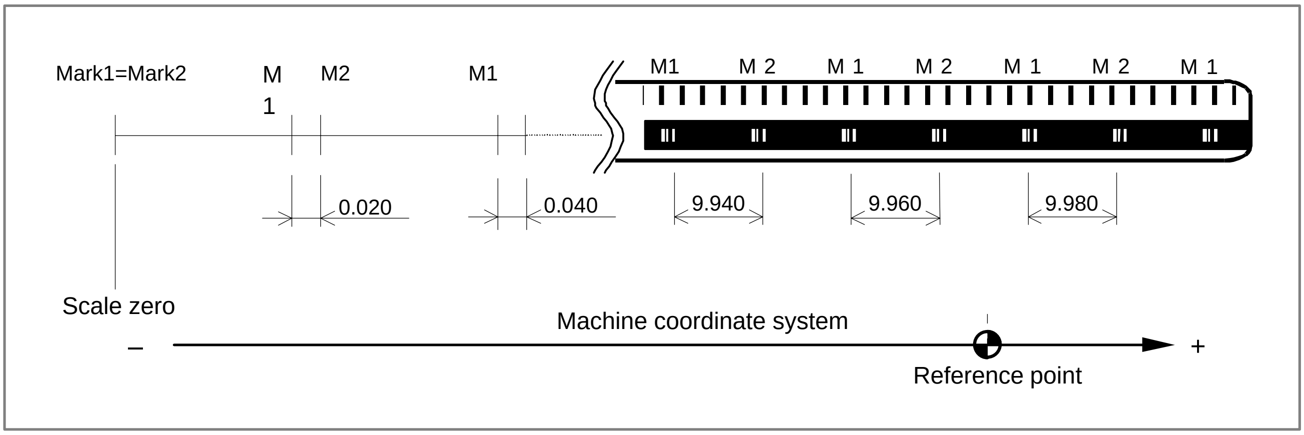
[Data type] Bit axis

SCPx If DC2x (Paramete No.1802#2) is set to ’1’, the scale–zero point of the linear scale with absolute addressing referenced mark is:

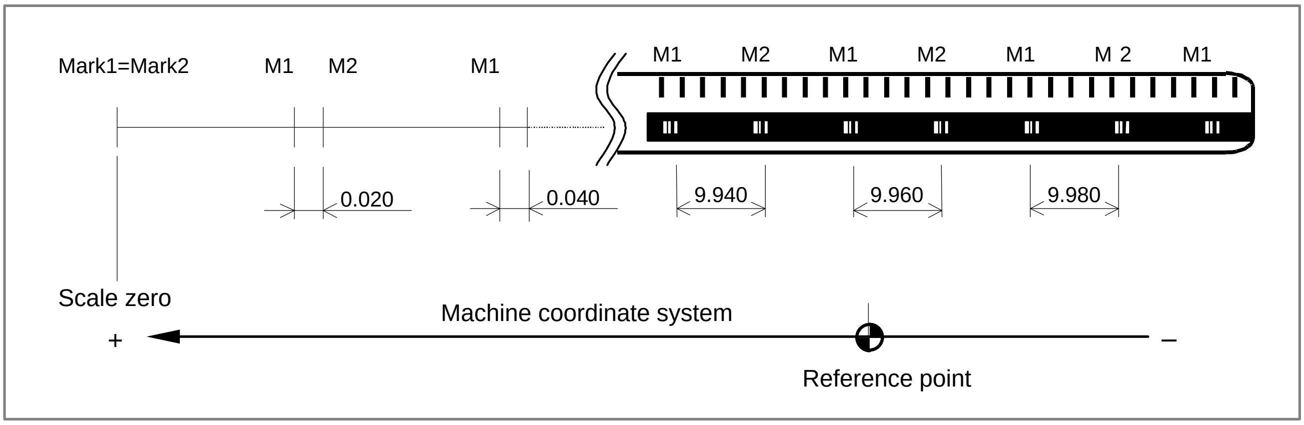
0 : On the negative direction side. (Viewed from the scale zero point, the reference position is on the positive direction side.)
1 : On the positive direction side. (Viewed from the scale zero point, the reference position is on the negative direction side.)

> **CAUTION**
> 1 An incorrect coordinate system will be established when a wrong value is set in this parameter. In such a case, please execute reference point return again after setting the correct value in this parameter.
> 2 This parameter is unavailable on linear scale with distance–coded reference marks (serial)

[Example]
When the parameter SCP is set to ”0”,



When the parameter SCP is set to ”1”,



	#7	#6	#5	#4	#3	#2	#1	#0
8301	SOF							

[Data type] Bit

SOF The synchronization function in simple synchronous control (one pair) is:
0 : Not used.
1 : Used.

	#7	#6	#5	#4	#3	#2	#1	#0
8303	SOFx							

SOFx In simple synchronous control, the synchronization function is:

0 : Not used.
1 : Used.

> **NOTE**
> Set this parameter on the master axis side.

14010	Allowable move amount of FL feedrate on establishment of reference point (linear scale with distance–coded reference marks (serial))

[Data type] 2–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

On the linear scale with distance–coded reference marks (serial type), the amount of movement on FL–speed during establishment of reference point is set. If the reference point is not established despite the amount of FL–speed movement exceeds this parameter, P/S alarm 5326 occurs. When this parameter is set to ’0’, the setting of allowable move amount of FL feedrate on establishment of reference point (linear scale with distance–coded reference marks (serial)) is ineffective.

> **CAUTION**
> 1 When a reference position is established on the M series by using the simple synchronous manual feed axis select signal <G140>, and this parameter is set for one of the master axis and slave axis, the setting is automatically applied to the other axis as well.
> 2 On angular axis control, this parameter setting of perpendicular axis is ignored during the reference position establishment of the angular axis.

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	In case of Linear scale I/F with absolute address referenced mark (A/B phase), the actual interval of reference marks is different from parameter (No.1821, 1882) setting value.
5220	REFERENCE POINT ADJUSTMENT MODE	In case of Linear scale I/F with absolute address referenced mark (A/B phase) or Linear scale with distance–coded reference marks (serial), the reference point auto setting parameter (No.1819#2) is set to "1". Move the machine to reference position by manual operation and execute manual reference return.
5325	SERIAL DCL: FOLLOW–UP ERROR	1. The amount of follow–up is more than 2147483648 or less than –2147483649 on detection unit. To reduce the follow–up amount, please adjust the point that we begin follow–up. 2. The value of parameter No.1884 is out of range.
5326	SERIAL DCL:REF–POS ESTABLISH ERR	The amount of movement for establish reference position was exceeded the amount of parameter 14010. Please try again or change parameter 14010 larger.
5327	SERIAL DCL:MISMATCH (SSYNC CTRL)	Master/slave axes of simple synchronized control, one of them is the linear scale with distance–coded reference marks (serial), and the other of them is not the linear scale with distance–coded reference marks (serial). Please establish reference position with the input signal SYNCJn<g140> setting to zero.
5328	SERIAL DCL:MISMATCH (ANGL–AXIS)	On angular axis control, one of the angular/perpendicular axes is the scale with ref–pos, and the other of them is not the scale with ref–pos. Such system is not admired.

(*) P/S alarm 5325–5328 is generated only on Linear scale with distance–coded reference marks (serial)

4.8 EXTENDED FUNCTION OF THE LINEAR SCALE WITH ABSOLUTE ADDRESSING REFERENCE MARKS

The linear scale with absolute addressing reference marks has reference marks at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. When a G00 command or a move command in jog feed is specified for an axis for which the linear scale with absolute addressing reference marks is used, this function establishes the reference position by measuring the reference mark intervals automatically. Therefore, after CNC power–up, the reference position can be established without performing reference position return operation.

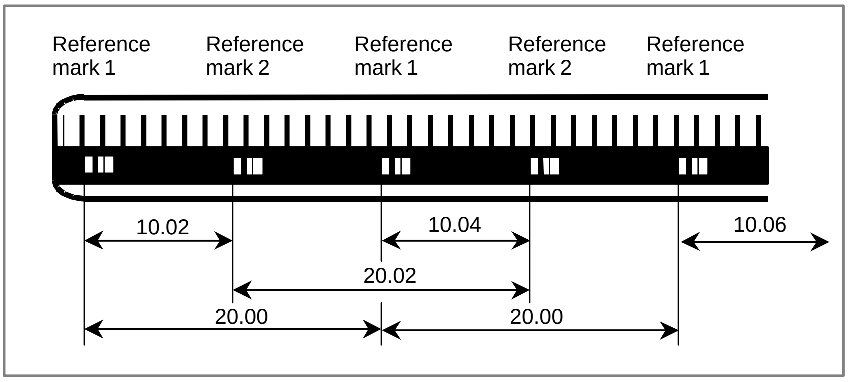


Fig. 4.8 (a) Sample linear scale with absolute addressing reference marks

To use this function, the interface option of the linear scale with absolute addressing reference marks is also required.

Reference position established by the G00 command

- **Activation conditions**

When the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> A G00 command is specified to cause a movement along an axis for which no reference position has been established.

<2> The movement direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.

<3> The specified axis is not in the following modes:

- Coordinate system rotation
- Mirror image (mirror image by signal or setting)
- Programmable mirror image (M series)
- Scaling (M series)
- Mirror image of facing tool posts (T series)

NOTE
If all the above conditions are not satisfied, the reference position establishment operation is not performed, and normal G00 command operation is performed.

- **Operation**

The reference position establishment procedure is explained below.

<1> The tool is fed along a specified axis at the reference position return FL feedrate (parameter No. 1425).

<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.

<3> Step <2> above is repeated until three or four reference marks are detected on the scale. Then, the reference position is established, and the reference position established signal (ZRF1, ZRF2, ZRF3, etc.) is set to 1.

<4> The tool is fed to a specified end point at a rapid traverse rate.

A time chart for the above procedure is shown below.

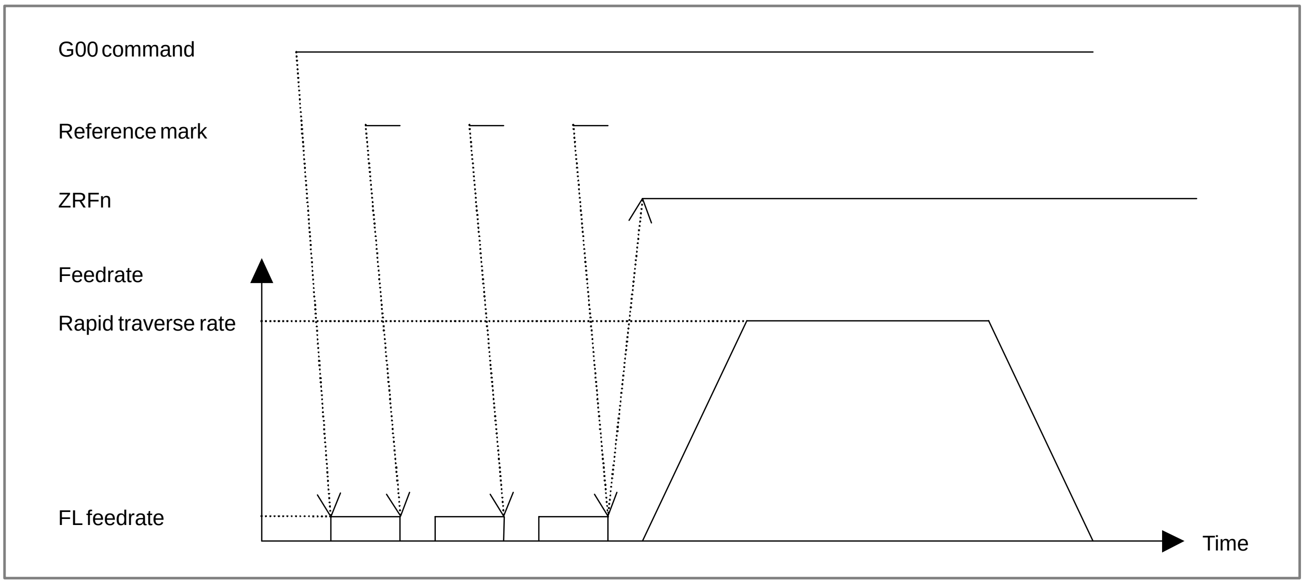


Fig. 4.8 (b) Time chart for reference position establishment (G00)

The specifications for the steps of detecting reference marks and establishing the reference position (steps <1> to <3> above) are the same as for the conventional linear scale with absolute addressing reference marks. The restrictions are also the same.
For details, see Section 4.7, ”Linear Scale I/F with Absolute Address Referenced Mark (A/B Phase)/Linear Scale with Distance–Coded Reference Marks (Serial)”.

- **Tool path**

The tool path in the G00 command is explained below.

(1) When no axis requires the reference position establishment operation

When the reference position has already been established for all specified axes, the reference position establishment operation is not performed.
For example, suppose that the reference position is already established for the X–, Y–, and Z–axes, and that G00 Xxx Yyy ZZz; is specified. Then, normal rapid traverse operation takes place. The tool path follows the setting in bit 1 (LRP) of parameter No. 1401.

(2) When all axes require the reference position establishment operation

Suppose that the reference position is not established for the X–, Y–, and Z–axes and that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the figure below.
Operation 1 in the figure establishes the reference position. During the establishment operation, the tool path is always of the non–linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.
Operation 2 performs positioning to a specified end point. During this operation, the tool path follows the setting in bit 1 (LRP) of parameter No. 1401.
After operation 1 is completed for all axes, operation 2 starts.

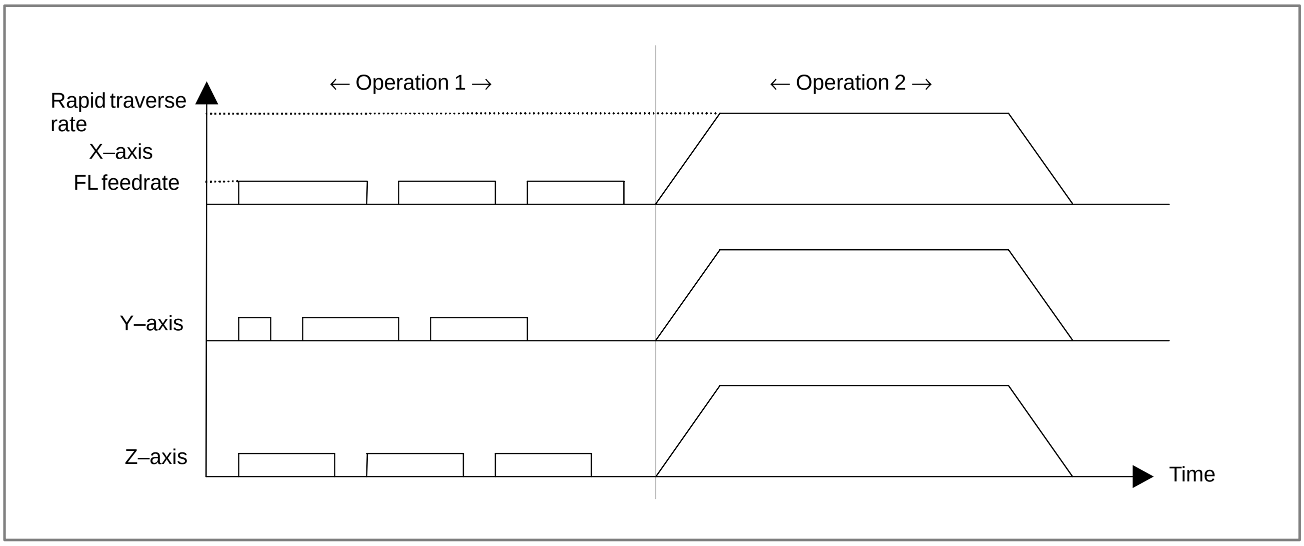


Fig. 4.8 (c) When the reference position is established for all axes

(3) When some axes require the reference position establishment operation and others do not require the establishment operation

For example, suppose that the reference position is already established for the X–axis and that the reference position is not yet established for the Y– and Z–axes. Also suppose that G00 Xxx Yyy Zzz; is specified. The operation in this case is shown in the figure below.
In operation 1 in the figure, movement to a specified position is made along the X–axis for which the reference position is already established. For the Y– and Z–axes for which no reference position is established, the reference position is established. During the establishment operation, the tool path is always of the non–linear interpolation type regardless of the setting in bit 1 (LRP) of parameter No. 1401.
In operation 2, positioning to a specified end point is performed along the Y– and Z–axes. The tool path along the Y– and Z–axes then follows the setting in bit 1 (LRP) of parameter No. 1401. Because positioning to the specified position is already made along the X–axis, no movement is made along the X–axis.
After operation 1 is completed for all axes, operation 2 starts.

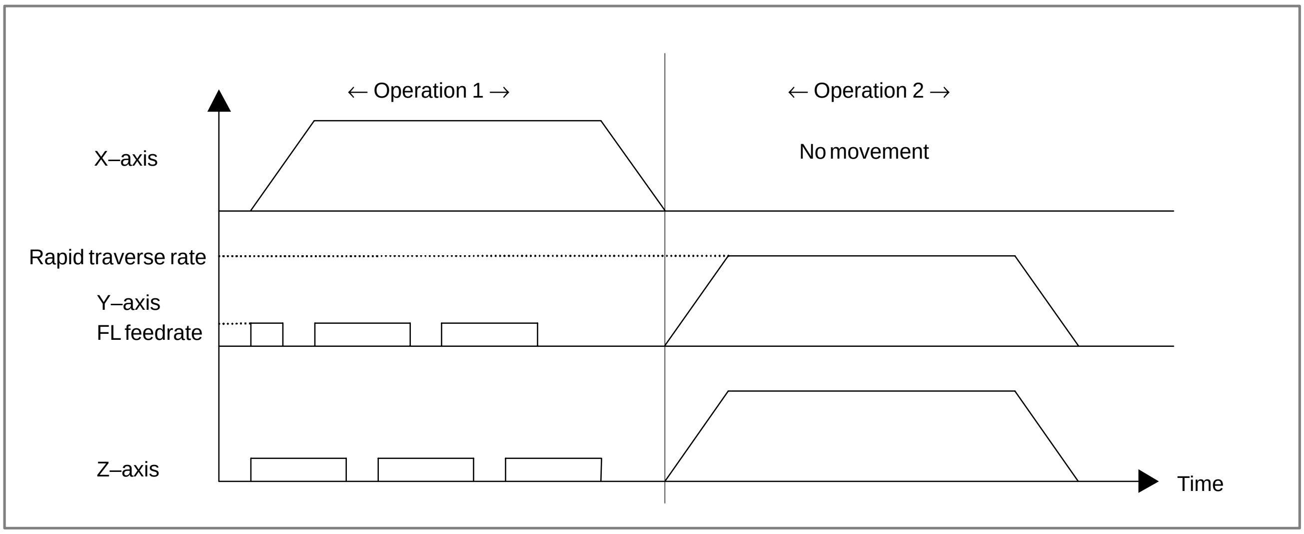


Fig. 4.8 (d) When an axis does not require the reference position establishment operation and others require the establishment operation

- **Absolute command and incremental command**

As explained in step <4>, after the reference position has been established, positioning to a specified end point is performed. This operation is explained below.

(1) When an absolute command is specified

Movement to the end point in the new coordinate system that has been established in step <3> is made. For example, when G90 G00 Xxx.Yyy.; (for the M series) or G00 Xxx.Zyy.; (for the T series) is specified, the tool moves as indicated with the bold line in the figure below. Note that, however, the figure shows the positional relationship among points and that the intermediate tool path is not always of the linear interpolation type.

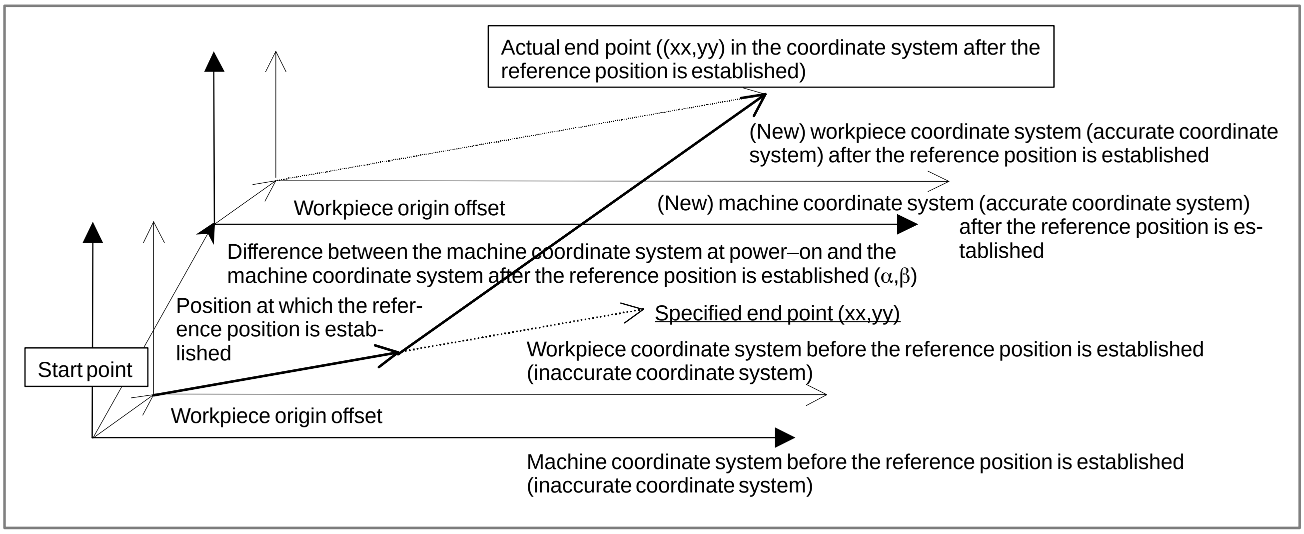


Fig. 4.8 (e) Operation when an absolute command is specified

(2) When an incremental command is specified
Movement is made along each axis by a specified distance. (The movement is indicated with the bold line in the figure below. Note that the intermediate tool path is not always of the linear interpolation type.) The coordinates at the end point are those in a newly established coordinate system. When there is a difference (α, β) between the newly established coordinate system and the old coordinate system, the coordinates at the end point are shifted by (α, β). For example, when G91 G00 X100.Y30.; (M series) or G00 W100.U30.; (T series) is specified from the position of which absolute coordinates are (0,0), the distance of movement along each axis is (100,30), and the coordinates of the end point are (100–α, 30–β).

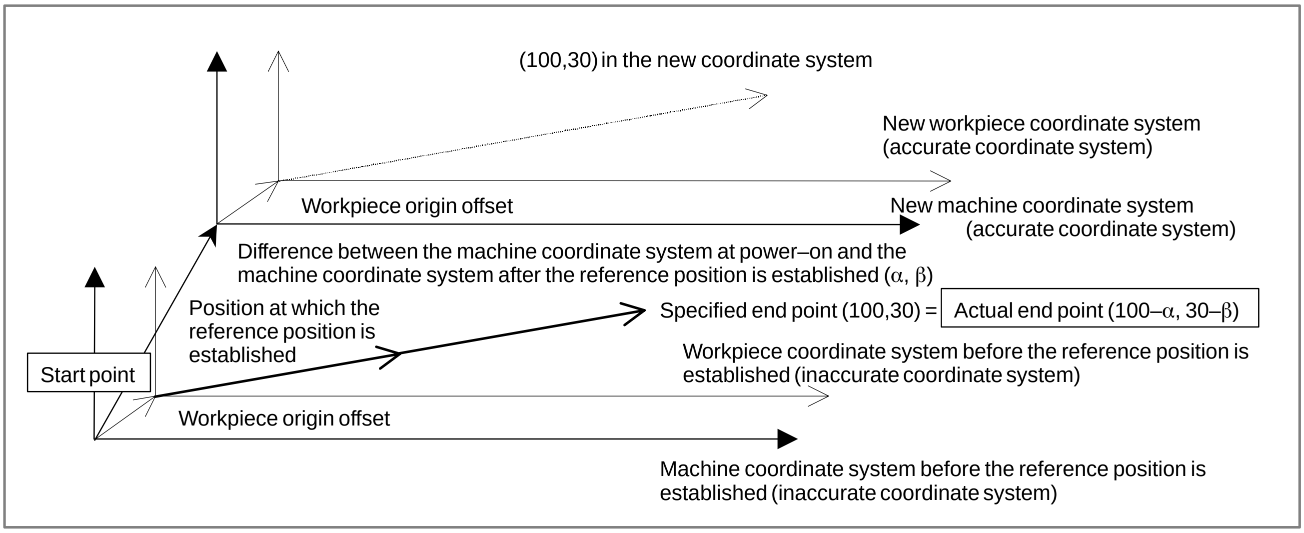


Fig. 4.8 (f) Operation when an incremental command is specified

- **When a short distance is specified**

When a short distance is specified, the end point can be reached before three or four reference marks are passed. Even in such a case, the CNC makes a movement at the FL feedrate while detecting reference marks, but the CNC does not establish the reference position even when the end point is reached.
If the reference position is not established, the next G00 command causes the reference position establishment operation again. The CNC does not use data on the reference marks detected through the previous movement and detects three or four reference marks again to establish the reference position.

- **Interruption by feed hold**

When a feed hold has been applied during reference position establishment operation, the reference position establishment operation is not performed after execution is restarted. After the restart, non–linear type positioning is performed. In this block, the reference position is not established, so reference position establishment operation is performed again when the next G00 command is specified.

- **Interruption by reset or emergency stop**

When a reset or emergency stop is applied during reference position establishment operation, the reference position establishment operation is interrupted. Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.

- **When an illegal reference mark interval is detected**

If a correct reference mark interval cannot be detected for a cause, the tool is positioned to the end point without establishing the reference position. Therefore, the machine position, absolute coordinates, and machine coordinates of the end point are set as if a G00 command without reference position establishment operation were executed. However, the intermediate tool path is different from the tool path obtained by executing the G00 command.
Since the reference position is not established, reference position establishment operation is performed again when the next G00 command is specified.
Even when the CNC detects an illegal reference mark interval, it does not issue the P/S090 alarm.

Reference position establishment by jog feed

- **Activation condition**

If the following conditions are satisfied, reference position establishment operation is performed automatically:

<1> For an axis for which no reference position has been established, feed axis direction selection signal +Jn or –Jn (G100, G102) is input in jog mode.

<2> The move direction in <1> above matches the reference position return direction set by bit 5 (ZMI) of parameter No. 1006.

<3> The specified axis is not in the following modes:
- Coordinate system rotation
- Mirror image (mirror image by signal or setting)
- Programmable mirror image (M series)
- Scaling (M series)
- Mirror image of facing tool posts (T series)

> **NOTE**
> If all the above conditions are not satisfied, reference position establishment operation is not performed, and the same operation as normal jog feed is performed.

- **Operation**

The reference position establishment procedure is explained below.

<1> When the feed axis direction selection signal is set to 1, the tool starts moving at the reference position return FL feedrate (parameter No. 1425).

<2> Upon detection of a reference mark on the scale, the tool is stopped briefly then fed at the reference position return FL feedrate.

<3> Step <2> above is repeated until three or four reference marks are detected on the scale. Then, the reference position is established, and the reference position established signal (ZRF1, ZRF2, ZRF3, etc.) is set to 1.

<4> The tool is fed in the direction selected by the feed axis direction selection signal at the jog feedrate.

When the feed axis direction selection signal is set to 0 during steps <2> to <4>, feed operation stops. When the feed axis direction selection signal is set to 1 again, the reference position is established.

A time chart for the above procedure is shown below.

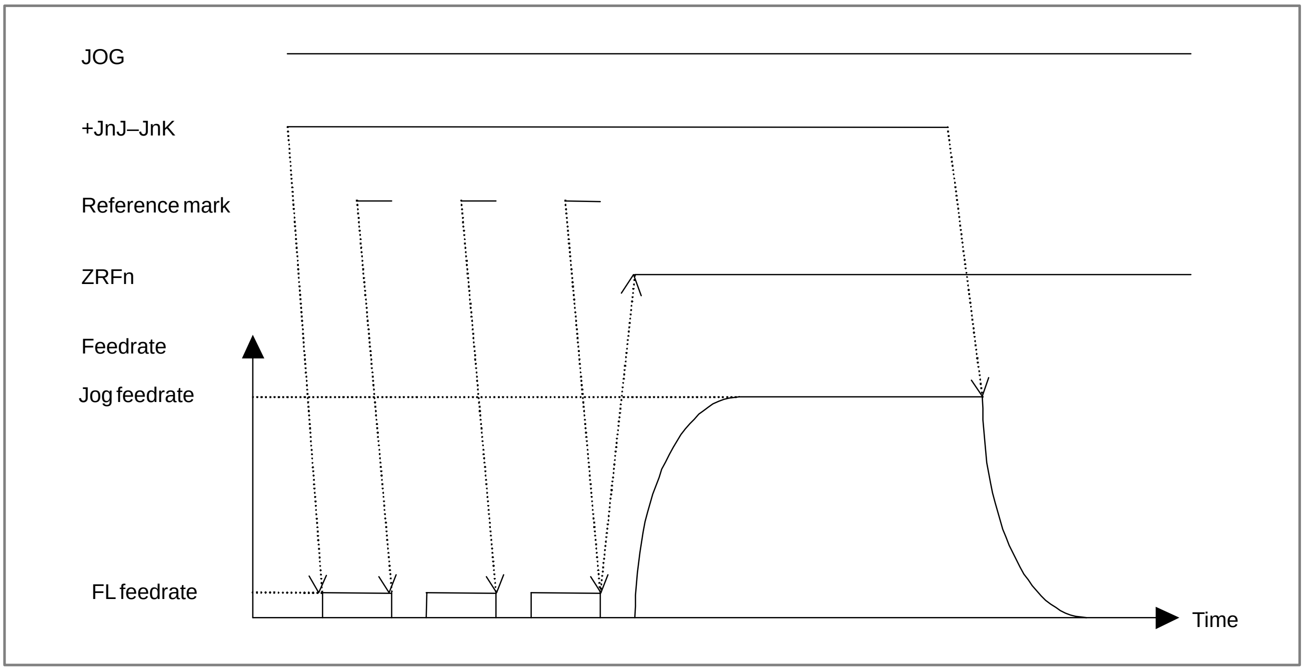


Fig. 4.8 (g) Time chart for reference position establishment (jog feed)

- **Interruption by a feed axis direction selection signal**

If feed axis direction selection signal +Jn (–Jn) is set to 0 during reference position establishment, the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis direction selection signal +Jn (–Jn) is set to 1 again, the reference position establishment operation is resumed, and the reference position is established after three or four reference marks are detected.

- **Interruption by reset or emergency stop**

When a reset or emergency stop is applied during reference position establishment operation, axis movement stops, and the reference position establishment operation is interrupted. In this case, data on the reference marks detected through the previous movement is not used for the next reference establishment operation. When the feed axis direction selection signal +Jn (–Jn) is set to 1 again, the reference position is established after three or four reference marks are detected.

- **When an illegal reference mark interval is detected**

If a correct reference mark interval cannot be detected for a cause, reference position establishment operation is repeated until the reference position is established. Even when the CNC detects an illegal reference mark interval, it does not issue the P/S090 alarm.

Caution

CAUTION

1 PMC axis control
In rapid traverse (axis control command 00h) and continuous feed (axis control command 06h) under PMC axis control, the reference position is not established.

2 Rapid traverse by other than G00
In rapid traverse operation generated automatically by a command such as a canned cycle command, the reference position is not established.

3 Reference position establishment operation
The specifications for detecting reference marks and establishing the reference position are the same as those of the conventional linear scale with absolute addressing reference marks. The parameter setting method and restrictions are also the same as those of the conventional linear scale with absolute addressing reference marks.
For details, see Section 4.7, "Linear Scale I/F with Absolute Address Referenced Mark (A/B Phase)/Linear Scale with Distance–Coded Reference Marks (Serial)".

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1818						DG0		

[Data type] Bit axis

DG0 Reference position establishment by the G00 command and jog feed is:

0 : Disabled.
1 : Enabled.

5 AUTOMATIC OPERATION

5.1 CYCLE START/ FEED HOLD

General

- **Start of automatic operation (cycle start)**

When automatic operation start signal ST is set to 1 then 0 while the CNC is in memory (MEM) mode, DNC operation mode (RMT), or manual data input (MDI) mode, the CNC enters the automatic operation start state then starts operating.

Signal ST, however, is ignored in the following cases:

1. When the mode is other than MEM, RMT, or MDI
2. When the feed hold signal (*SP) is set to 0
3. When the emergency stop signal (*ESP) is set to 0
4. When the external reset signal (ERS) is set to 1
5. When the reset and rewind signal (RRW) is set to 1
6. When MDI RESET key is pressed
7. When the CNC is in the alarm state
8. When the CNC is in the NOT READY state
9. When automatic operation is executing
10. When the program restart signal (SRN) is 1
11. When the CNC is searching for a sequence number.

The CNC enters the feed hold state and stops operation in the following cases during automatic operation:

1. When the feed hold signal (*SP) is set to 0
2. When the mode is changed to manual operation mode (JOG, INC, HND, REF, TJOG, or THND).

The CNC enters the automatic operation stop state and stops operating in the following cases during automatic operation:

1. When a single command block is completed during a single block operation
2. When operation in manual data input (MDI) mode has been completed
3. When an alarm occurs in the CNC
4. When a single command block is completed after the mode is changed to other automatic operation mode or memory edit (EDIT)

The CNC enters the reset state and stops operating in the following cases during automatic operation:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

The state of the CNC (automatic operation start, feed hold , automatic operation stop, or reset) is posted to the PMC with status output signals OP, SPL, and STL. See the table in the “Signal” section for details.

- **Halt of automatic operation (feed hold)**

When the feed hold signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to 0 and feed hold lamp signal SPL is set to 1. Re–setting signal *SP to 1 in itself will not restart automatic operation. To restart automatic operation, first set signal *SP to 1, then set signal ST to 1 and then to 0.

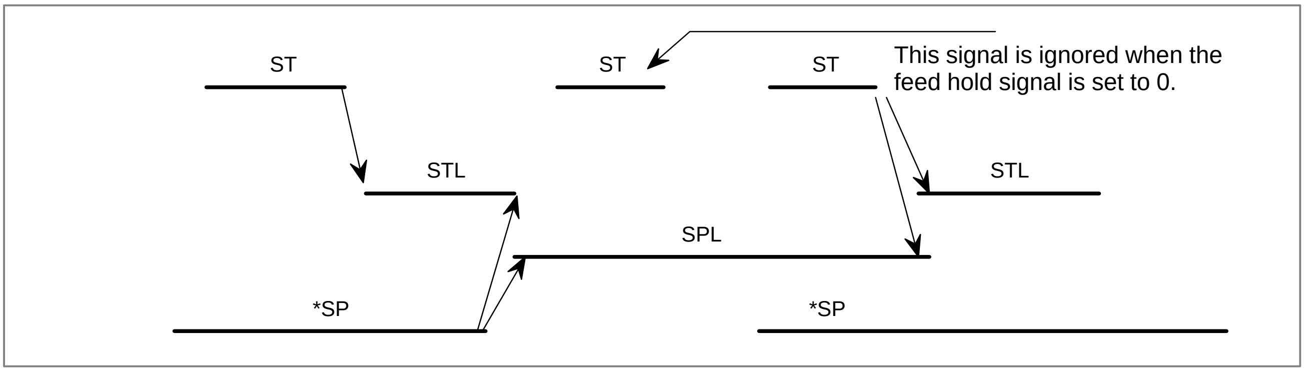


Fig. 5.1 Time chart for automatic operation

When signal *SP is set to 0 during the execution of a block containing only the M, S, T, or B function, signal STL is immediately set to 0, signal SPL is set to 1, and the CNC enters the feed hold state. If the FIN signal is subsequently sent from the PMC, the CNC executes processing up until the end of the block that has been halted. Upon the completion of that block, signal SPL is set to 0 (signal STL remains set to 0) and the CNC enters the automatic operation stop state.

(a) During threading
When signal *SP is set to 0 during threading, the CNC enters the feed hold state after executing a non–threading block after the threading blocks.
When signal *SP is set to 0 during threading with the G92 command (threading cycle), signal SPL is immediately set to 1 but operation continues up until the end of the retraction block following threading.
When signal *SP is set to 0 during threading with the G32 (M series: G33) command, signal SPL is immediately set to 1 but operation continues until the end of a non–threading block following the threading blocks. (Stopping feeding during threading is dangerous because the amount of cutting will increase.)

(b) During tapping in a canned cycle (G84)
When signal *SP is set to 0 during tapping in a canned cycle (G84), signal SPL is immediately set to 1 but operation continues until the tool returns to the initial level or R point level after the completion of tapping.

(c) When a macro instruction is being executed
Operation stops after the currently executing macro instruction has been completed.

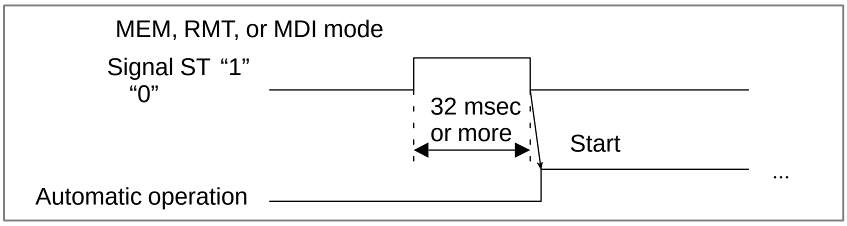
Signal

Cycle start signal ST <G007#2>

[Classification] Input signal

[Function] Starts automatic operation.

[Operation] When signal ST is set to 1 then 0 in memory (MEM) mode, DNC operation mode (RMT) or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.

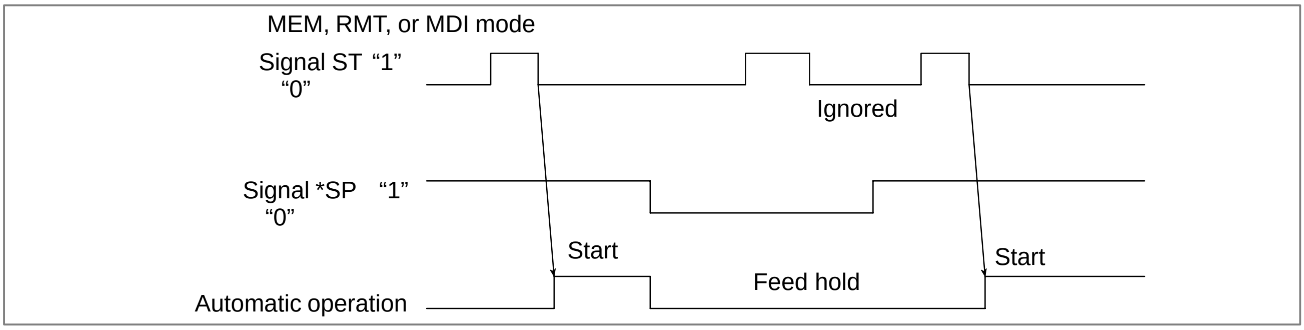


Feed hold signal *SP <G008#5>

[Classification] Input signal

[Function] Halts automatic operation.

[Operation] When signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. Automatic operation cannot be started when signal *SP is set to 0.



Automatic operation signal OP <F000#7>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation is in progress.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Cycle start lamp signal STL <F000#5>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation start is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Feed hold lamp signal SPL <F000#4>

[Classification] Output signal

[Function] Notifies the PMC that feed hold state is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.
Signals OP, STL, and SPL are the signals to inform PMC of the operation condition of CNC.

Table 5.1 Status of operation

Signal name / State of the operation	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start state	1	0	1
Feed hold state	0	1	1
Automatic operation stop state	0	0	1
Reset state	0	0	0

· Cycle start state
The CNC is executing memory operation or manual data input operation commands.

· Feed hold state
The CNC is not executing memory operation nor manual data input operation commands while the commands to be executed remain.

· Automatic operation stop state
Memory operation or manual data input operation has been completed and stopped.

· Reset state
The automatic operation has been forcibly terminated.

NOTE
If the sequence number search is performed through MDI panel during Memory mode (MEM), the signal OP turns to "1".

- **Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G007						ST		
G008			*SP					

	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP		STL	SPL				

Alarm and message

- **Self–diagnosis information**

During automatic operation, the machine may sometimes show no movement while no alarm is detected. In that case, the CNC may be performing processing or waiting for the occurrence of an event. The state of the CNC can be obtained using the CNC self–diagnosis function (diagnosis numbers 000 to 015).

Detailed information on the automatic operation stop or feed hold state can also be displayed (diagnosis numbers 020 to 025).

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

5.2 RESET AND REWIND

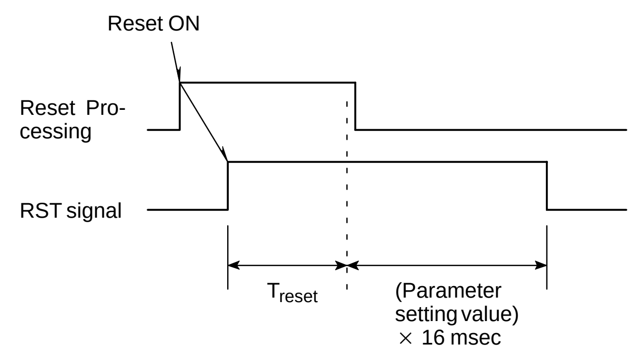
General

The CNC is reset and enters the reset state in the following cases:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

When the CNC is reset, the resetting signal (RST) is output to the PMC. The resetting signal (RST) is set to 0 when the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released.

RST signal output time = T_{reset} (Reset processing time) + (parameter setting value) $\times$ 16 msec.



> **CAUTION**
> T_{reset} requires at least 16 msec. This time will be longer on optional configurations.

When the CNC is reset during automatic operation, automatic operation is stopped and tool movement along the controlled axis is decelerated and stopped(*1). When the CNC is reset during the execution of the M, S, T, or B function, signal MF, SF, TF, or BF is set to 0 within 100 ms.

Tool movement along the controlled axis is also decelerated and stopped(*1) in manual operation (jog feed, manual handle feed, incremental feed, or etc).

> **CAUTION**
> *1 When the emergency stop signal (*ESP) is set to 0, the tool is stopped by an emergency stop.

Bit 6 (CLR) of parameter No. 3402 is used to select whether the CNC internal data (such as modal G codes) is cleared or reset when the CNC is reset. Refer to the Appendix E, ”Status when turning on power, when cleared, and when reset” in the Operator’s manual for the state of the internal data when cleared or reset.

The following parameters are also used to select how to handle processing for CNC data when the CNC is reset.

- Bit 7 (MCL) of parameter No. 3203
  Whether programs created in MDI mode are erased or stored
- Bit 6 (CCV) of parameter No. 6001
  Whether custom macro variables #100 to #149 are cleared or stored
- Bit 7 (CLV) of parameter No. 6001
  Whether custom macro local variables #1 to #33 are cleared or stored

● Reset & Rewind

When the reset & rewind signal (RRW) is set to 1, reset is performed and the following rewinding operation is also performed.

1. When the DNC operation mode, and a portable tape reader is connected as the current input/output device, the tape reader is rewound.

   While the tape reader is being rewound, the rewinding–in–progress signal (RWD) is output. This signal goes 0 when the tape reader has been rewound.

2. In cases other than case 1, the head of the selected main program is searched for. Setting RWM, bit 2 of parameter no. 3001, determines whether the rewinding–in–progress signal is output.

   When RWM is set to 1:

   The rewinding–in–progress signal is output. It is set to 1, then set to 0 after about 100 ms. Since searching for the main program in memory takes little time, when the rewinding–in–progress signal (RWD) is set to 0, the main program has already been searched for.

Signal

External reset signal ERS<G008#7>

[Classification] Input signal

[Function] Reset the CNC.

[Operation] Turning the signal ERS to 1 resets the CNC and enters the reset state. While the CNC is reset, the resetting signal RST turns to 1.

Reset & rewind signal RRW<G008#6>

[Classification] Input signal

[Function] CNC is reset and a program under an automatic operation is rewound.

[Operation] As described in the item, “Reset & Rewind”.

Resetting signal RST <F001#1>

[Classification] Output signal

[Function] Notifies the PMC that the CNC is being reset. This signal is used for reset processing on the PMC.

[Output condition] This signal is set to 1 in the following cases:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset & rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

This signal is set to 0 in the following case:
When the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released and the CNC is reset

Rewinding signal RWD <F000#0>

[Classification] Output signal

[Function] Notifies the PMC that the CNC is being rewound.

[Output condition] As described in the item, “Reset and Rewind”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G008	ERS	RRW						

	#7	#6	#5	#4	#3	#2	#1	#0
F000								RWD
F001							RST	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

[Data type] Bit

RWM RWD signal indicating that rewinding is in progress

0 : Output only when the tape reader is being rewound by the reset and rewind signal RRW

1 : Output when the tape reader is being rewound or a program in memory is being rewound by the reset and rewind signal RRW

3017	Output time of reset signal RST

[Data type] Byte

[Unit of data] 16 ms

[Valid data range] 0 to 255

To extend the output time of reset signal RST, the time to be added is specified in this parameter.
RST signal output time = time required for reset + parameter value × 16 ms

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL							

[Data type] Bit

MCL Whether a program prepared in the MDI mode is cleared by reset
0: Not deleted
1: deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR						

[Data type] Bit

CLR Reset key on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

[Data type] Bit

CCV Custom macro's common variables Nos. 100 through 149
0: Cleared to "vacant" by reset
1: Not cleared by reset

CLV Custom macro's local variables Nos. 1 through 33
0: Cleared to "vacant" by reset
1: Not cleared by reset

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
	OPERATOR'S MANUAL (T series) (B–64114EN)	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
	OPERATOR'S MANUAL (T series) (B–64134EN)	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

5.3 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine or viewing the position display change without running the machine.

5.3.1 Machine Lock

General

The change of the position display can be monitored without moving the machine.
When all–axis machine lock signal MLK, or each–axis machine lock signals MLK1 to MLK4 are set to 1, output pulses (move commands) to the servo motors are stopped in manual or automatic operation. The commands are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the commands are correct by monitoring the position display.

Signal

All–axis machine lock signal MLK <G044#1>

[Classification] Input signal

[Function] Places all controlled axes in the machine lock state.

[Operation] When this signal is set to 1, pulses (move commands) are not output to the servo motors for all axes in manual or automatic operation.

Each–axis machine lock signals MLK1 to MLK4 <G108#0 to #3>

[Classification] Input signal

[Function] Place the corresponding controlled axes in the machine lock state.
These signals are provided for each controlled axis. The signal number corresponds to the number of the controlled axis.

MLK 1

1. Machine lock for the 1st axis
2. Machine lock for the 2nd axis
3. Machine lock for the 3rd axis

⋮

[Operation] When these signals are set to 1, pulses (move commands) are not output to the servo motors for the corresponding axes (1st to 4th) in manual or automatic operation.

All–axis machine lock check signal MMLK <F004#1>

[Classification] Output signal

[Function] Notifies the PMC of the state of the all–axis machine lock signal.

[Output condition] This signal is set to 1 in the following case:
– When all–axis machine lock signal MLK is set to 1

This signal is set to 0 in the following case:
– When all–axis machine lock signal MLK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G044							MLK	
G108					MLK4	MLK3	MLK2	MLK1
F004							MMLK	

Note

NOTE

1 **Automatic operation in the machine lock state (M, S, T, and B commands)**
Machine lock applies only to move commands along controlled axes. Updating modal G codes or setting a coordinate system is performed normally. M, S, T, and B (2nd auxilialy function) commands are also performed normally.

2 **Reference position return in the machine lock state (G27, G28, and G30)**
When the reference position return command (G28), or 2nd to 4th reference position return command (G30), is executed for an axis in the machine lock state, distribution and position updating are performed. The tool, however, is not returned to the reference position. The reference position return completion signals (ZP1 to ZP4) are not output.
The reference position return check command (G27) is ignored in the machine lock state.

3 **Turning on/off the machine lock signal during movement along an axis**
When the machine lock signal for an axis is set to 1 during movement along the axis that is not in the machine lock state, the axis is immediately placed in the machine lock state and output pulses (move commands) to the servo motor are stopped. The tool is decelerated and stopped with the automatic acceleration/deceleration function.
On the other hand, when the machine lock signal for an axis is set to 0 during distribution of the move command along the axis in the machine lock state, pulse (move command) output for the axis is immediately restarted. The tool is accelerated with the automatic acceleration/deceleration function.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK

5.3.2 Dry Run

General

Dry run is valid only for automatic operation.
The tool is moved at a constant feedrate(*1) regardless of the feedrate specified in the program. This function is used, for example, to check the movement of the tool without a workpiece.

> **CAUTION**
> This feedrate depends on the specified parameters, the manual rapid traverse switching signal (RT), manual feedrate override signals (*JV0 to *JV15), and whether the command block specifies rapid traverse or cutting feed, as listed in the table below.

Manual rapid traverse switching signal (RT)	Program command	
	Rapid traverse	Feed
1	Rapid traverse rate	Dry run feedrate × JV_{max} *2
0	Dry run speed × JV,or rapid traverse rate *1	Dry run feedrate × JV *2

Max. cutting feedrate . . Setting by parameter No.1422
Rapid traverse rate . . . . Setting by parameter No.1420
Dry run feedrate . . . . . . Setting by parameter No.1410
JV . . . . Manual feedrate override
JV_{max} . . . . Maximum value of manual feedrate override
*1:Dry run feedrate × JV when parameter RDR (bit 6 of No. 1401) is 1.
Rapid traverse rate when parameter RDR is 0.
*2 Clamped by max. cutting feedrate.

Signal

Dry run signal DRN <G046#7>

[Classification] Input signal

[Function] Enables dry run.

[Operation] When this signal is set to 1, the tool is moved at the feedrate specified for dry run.
When this signal is set to 0, the tool is moved normally.

> **CAUTION**
> When the dry run signal is changed from 0 to 1 or 1 to 0 during the movement of the tool, the feedrate of the tool is first decelerated to 0 before being accelerated to the specified feedrate.

Dry run check signal MDRN <F002#7>

[Classification] Output signal

[Function] Notifies the PMC of the state of the dry run signal.

[Output condition] This signal is set to 1 in the following case:
– When dry run signal DRN is set to 1

This signal is set to 0 in the following case:
– When dry run signal DRN is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046	DRN							
F002	MDRN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR					

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84; rigid tapping)

0 : Enabled
1 : Disabled

RDR Dry run for rapid traverse command

0 : Disabled
1 : Enabled

1410	Dry run rate

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the dry run rate when the manual feedrate is overridden by 100%.

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	6 – 100000
Inch machine	0.1 inch/min	30 – 96000	6 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead. (M series)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.5.4	Dry run
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.5.4	Dry run
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.5.4	Dry run
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.5.4	Dry run

5.3.3 Single Block

General

Single block operation is valid only for automatic operation.
When the single block signal (SBK) is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to 0, normal automatic operation is restored.

Single block operation during the execution of custom macro statements depends on the setting of bit 5 (SBM) of parameter No. 6000, as follows:

SBM = 0, SBV = 0: Operation does not stop in the custom macro statements but stops once the next NC command has been executed.

SBM = 1: Operation stops after each block in the custom macro statements.

SBV = 1: Single block operation in custom macro statements is suppressed using macro system variable #3003. Execution of custom macro statements is stopped after block execution.

When the CNC is in the automatic operation stop state during single block operation, the mode can be changed to manual data input (MDI), manual handle feed (HNDL), incremental feed (INC), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK <G046#1>

[Classification] Input signal

[Function] Enables single block operation.

[Operation] When this signal is set to 1, single block operation is performed. When this signal is set to 0, normal operation is performed.

Single block check signal MSBK <F004#3>

[Classification] Output signal

[Function] Notifies the PMC of the state of the single block signal.

[Output condition] This signal is set to 1 in the following case:
– When single block signal SBK is set to 1

This signal is set to 0 in the following case:
– When single block signal SBK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046							SBK	
F004					MSBK			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000	SBV		SBM					

[Data type] Bit

SBM Custom macro statement
0: Not stop the single block
1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

> **NOTE**
> This bit is invalid when bit 0 (NOP) of parameter No. 6000 is set to 1. (M series)

SBV Custom macro statement
0 : Not stop the single block
1 : Stops the single block

To control single blocks in custom macro statements using system variable #3003, use this parameter to enable or disable single blocks in custom macro statements.

This bit is valid when bit 5 (SBM) of parameter No. 6000 is set to 0.

Caution

> **CAUTION**
> 1 **Operation in thread cutting**
> When the SBK signal turns to “1” during thread cutting, operation stops after execution of the first non-thread cutting block after the thread cutting command.
> 2 **Operation in canned cycle**
> When the SBK signal turns to “1” during canned cycle operation, the operation stops at each positioning, approach, drilling and retreat instead of the end of the block. The SPL signal turns to “1” while the STL signal turns to “0”, showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals turn to “0” and the operation is stopped.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.5.5	Single block
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.5.5	Single block
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.5.5	Single block
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.5.5	Single block

5.4 MANUAL ABSOLUTE ON/OFF

General

This function selects whether the movement of the tool with manual operation (such as jog feed and manual handle feed) is counted for calculating the current position in the workpiece coordinate system. A check signal is also output to indicate whether the manual absolute function in the CNC is turned on or off.

When manual absolute turns on (manual absolute signal *ABSM =0)

When manual operation interrupts during automatic operation:

i) At the end of the block where manual operation interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command.

ii) In subsequent blocks, the parallel-moved tool position remains unchanged until an absolute command block appears. Therefore, if all blocks are programmed by incremental commands, the tool keeps the parallel-moved position until machining ends.

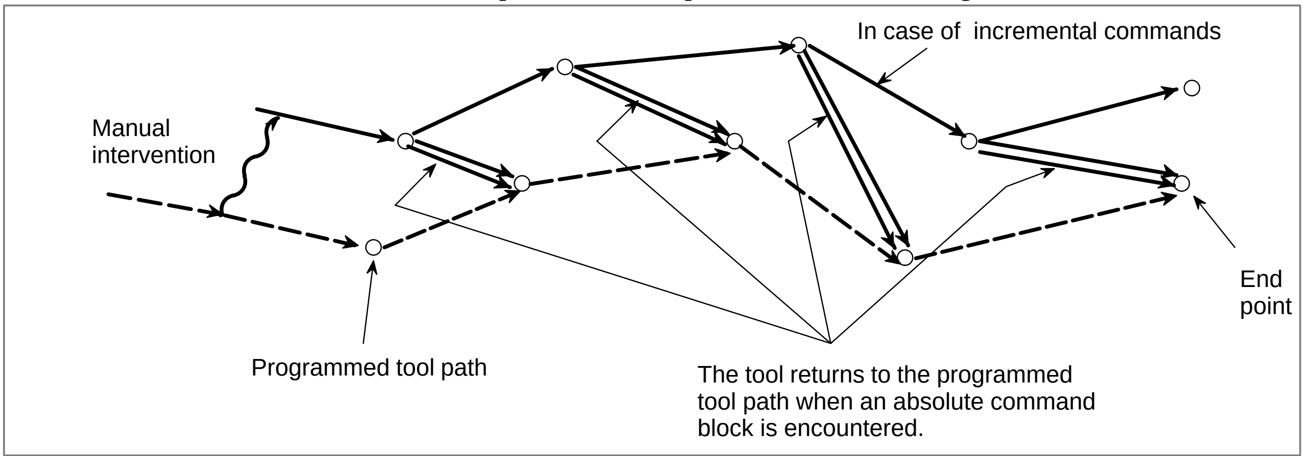


Fig. 5.4 (a) Manual absolute ON

> **CAUTION**
> If the machining end position has shifted by the manual move amount because all blocks are programmed by incremental commands only, the present position is displayed shifted by the manual move amount.

When manual absolute turns off (manual absolute signal *ABSM=1)

The manual move amount is not counted to the present position on the workpiece coordinate system. The present position display on the CRT includes the manual move amount. The display is reset to the initial value (before manual operation) when the control is reset, or when operation in the automatic operation mode MEM, RMT, or MDI is started after the manual operation.

During automatic operation, if manual intervention of a block interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command at the end point of that block, as well as at the end point of subsequent blocks.

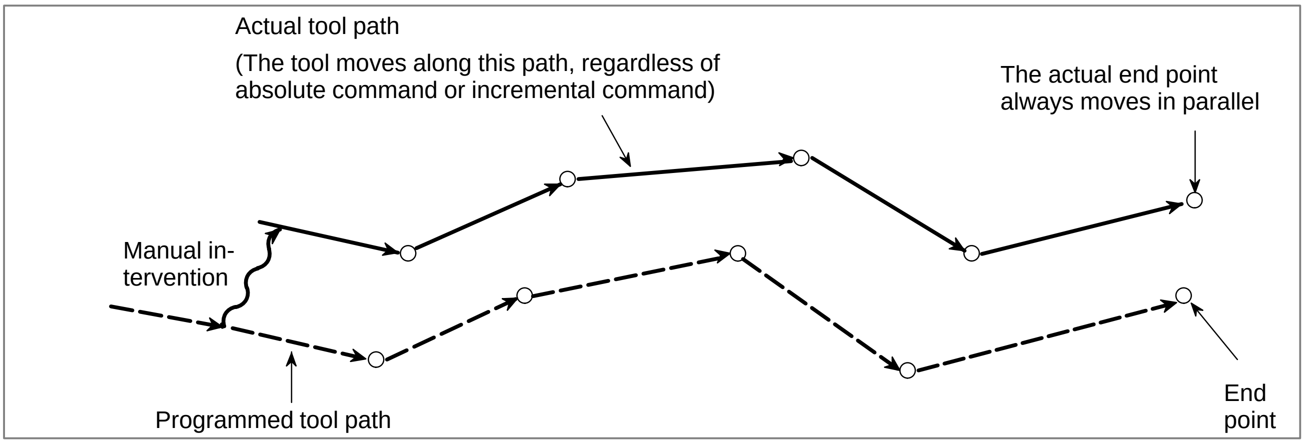


Fig. 5.4 (b) Manual absolute OFF

The present position display at the finish of the operation shows an end point value on the program as if manual intervention had not been executed. However, the tool position moves in parallel.

Signal

Manual absolute signal *ABSM <G006#2>

[Classification] Input signal

[Function] Turns the manual absolute function on or off.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Turns off the manual absolute function.

When this signal is set to 0, the control unit operates as follows:
– Turns on the manual absolute function.

Manual absolute check signal MABSM <F004#2>

[Classification] Output signal

[Function] Notifies the PMC of the state of the manual absolute signal.

[Output condition] This signal is set to 1 in the following case:
– When the manual absolute signal *ABSM is set to 0

This signal is set to 0 in the following case:
– When manual absolute signal *ABSM is set to 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006						*ABSM		
F004						MABSM		

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.3.5	Manual absolute ON/OFF
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.3.5	Manual absolute ON/OFF
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.3.5	Manual absolute ON/OFF
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.3.5	Manual absolute ON/OFF

5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP

General

When a slash followed by a number (/n, where n = 1 to 9) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information contained in the block for which /n, corresponding to signal BDTn, is specified is ignored (from /n to the end of the block).

(Example)   /2 N123 X100. Y200. ;

Input signal	Code specified at the head of a block
BDT1	/ or /1 (Note 1)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

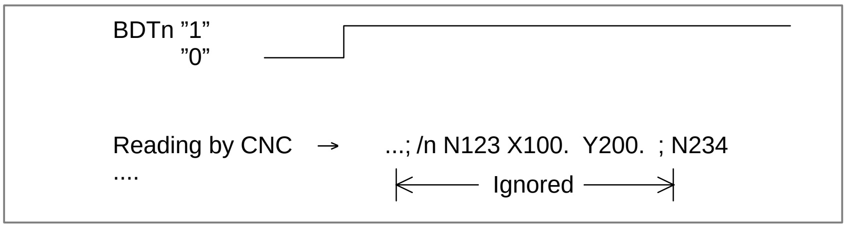
NOTE
1 Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.

(Example)

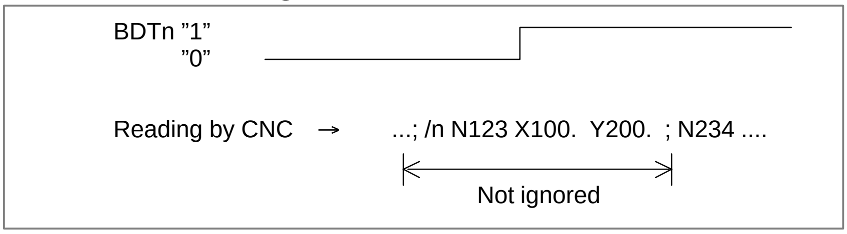
//3 N123 X100. Y200. ; — Invalid
/1 /3 N123 X100. Y200. ;— Valid

The following figures show the relationship between the timing, when optional block skip signals (BDT1 to BDT9) are set to 1, and the ignored information:

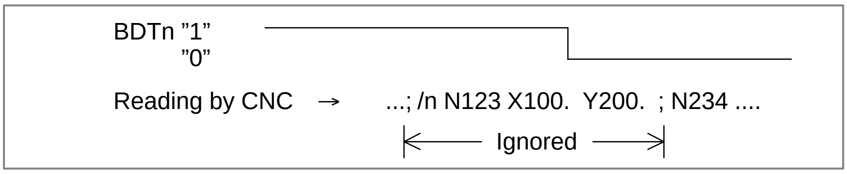
1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.



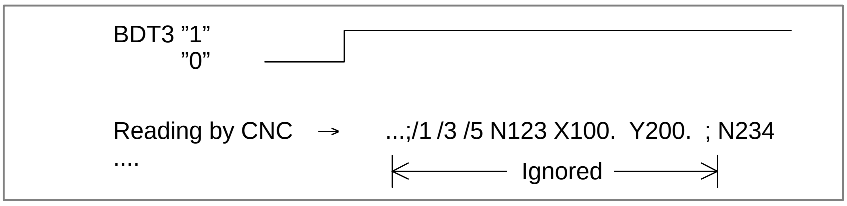
2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.



3. When BDTn, currently set to 1, is set to 0 while the CNC is reading a block containing /n, the block is ignored.



4. When two or more optional block skip switches are specified in a block and BDTn, corresponding to one of them, is set to 1, the block is ignored.



Signal

Optional block skip signals BDT1 <G044#0> BDT2 to BDT9 <G045>

[Classification] Input signal

[Function] Select whether a block containing /n is to be executed or ignored.

[Operation] During automatic operation, a block containing /n in the program is ignored when the corresponding optional block skip signal is set to 1. It is executed normally when the signal is set to 0.

Optional block skip check signals MBDT1 <F004#0> MBDT2 to MBDT9 <F005>

[Classification] Output signal

[Function] Notify the PMC of the states of the optional block skip signals BDT1 to BDT9. Nine signals are provided, corresponding to the nine optional block skip signals. Signal MBDTn corresponds to signal BDTn.

[Output condition] Signal MBDTn is set to 1 in the following case:
– When the corresponding optional block skip signal (BDTn) is set to 1

Signal MBDTn is set to 0 in the following case:
– When the corresponding optional block skip signal (BDTn) is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G044								BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
F004								MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2

Note

NOTE

1 This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip signal is set.
Programs stored in memory can be output, regardless of how the optional block skip signals are set.
Optional block skip is effective even during sequence number search operation.

2 Position of a slash
A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

3 TV and TH check
When an optional block skip signal is "1". TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is "0".

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.12.2	Program section configuration
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.12.2	Program section configuration
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.12.2	Program section configuration
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.12.2	Program section configuration

5.6 SEQUENCE NUMBER COMPARISON AND STOP

General

During program execution, this function causes a single block stop right after a block with a specified sequence number is executed.

To use this function, first specify the program number (1 to 9999) of a program that contains a sequence number where operation is to be stopped and the sequence number on the setting data screen:
With this setting, a single block stop occurs after the execution of the block with the specified sequence number during automatic operation.

Parameter

Setting data

– SEQUENCE STOP (PROGRAM NO.)
  Specify the program number (1 to 9999) of a program to which a sequence to be stopped belongs.

– SEQUENCE STOP (SEQUENCE NO.)
  Specify the sequence number (1 to 99999) of a sequence to be stopped.

Note

NOTE
After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.4	Sequence Number Comparison and Stop
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.8	Sequence Number Comparison and Stop
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.4	Sequence Number Comparison and Stop
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.7	Sequence Number Comparison and Stop

5.7 PROGRAM RESTART

General

A program may be restarted at a block by specifying the sequence number of the block, after automatic operation is stopped because of a broken tool or for holidays. This function can also be used as a high–speed program check function.

There are two types of restart methods.

P type: Restart after a tool is broken down
Q type: Restart after holidays

Signal

Program restart signal SRN<G006#0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to “1” to search for the sequence number of the block to be restarted, the CRT screen changes to the program restart screen. When the program restart signal is set to “0”, and automatic operation is activated, the tool is moved back to the machining restart point at dry run speed along the axes one by one in the sequence specified in parameter No. 7310. When the tool is set to the restart point, machining restarts.

Program restart under way signal SRNMV<F002#4>

[Classification] Output signal

[Function] Indicates the program is being restarted.

[Output condition] The program restart under way signal becomes “1” when:

– The program restart signal is set to “0” after the CRT screen changes to the program restart screen.

The signal is set to “0” when:

– The program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006								SRN
F002				SRNMV				

Parameter

7310	Movement sequence to program restart position

This parameter can be set at the "Setting Screen."

[Data type] Byte axis

[Valid data range] 1 to no. of controlled axes

This parameter sets the axis sequence when the machine moves to the restart point by dry run after a program is restarted.

[Example]
The machine moves to the restart point in the order of the fourth, first, second, and third axes one at a time when the first axis = 2, the second axis = 3, the third axis = 4, and the fourth axis = 1 are set.

Alarm and message

Number	Message	Description
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.)
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be specified when the program is restarted. (After power ON, after emergency stop or P / S alarm 94 to 97 were reset, no automatic operation was performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.

Warning

WARNING

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.4.4	PROGRAM RESTART
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.4.4	PROGRAM RESTART
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.4.4	PROGRAM RESTART
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.4.4	PROGRAM RESTART

5.8 EXACT STOP/ EXACT STOP MODE/ TAPPING MODE/ CUTTING MODE (M SERIES)

General

NC commands can be used to control a feedrate in continuous cutting feed blocks as described below.

- **Exact stop (G09)**

The tool is decelerated in a block specifying G09, and an in–position check (*1) is performed. When the feed motor falls in–position, the tool is moved by the next block. This function may be used to produce a sharp edge at the corner of a workpiece.

- **Exact Stop Mode (G61)**

When G61 is commanded, deceleration of cutting feed command at the end point and inposition check is performed per block thereafter. This G61 is valid until G62 (automatic corner override), G63 (tapping mode), or G64 (cutting mode), is commanded.

- **Tapping Mode (G63)**

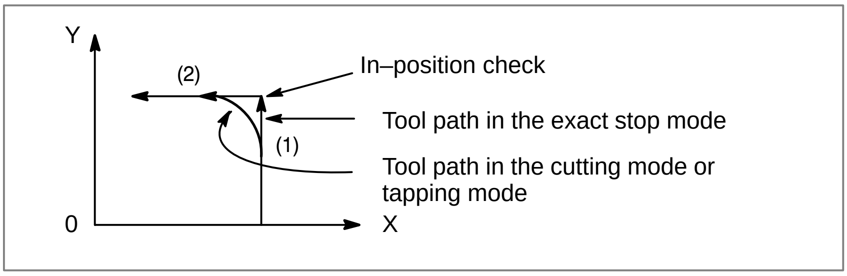
When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. This G63 is valid until G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

- **Cutting Mode (G64)**

When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode), G62 (automatic corner override), or G63 (tapping mode) is commanded.
However, in G64 mode, feed rate is decelerated to zero and in–position check is performed in the following case;
1) Positioning mode (G00, G60)
2) Block with exact stop check (G09)
3) Next block is a block without movement command

*1 The term in–position indicates that the servo motor reaches in a range of positions specified by a parameter. See Section 7.2.5.1 and 7.2.5.2 for details.

(Example) Tool paths from block (1) to block (2)



Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)

5.9 DNC OPERATION

General

By starting automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read from the reader/puncher interface or memory card.
It is possible to select files (programs) saved in an external input/output unit of a floppy format (Handy File, Floppy Cassettes, or FA card) and specify (schedule) the sequence and frequency of execution for automatic operation.
To use the DNC operation function, it is necessary to set the parameters related to the reader/puncher interface in advance.

Signal

DNC operation select signal DNCI<G043#5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), it is necessary to select the memory operation mode (MEM) and set the DNC operation select signal to “1”.

[Operation] When the DNC operation select signal becomes “1”, the control unit operates as follows:

– If the memory mode (MEM) has not been selected, the signal is ignored, and nothing happens.
– If the memory operation mode (MEM) has been selected, the DNC operation mode (RMT) is selected, and DNC operation becomes possible. In this case, the DNC operation selection confirm signal MRMT becomes “1”.

DNC operation selection confirm signal MRMT<F003#4>

[Classification] Output signal

[Function] Indicates that the DNC operation mode (RMT) has been selected.

[Output condition] The DNC operation selection confirm signal becomes “1” when:
– The DNC operation mode (RMT) is selected.

The DNC operation selection confirm signal becomes “0” when:
– The DNC operation mode (RMT) is not selected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043			DNCI					

	#7	#6	#5	#4	#3	#2	#1	#0
F003				MRMT				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100			ND3					

This parameter can be set at the "Setting Screen."

[Data type] Bit

ND3 In DNC operation, a program is:

0 : Read block by block. (A "DC3" code is output for each block.)
1 : Read continuously until the buffer becomes full. (A "DC3" code is output when the buffer becomes full.)

NOTE
In general, reading is performed more efficiently when ND3=1. This specification reduces the number of buffering interruptions caused by reading of a series of blocks specifying short movements. This reduces the cycle time.

	#7	#6	#5	#4	#3	#2	#1	#0
0138	DNM							

[Data type] Bit

DNM DNC operation with a memory card is:

0 : Disabled.
1 : Enabled. (A PCMCIA card attachment is required separately.)

NOTE
1 It is necessary to secure the memory card by using a PCMCIA card attachment suitable for the memory card CNC.
2 While DNC operation using a memory card is being performed, display of a directory in the memory card and other operations that access to the memory card cannot be performed.
3 A set DNC operation file name is erased at power–off.
4 During DNC operation using a memory card, do not remove or insert the memory card.
5 A program executed in DNC operation cannot call a program stored on the memory card.

Alarm and message

Number	Message	Description
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M99	M198 and M99 are executed in the schedule operation. Or M198 is executed in the DNC operation.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.4.5	SCHEDULING FUNCTION
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.4.5	SCHEDULING FUNCTION
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.4.5	SCHEDULING FUNCTION
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.4.5	SCHEDULING FUNCTION
CONNECTION MANUAL (This manual)		13.1	READER/PUNCHER INTERFACE
		15.6	DIRECT OPERATION BY PMC OPEN CNC

5.10 MANUAL INTERVENTION AND RETURN

General

If the tool movement along the axes is stopped by a feed hold during automatic operation, then restarted after manual intervention such as tool exchange, the tool moves back to the point of intervention before automatic operation is resumed.
This function is easy to operate because unlike the program restart function or the tool retract and restore function, it is unnecessary to operate switches on the operator's panel or MDI keys.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001								MIN

[Data type] Bit

MIN The manual intervention and return function is:
0 : Disabled.
1 : Enabled.

Warning

WARNING
If you do not make manual intervention correctly according to the direction of machining and the shape of the workpiece, the machine and tool may be broken down. Use sufficient care.

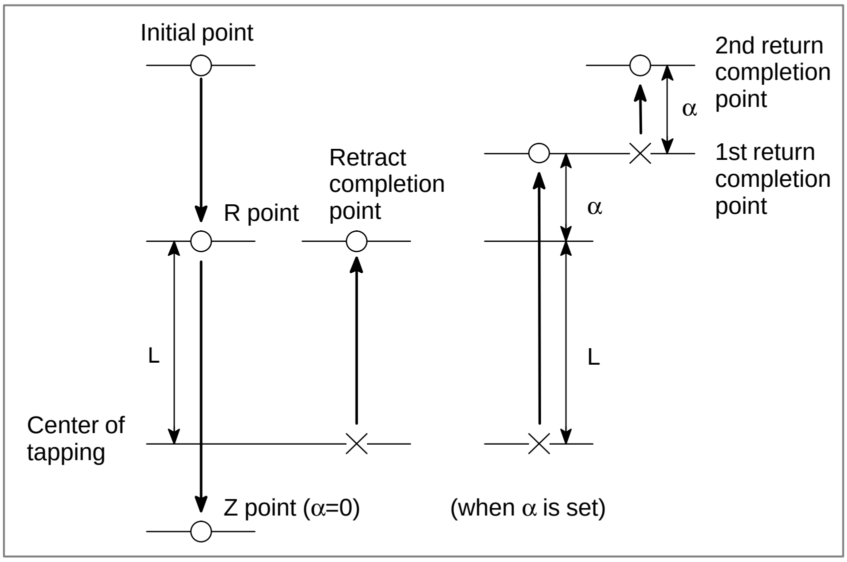
Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.4.9	MANUAL INTERVENTION AND RETURN
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.4.9	MANUAL INTERVENTION AND RETURN
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.4.9	MANUAL INTERVENTION AND RETURN
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.4.8	MANUAL INTERVENTION AND RETURN

5.11 RETRACTION FOR RIGID TAPPING (M SERIES)

General

When rigid tapping is stopped, either as a result of an emergency stop or a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap retraction signal is input, the tap is removed from the hole, based on the stored information. The tap is pulled toward the R point. When a retract value α is set in parameter No. 5382, the retraction distance can be increased by α.



Basic procedure

(1) Start

Reset the CNC, then select MDI mode. Setting rigid tapping retraction start signal RTNT to “1” starts rigid tapping retraction.

(2) Completion

Upon the completion of rigid tapping retraction, rigid tapping retraction completion signal RTPT is set to “1”, with which the CNC automatically enters the reset state. Setting rigid tapping retract start signal RTNT to “0” sets rigid tapping retraction completion signal RTPT to “0”.

(3) Stop

During rigid tapping retraction, setting rigid tapping retraction start signal RTNT to “0” stops rigid tapping retraction, placing the CNC in the reset state. To resume rigid tapping retraction, set rigid tapping retraction start signal RTNT to “1”. Rigid tapping retraction can also be stopped by means of a reset or feed hold.

(4) Resume

Once rigid tapping retraction has been stopped, it can be resumed by performing the same operation as that used for starting rigid tapping retraction. If rigid tapping retraction has been completed, however, the start operation does not restart rigid tapping retraction. If retract value α is set in parameter No. 5382, however, the start operation performs rigid tapping retraction using α only.

Start and completion time chart

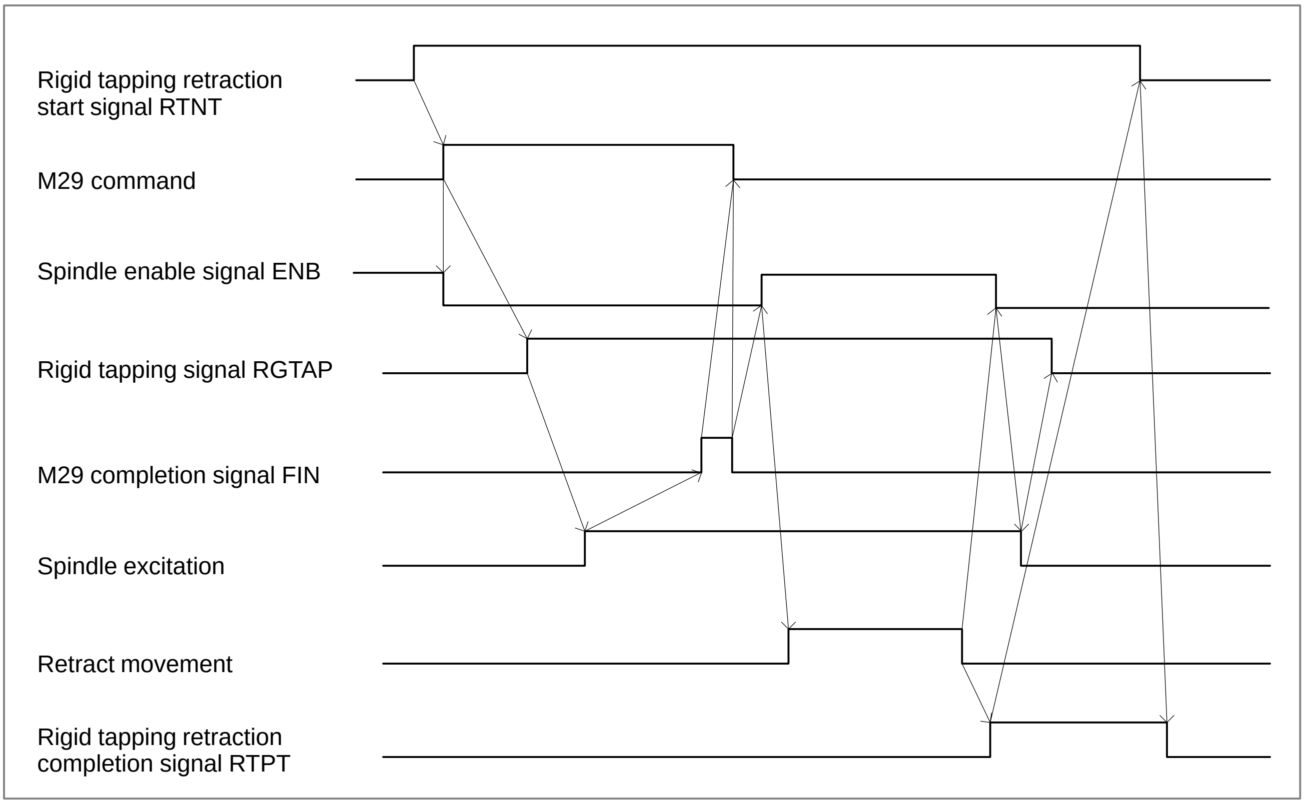
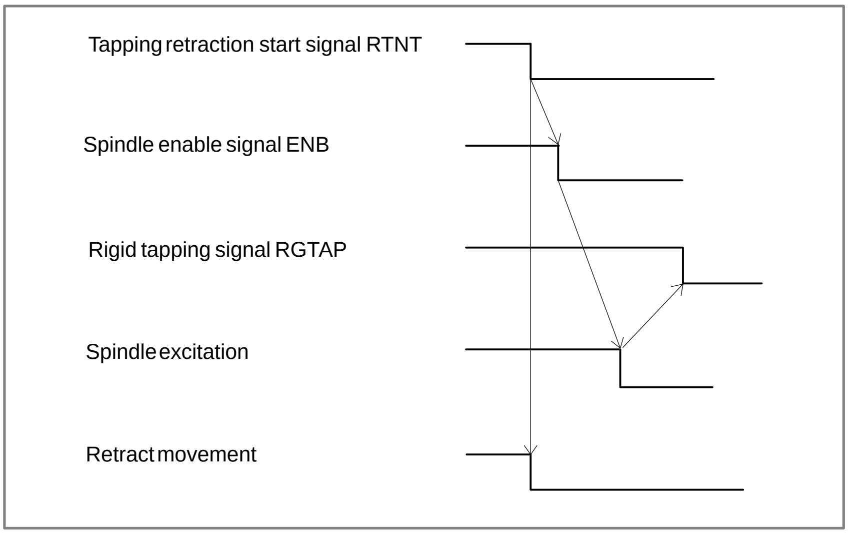


Fig. 5.11 Start and completion time chart

In the reset state, setting rigid tapping retraction start signal RTNT to “1” in MDI mode causes the rigid tapping M command to be output. For rigid tapping retraction, specify neither gear switching nor orientation. Spindle function strobe signal SF is also output if no S command has been specified after power–on.

Upon the completion of rigid tapping retraction, spindle enable signal ENB is set to “0”, in the same way as at the end of ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. Once rigid tapping retraction has been completed, rigid tapping retraction completion signal RTPT is set to “1” and the CNC enters the reset state.

Time chart for stopping tapping retraction



When tapping retraction is stopped, spindle enable signal is set to 0, in the same way as for ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. The CNC also automatically enters the reset state when tapping retraction is stopped.

Rigid tapping retraction by G30 command

Parameter setting allows a programmed G30 command to be used also to draw out the tool. When the power is disconnected during rigid tapping because of an accident such as a blackout, this function can draw out the tapping tool from the workpiece if an absolute position detector is provided for the servo axis.

• Specification method

When rigid tapping has been stopped as a result of an emergency stop or a reset, or when the power has been disconnected during rigid tapping, execute the command in the format shown below in MEM operation or MDI operation. Then, based on the rigid tapping command information in the machining program, the tool moves along the tapping axis to the initial point or R point in synchronization with the spindle.
The rigid tapping retraction command is a one–shot command.

G30 P99 M29 S <u>min^{-1}</u> ;

• Time chart

The time chart for activation is the same as that of ordinary rigid tapping.

Since the G30 rigid tapping retraction command is a one–shot command, the G80 command for canceling rigid tapping is not provided.
Therefore, the processing of canceling the rigid tapping signal RGTAP <G061#0> in response to the tapping retraction completion signal RTPT <F066#1> must be added to the time chart for terminating ordinary rigid tapping.

• Limitations

When the G30 command is used to perform rigid tapping (bit 1 (RG3) of parameter No. 5201 is set to 1), there are limitations as follows:

1. Rigid tapping retraction using input signal RTNT <G62#6> is not allowed.
2. Be sure to set bit 0 (G84) of parameter No. 5200 to 0.

3. The override value (parameter No. 5381) for rigid tapping retraction is ignored.

Example:

- Machining program

```
M29 S1000 ;
G84 X20. Y20. R–10. Z–30. F500 ;
X50. Y50. ;
X100. Y100. ;
G80
```

- Retraction program

```
G30 P99 M29 S1000 ;
G00 Z–10. ;
```

(Supplementary) If a value other than 0 is set in parameter No. 5210 or 5212, the M code with the parameter–set value is specified instead of M29 in the above program.

Signal

Rigid tapping retraction start signal RTNT <G062#6>

[Classification] Input signal

[Function] Starts rigid tapping retraction.

[Operation] When this signal is set to “1”, the control unit operates as follows:

- Starts rigid tapping retraction.

Rigid tapping retraction completion signal RTPT <F066#1>

[Classification] Output signal

[Function] Notifies the completion of rigid tapping retraction.

[Output condition] This signal is set to “1” in the following case:

- Rigid tapping retraction has been completed.

This signal is set to “0” in the following case:

- Rigid tapping retraction start signal has been set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G062		RTNT						

	#7	#6	#5	#4	#3	#2	#1	#0
F066							RTPT	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5200				DOV				

[Data type] Bit

DOV For tool extraction during rigid tapping, override is:

0 : Disabled.
1 : Enabled. (The override value is set in parameter No.5211 or No.5381.)

	#7	#6	#5	#4	#3	#2	#1	#0
5202							RG3	

[Data type] Bit

RG3 Rigid tapping retraction is performed using:

0 : Input signal RTNT <G62#6>
1 : One–shot G code G30 command

NOTE
1 When this parameter has been set, the power must be turned off before operation is continued.
2 When this parameter is to be set to 1, be sure to set bit 0 (G84) of parameter No. 5200 to 0.

5381	Override for rigid tapping retraction

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 200

Sets an override value to be applied to rigid tapping retraction. No override is applied if 0 is set.

NOTE
1 This parameter is valid when bit 4 (DOV) of parameter No. 5200 is set to 1.
2 If bit 3 (OVU) of parameter No.5201 is set to 1, 10% is set as the units of data. Thus, an override of up to 2000% can be applied during extraction.

5382	Retract value α for rigid tapping return

[Data type] 2–word

[Unit of data] Input increments

[Valid data range] 0 to 99999999

Sets an extra retract value for rigid tapping retraction. The tool will be pulled beyond the R point by α. If rigid tapping retraction has already been completed, the tool is pulled by α only.

Caution

CAUTION

1 If rigid tapping is stopped as a result of an emergency stop, the position on the tapping axis (Z–axis) is maintained but the spindle position is lost. In such a case, therefore, the positional relationship between the spindle and tapping axis is not guaranteed when operation is resumed.
2 Rigid tapping retraction is performed based on the tapping axis (Z–axis) commands accumulated for tapping. If rigid tapping is stopped as a result of an emergency stop, therefore, rigid tapping retraction may fail to draw the tapping tool completely out of the workpiece. In such a case, set retract value α (parameter No. 5382).
3 During rigid tapping retraction, switching the mode to manual mode stops rigid tapping retraction.
4 For rigid tapping retraction, the CNC internally activates a return program. Rigid tapping retraction may, therefore, cause some G codes or M/F/S codes to be overwritten (G80/G84/G74, G94/G95, G30).

Note

NOTE

1 Setting rigid tapping retraction start signal RTNT to "1" starts rigid tapping retraction only when the CNC is placed in both the reset state and MDI mode.
2 The machining data for rigid tapping retraction is maintained until a rigid tapping command is subsequently specified, even while the power is turned off. Rigid tapping retraction can, therefore, be specified even if the power has been turned off after rigid tapping.
3 Rigid tapping retraction is not performed if the input increments (inches or mm) selected when tapping return is specified differ from those selected when the machining data for tapping retraction was stored.
4 An override can be applied to rigid tapping retraction, if it is enabled with the corresponding parameter.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.13.2	Rigid tapping
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.13.2	Rigid tapping
CONNECTION MANUAL (This manual)		9.10	Rigid tapping

6 INTERPOLATION FUNCTION

6.1 POSITIONING

General

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

The tool path is determined by selecting one of the following with parameter LRP (No. 1401#1):

- Linear interpolation type positioning
  The tool is positioned using a straight path and a speed that is not higher than the rapid traverse of each axis but that assures the shortest positioning time. By changing the acceleration/deceleration type from the constant acceleration/deceleration (inclination) type to the constant time (time constant) type with bit 4 (PRT) of parameter No. 1603, the tool can be moved along a specified path.
- Non–linear interpolation type positioning
  Positioning is performed with each axis independently at the rapid traverse rate. Generally, the tool path is not a straight line.

The rapid traverse rate in the G00 command is set to the parameter No.1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Data type] Bit

LRP Positioning (G00)

0 : Positioning is performed with non–linear type positioning so that the tool moves along each axis independently at rapid traverse.
1 : Positioning is performed with linear interpolation so that the tool moves in a straight line.

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotaion axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				PRT				

[Data type] Bit

PRT The acceleration/deceleration of interpolation–type rapid traverse is performed:
0: With a constant inclination.
1: With a constant time.

> **NOTE**
> This parameter is invalid if the function of bell–shaped acceleration/deceleration after rapid–traverse interpolation is provided. The acceleration/deceleration time constant and override for rapid traverse are used.

Note

> **NOTE**
> The rapid traverse rate cannot be specified in the address F.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.1	POSITIONING (G00)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.1	POSITIONING (G00)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.1	POSITIONING (G00)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.1	POSITIONING (G00)

6.2 LINEAR INTERPOLATION

General

Tools can move along a line

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path.
If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01$\alpha\underline{\alpha}\beta\underline{\beta}\gamma\underline{\gamma}\zeta\underline{\zeta}$ $F\underline{f}$;

Feed rate of α axis direction : $F\alpha = \dfrac{\alpha}{L} \times f$

Feed rate of β axis direction : $F_\beta = \dfrac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \dfrac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F_\zeta = \dfrac{\zeta}{L} \times f$

$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$$

The feedrate of the rotary axis is commanded in the unit of deg/min (if the feedrate is 12 deg/min, F12.0 is commanded).

When the straight line axis α(such as X, Y, or Z) and the rotating axisβ (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the αandβ cartesian coordinate system is commanded by F(mm/min).
β–axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above fromula, then the β –axis feedrate unit is changed to deg/min.

A calculation example is as follows.

(Example)

G91 G01 X20.0C40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2 + 40^2}}{300} \doteq 0.14907 \text{ (min)}$$

The feed rate for the C axis is

$$\frac{40 \ deg}{0.14907 \ min} \doteq 268.3 \text{ deg/min}$$

In simultaneous 3 axes control, the feed rate is calculated the same way as in 2 axes control.

Parameter

1411	
	Cutting feedrate when the power is turned on

This parameter can be set in "Setting screen".

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 32767	6 – 32767
Inch machine	0.01 inch/min	6 – 32767	6 – 32767

When the machine requires little change in cutting feedrate during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate in the NC command data.

The feedrate set in this parameter is effective between the CNC being cleared, upon a power–on or a reset, and a feedrate being specified with a program command (F command). Once a feedrate has been specified with an F command, that feedrate becomes effective.

1422	Maximium cutting feedrate for all axes

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.
A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

> **NOTE**
> To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead. (M series)

1430	
	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotaion axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.
A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

NOTE
1 This parameter is effective only in linear and circular interpolation. In polar coordinate and cylindrical, the maximum feedrate for all axes specified in parameter No. 1422 is effective.
2 If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								G01

[Data type] Bit

G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

Alarm and message

No.	Message	Description
011	NO FEEDRATE COMMAN- DED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.

Reference item

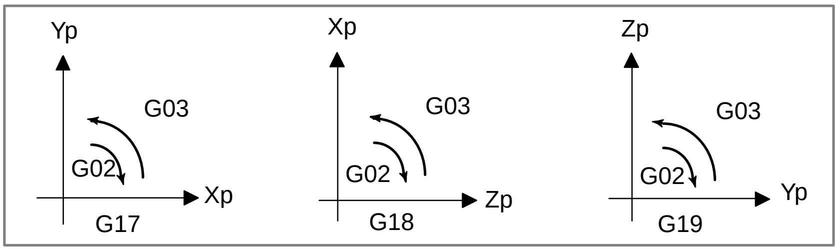
Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.3	LINEAR INTERPOLATION (G01)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.2	LINEAR INTERPOLATION (G01)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.3	LINEAR INTERPOLATION (G01)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.2	LINEAR INTERPOLATION (G01)

6.3 CIRCULAR INTERPOLATION

General

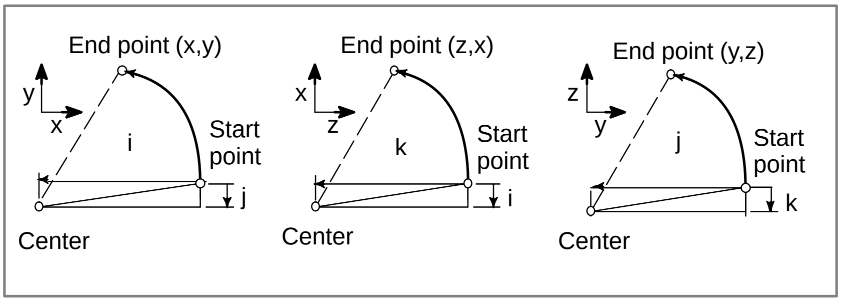
The command below can move a tool along a circular arc in the defined plane.

“Clockwise”(G02) and “counterclockwise”(G03) on the X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive–to–negative direction of the Z_p axis (Y_p axis or X_p axis, respectively) in the Cartesian coordinate system. See the figure below.



The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified with a sign.

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value, as shown below.
I, J, and K must be signed according to the direction.
G02; Command for a circle



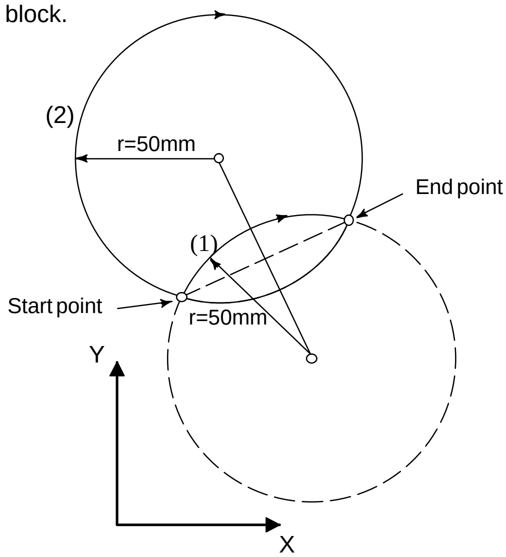
I0,J0, and K0 can be omitted. When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02 Ii; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.020) occurs.

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K. In this case, one arc is less than 180°, and the other is more than 180° are considered.

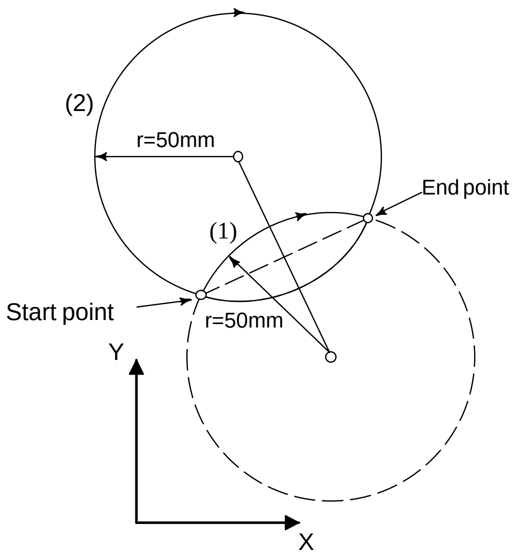
For T series, an arc with a sector angle of 180° or wider cannot be specified (P/S alarm No. 023).
For M series, specify an arc more than 180° with a negative radius value commanded.

If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0°is programmed.
G02R_ ; (The tool does not move.)

(Example) (T series)
For arc (1) (less than 180°)
G02 W60.0 U10.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
An arc with a sector angle of 180° or wider cannot be specified within a single block.



(Example) (M series)
For arc (1) (less than 180°)
G91 G02 X60.0 Y20.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
G91 G02 X60.0 Y20.0 R–50.0 F300.0 ;



NOTE

1 Specifying an arc center with addresses I, K, and J
When the distance from the arc start point to the arc center is specified with addresses I, K, and J, a P/S alarm (No. 5059) is issued if:

Maximum value which can be specified < $\sqrt{I^2 + K^2}$

Example:
When IS–B and metric input are selected, issuing the following command (radius specification) will result in the issue of a P/S alarm (No. 5059):

G50 X0 Z0;
G18 G02 X11.250 Z10. I–800000.000 K900000.000 F5.0;

$$\because \sqrt{I^2 + K^2} = \sqrt{(-800000.000)^2 + 900000.000^2}$$
$$= 1204159.458$$
$$> 999999.999$$

2 Tool nose radius compensation
In tool nose radius compensation mode, a P/S alarm (No. 5059) is issued if the distance from the tool nose radius center to the arc center exceeds the maximum value which can be specified.

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the cutter compensation (M series) or tool nose radius compensation (T series) is applied.

Parameter

1022	Setting of each axis in the basic coordinate system

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

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
						G19	G18	

[Data type] Bit

G18 and G19 Plane selected when power is turned on or when the control is cleared

G19	G18	G17, G18 or G19 mode
0	0	G17 mode (plane XY)
0	1	G18 mode (plane ZX)
1	0	G19 mode (plane YZ)

3410	Tolerance of arc radius

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

When a circular interpolation command (G02, G03) is executed, the tolerance for the radius between the start point and the end point is set. If the difference of radii between the start point and the end point exceeds the tolerance set here, a P/S alarm No. 20 is informed.

NOTE
When the set value is 0, the difference of radii is not checked.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.

Number	Message	Description
023	ILLEGAL RADIUS COMMAND (T series)	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03 (M series)	F0 (rapid traverse) was instructed by F1 –digit command in circular interpolation. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
5059	RADIUS IS OUT OF RANGE	For circular interpolation, the radius specified with addresses I and K exceeds the allowable range.

Note

NOTE

1 For T series, the U, V and W axes (parallel with the basic axis) can be used with G–code system B and C.
2 If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
3 If an axis not comprising the specified plane is commanded, an alarm is displayed.
For example, when G code system B or C is used, if U axis with X axis is specified as a parallel axis to X axis when plane XY is specified, an P/S alarm (No.028)is displayed.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.4	CIRCULAR INTERPOLATION (G02,G03)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.4	CIRCULAR INTERPOLATION (G02,G03)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)

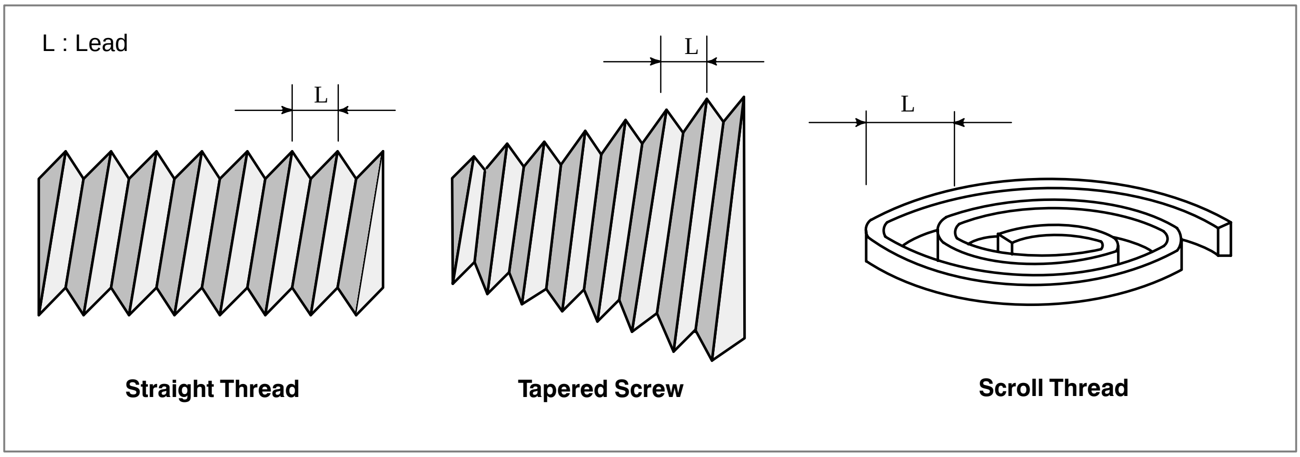
6.4 THREAD CUTTING

6.4.1 Thread Cutting

General

Tool movement can be synchronized with spindle rotation when cutting threads.

The spindle speed is continuously read through the position coder attached to the spindle. Then, it is converted to a cutting feedrate (feed per minute) to feed the tool.



In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

Signal

Thread cutting signal THRD<F002#3>

[Function] This signal indicates that thread cutting is in progress.

[Output condition] This signal turns to “1” in the following cases:
- Thread cutting mode in progress
- Thread cutting cycle for turning

This signal turns to “0” in the following case.
- Neither thread cutting mode nor thread cutting cycle are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002					THRD			

Parameter

Enabling/disabling dry run operation during threading

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)

0 : Enabled
1 : Disabled

Checking the spindle speed arrival signal before starting threading

	#7	#6	#5	#4	#3	#2	#1	#0
3708							SAT	SAR
								SAR

[Data type] Bit

SAR: The spindle speed arrival signal is:

0 : Not checked
1 : Checked

SAT: Check of the spindle speed arrival signal at the start of executing the thread cutting block

0 : The signal is checked only when SAR, #0 of parameter 3708, is set.
1 : The signal is always checked irrespective of whether SAR is set.

> **CAUTION**
> When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

Setting the time constant for the threading cycle

1626	Time constant of exponetial acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the FL feedrate for the thread cutting cycle

1627	FL rate of exponential acceleration /deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the chamfering distance for the thread cutting cycle

5130	Chamfering distance in the thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in the thread cutting cycles G76 and G92.

Setting the minimum depth of cut for the multiple repetitive canned cycle G76

5140	Minimium depth of cut in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

This parameter sets the minimum depth of cut in the multiple repetitive canned cycle G76.

Setting the finishing allowance for the multiple repetitive canned cycle G76

5141	Finishing allowance in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the finishing allowance in the multiple repetitive canned cycle G76.

Setting the repetition count of finishing for the multiple repetitive canned cycle G76

5142	Repetition count of final finishing in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in the multiple repetitive canned cycle G76.

Setting the tool angle for the multiple repetitive canned cycle G76

5143	Tool nose angle in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 (When FS10/11 TAPE FORMAT is used)
0, 29, 30, 55, 60, 80 (When FS10/11 TAPE FORMAT is not used)

This parameter sets the tool nose angle in the multiple repetitive canned cycle G76.

Warning

WARNING

During threading, stopping feed without stopping the spindle is dangerous because the cutting depth will abruptly increase. Feed hold is, therefore, disabled during threading. If attempted during threading, feed stops in the same way as single block stop upon the completion of the first non–threading block after the termination of threading mode. The feed hold lamp (SPL lamp), however, lights immediately after the feed hold button (on the machine operator's panel) is pressed. The lamp goes off when feed stops (the CNC enters the single block stop state).

Caution

CAUTION

1 Feedrate override is ignored during thread cutting, 100% being assumed.
2 During threading, spindle override is ignored, 100% being assumed.
3 When the first non–threading block is executed after threading mode has been finished, and the feed hold button is pressed again (or the feed hold button has been held down), the execution of the non–threading block is stopped immediately.
4 When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.
5 When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1–turn signal even if the present block is a thread cutting block.
6 When a dry run operation is performed the dry run rate becomes the longitudinal axis feedrate.
7 For T series, the thread cutting retract function is supported only for the threading cycle.

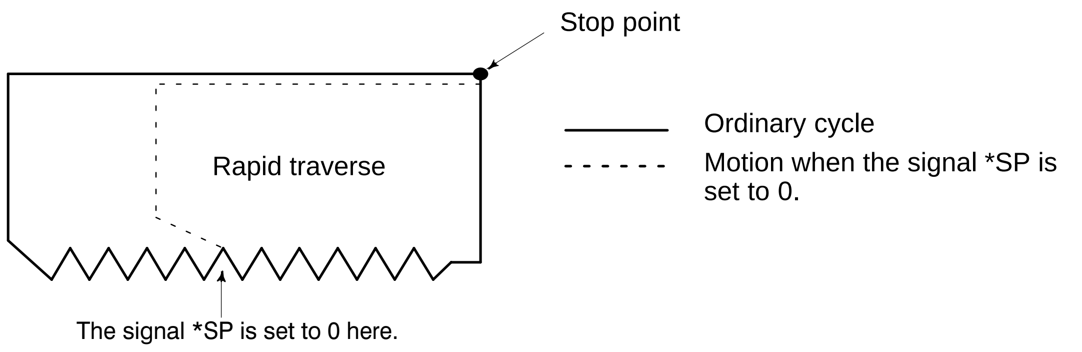
Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.7	THREAD CUTTING
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.7	CONSTANT LEAD THREAD CUTTING
		II.4.8	VARIABLE LEAD THREAD CUTTING
		II.4.9	CONTINUOUS THREAD CUTTING
		II.4.10	MULTIPLE THREAD CUTTING
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.6	THREAD CUTTING
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.6	CONSTANT LEAD THREAD CUTTING
		II.4.7	CONTINUOUS THREAD CUTTING
		II.4.8	MULTIPLE THREAD CUTTING

6.4.2 Thread Cutting Cycle Retract (T series)

General

When the automatic operation stop signal *SP <G008#5> is set to 0 during threading in a threading cycle, the tool immediately retracts while performing chamfering, then returns to the start point of the current cycle, first along the X–axis, then along the Z–axis.



Parameter

- **Setting to enable the override function during thread cutting cycle retraction**

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV							

[Data type] Bit

RTV Override while the tool is retracting in threading

0 : Override is effective.
1 : Override is not effective.

- **Setting a chamfering distance in thread cutting cycle retraction**

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in thread cutting cycles G76 and G92.

Caution

CAUTION
While the tool is retracting, automatic operation stop signal *SP <G008#5> is ignored.

Note

NOTE
The chamfering distance for retraction is determined by the setting of parameter No. 5130.

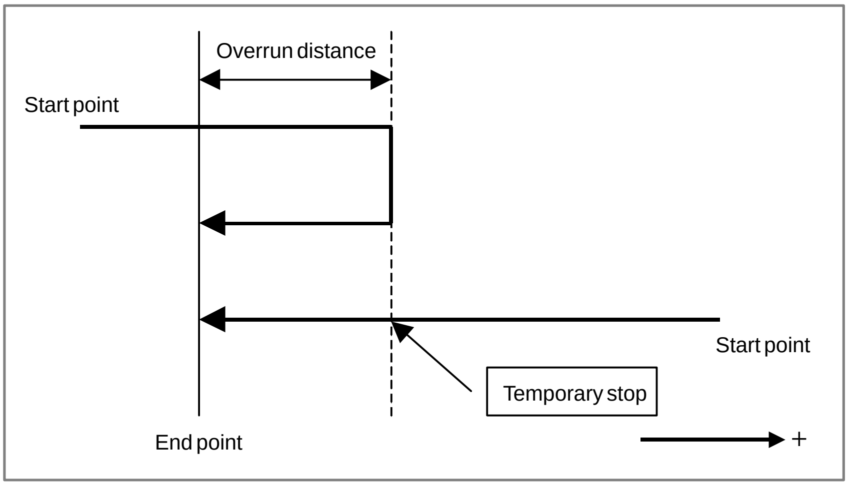
Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.13.1.2	Thread Cutting Cycle
		II.13.2.7	Multiple Thread Cutting Cycle
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.13.1.2	Thread Cutting Cycle
		II.13.2.7	Multiple Thread Cutting Cycle

6.5 SINGLE DIRECTION POSITIONING

General

For accurate positioning without play of the machine (lost motion), positioning is performed in one direction finally.



Example where positioning is performed in the minus direction

Format

G60 IP_;

IP_: Coordinates of an end point for tool movement when an absolute command is specified

Explanation

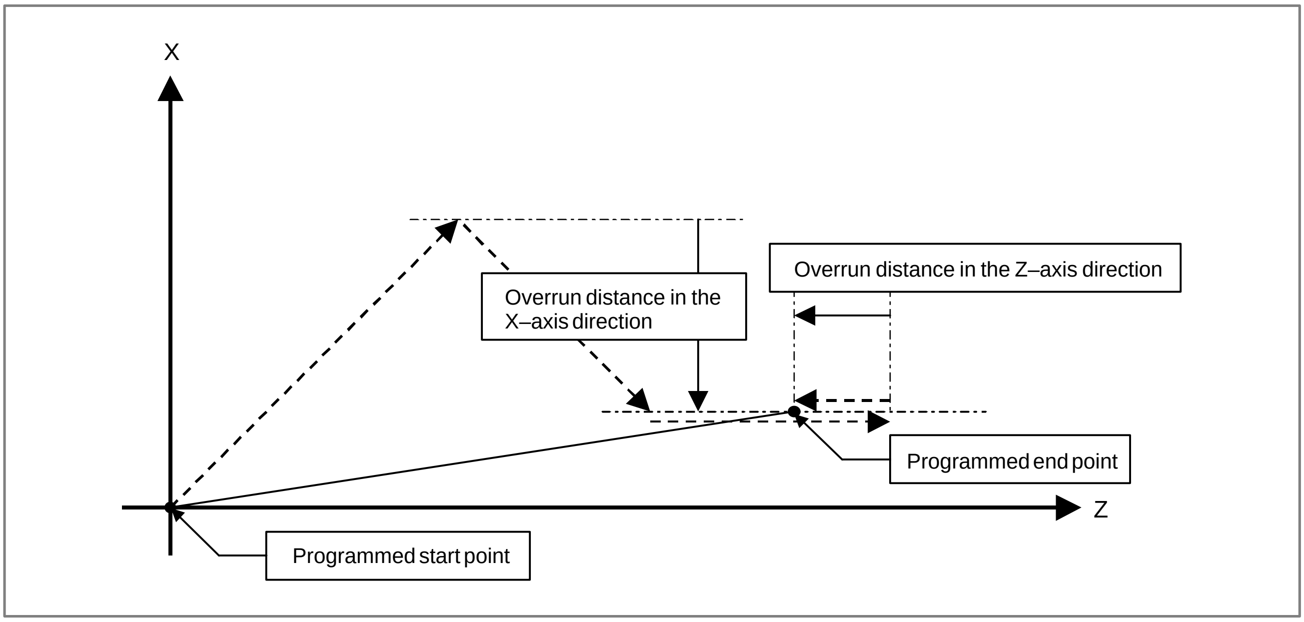
Parameter No. 5440 is used to set an overrun distance and positioning direction. The tool stops once before a specified end point also when a specified positioning direction matches the positioning direction set in the parameter.

By setting bit 0 (MDL) of parameter No. 5431 to 1, the one–shot G code G60 can be used as a modal G code of group 01. This eliminates the need to specify G60 for each block, and enables one–shot G code to be specified in the single direction positioning mode. The other specifications are the same as for the one–shot G code G60.

(Example)

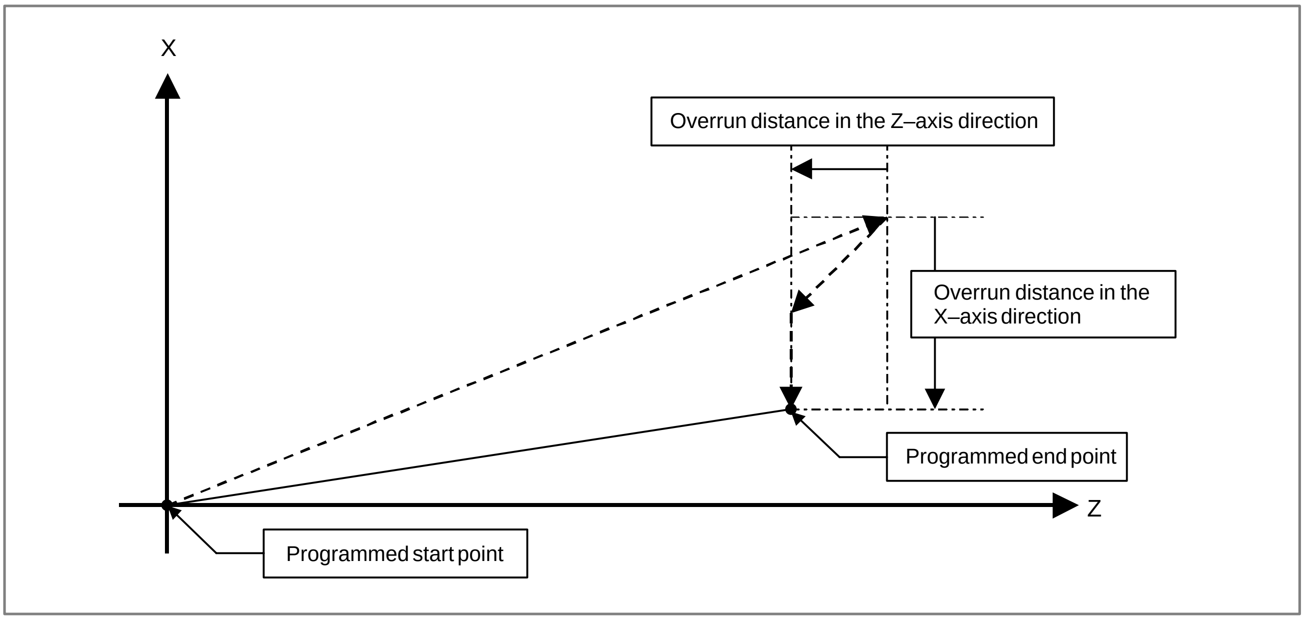
When the one–shot G code G60 is used		When the modal G code G60 is used	
:		:	
G90 ;		G90 G60 ;	Starts the single direction positioning mode.
G60 X0 Y0 ;	Single direction positioning	X0 Y0 ;	Single direction positioning
G60 X100 ;		X100 ;	
G60 Y100 ;		Y100 ;	
G04 X10 ;		G04 X10 ;	
G00 X0 Y0 ;		G00 X0 Y0 ;	Cancels the single direction positioning mode.
:		:	

- **Overview of operation**



In the case of positioning of non–linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 0)

As shown above, single direction positioning is performed independently along each axis.



In the case of positioning of linear interpolation type (bit 1 (LRP) of parameter No. 1401 = 1)

Positioning of interpolation type is performed until the tool once stops before or after a specified end point. Then, the tool is positioned independently along each axis until the end point is reached.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5431							PDI	MDL

[Data type] Bit

MDL Specifies whether the G code for single direction positioning (G60) is included in one–shot G codes (00 group) or modal G codes (01 group)

0: One–shot G codes (00 group)
1: Modal G codes (01 group)
(The setting is valid until a G code in group 01 other than G60 is specified.)

PDI When the tool is stopped before or after a specified end point with the single direction positioning function:

0 : No in–position check is performed. (The tool waits only for the end of acceleration/deceleration.)
1 : An in–position check is performed.

5440	Positioning direction and overrun distance in single direction positioning for each axis

[Data type] Word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –16383 to +16383

This parameter sets the positioning direction and overrun distance in single directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.

Approach > 0: The positioning direction is positive (+).
Approach < 0: The positioning direction is negative (−).
Approach = 0: Uni–directional positioning is not performed.

P/S alarm

Number	Message	Description
0146	IMPROPER G CODE	In the polar coordinate interpolation mode, a G code that must not be specified was specified. Correct the program.
0176	IMPROPER G CODE IN G107	In the cylindrical interpolation mode, a G code that must not be specified was specified. Correct the program.

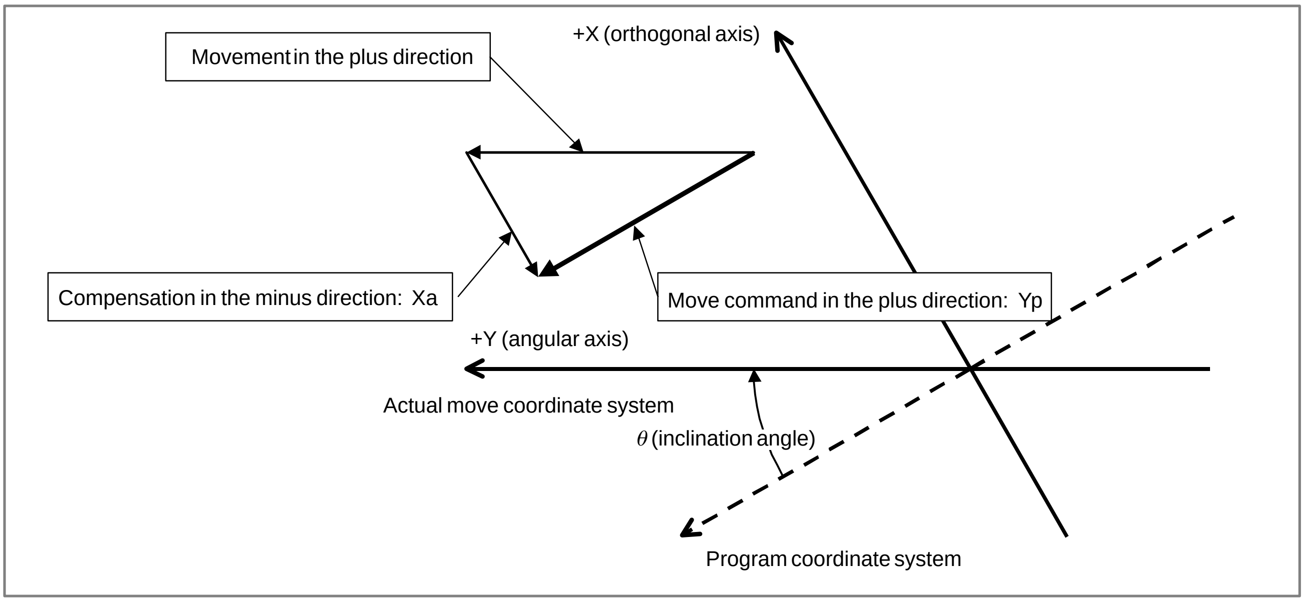
Notes

1. Single direction positioning is not performed along an axis for which no overrun distance is set in parameter No. 5440.
2. Single direction positioning is not performed along an axis for which travel distance 0 is specified.
3. The mirror image function is not applied in a parameter–set direction. Even in the mirror image mode, the direction of single direction positioning remains unchanged.
4. Single direction positioning cannot be used for a hole machining axis in a hole machining canned cycle. However, single direction positioning can be used for positioning operation.
5. In the cylindrical interpolation mode (G07.1), single direction positioning cannot be used.
6. In the polar coordinate interpolation mode (G12.1), single direction positioning cannot be used.
7. Single direction positioning is not performed for an axis along which a movement is made by a shift amount in a G76 or G87 canned cycle. (M series)
8. The G code for single direction positioning is G60, regardless of whether the G code system is A, B, or C. (T series)
9. In a multiple repetitive turning canned cycle (G70 to G76), single direction positioning cannot be used. (T series)
10. In a grind canned cycle (G71 to G74), single direction positioning cannot be used. (T series)
11. In a canned cycle (G90, G92, G94), single direction positioning cannot be used. (T series)

- **Notes on using single direction positioning and angular axis control simultaneously**

If a move command is specified on an angular axis in angular axis control, a compensation command is output for the orthogonal axis. Let Yp be a move command for an angular axis, and let (be the inclination angle of the angular axis. Then, the compensation command Xa for the orthogonal axis is:

Xa = –Yp•tan θ

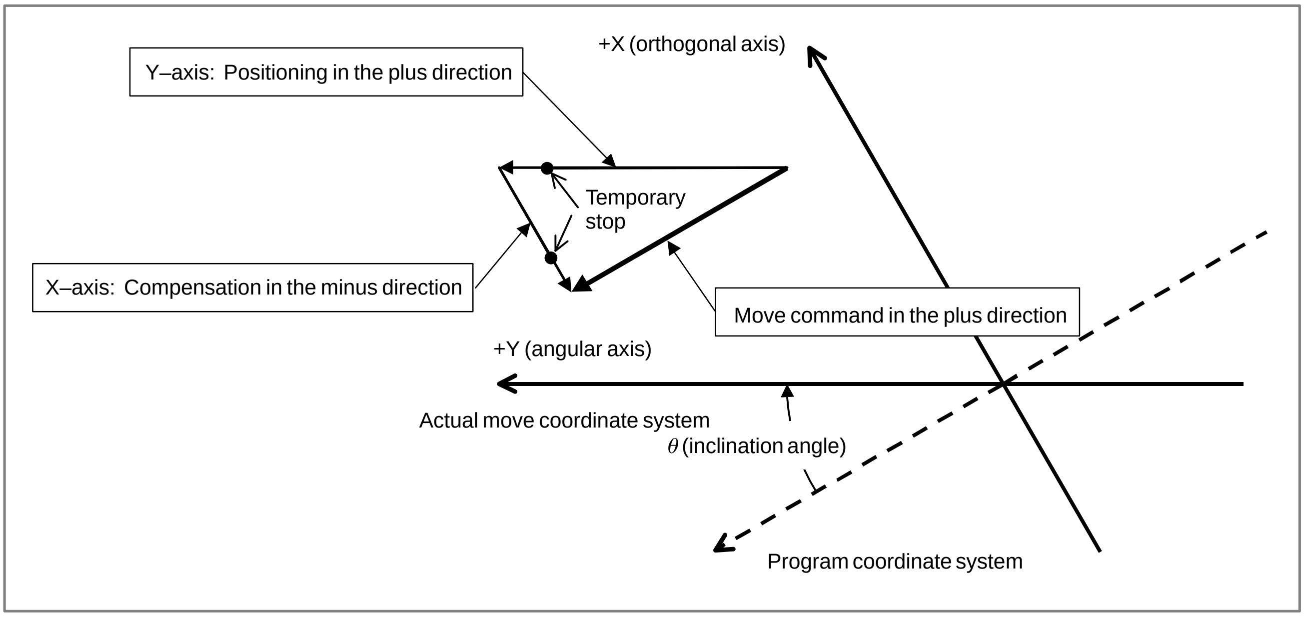


The direction of the compensation command Xa is determined by the inclination angle θ of the angular axis and the direction of the move command Yp for the orthogonal axis. When tan θ is plus, the direction of the move command for the angular axis is opposite to that of the compensation command for the orthogonal axis. (For example, if a move command in the plus direction is specified for an angular axis when the inclination angle is +30°, a compensation command in the minus direction is output for the orthogonal axis.)

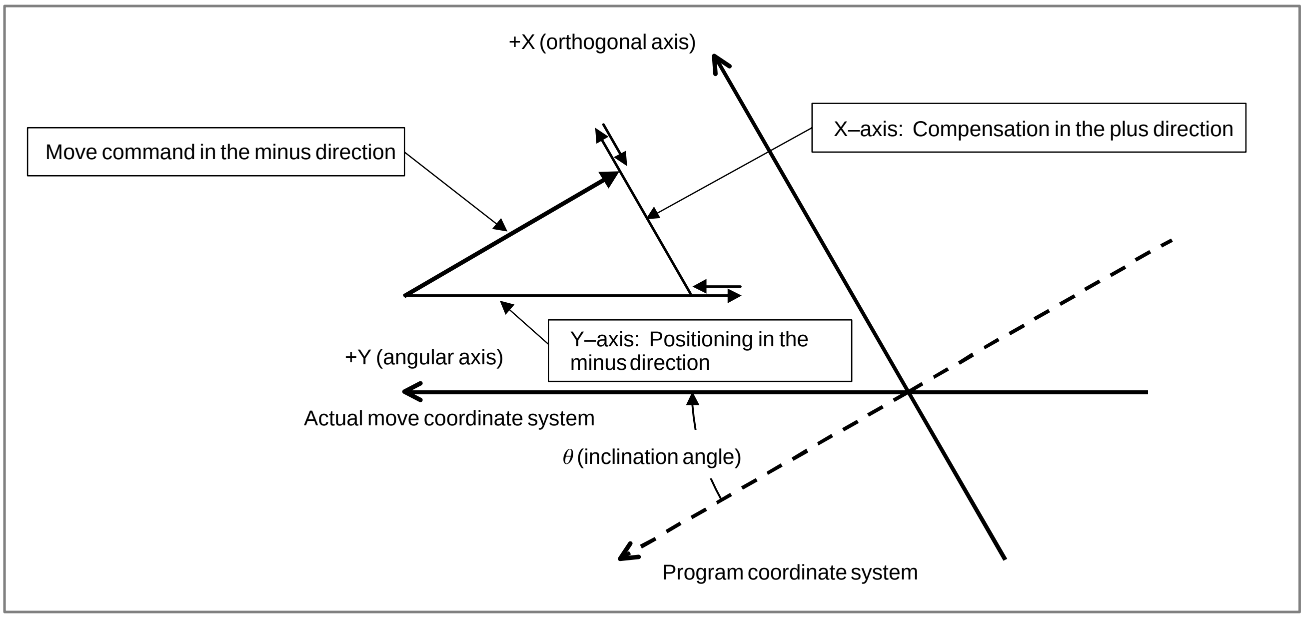
So, when single direction positioning is performed during angular axis control, positioning may be performed by a compensation command along the orthogonal axis in the direction opposite to the direction set in parameter No. 5440. To prevent this situation from occurring, set the parameter as described below so that the direction of the compensation command matches the positioning direction for the orthogonal axis.

- **When the inclination angle of an angular axis (parameter No. 8210) is 1° to 89° or 181° to 269°**

Set the directions of single direction positioning (parameter No. 5440) for the angular axis and orthogonal axis so that the directions for the angular axis and for the orthogonal axis are opposite (plus/minus) to each other. When the positioning direction for the X–axis (orthogonal axis) is minus, and the positioning direction for the Y–axis (angular axis) is plus, the operation shown below is performed.



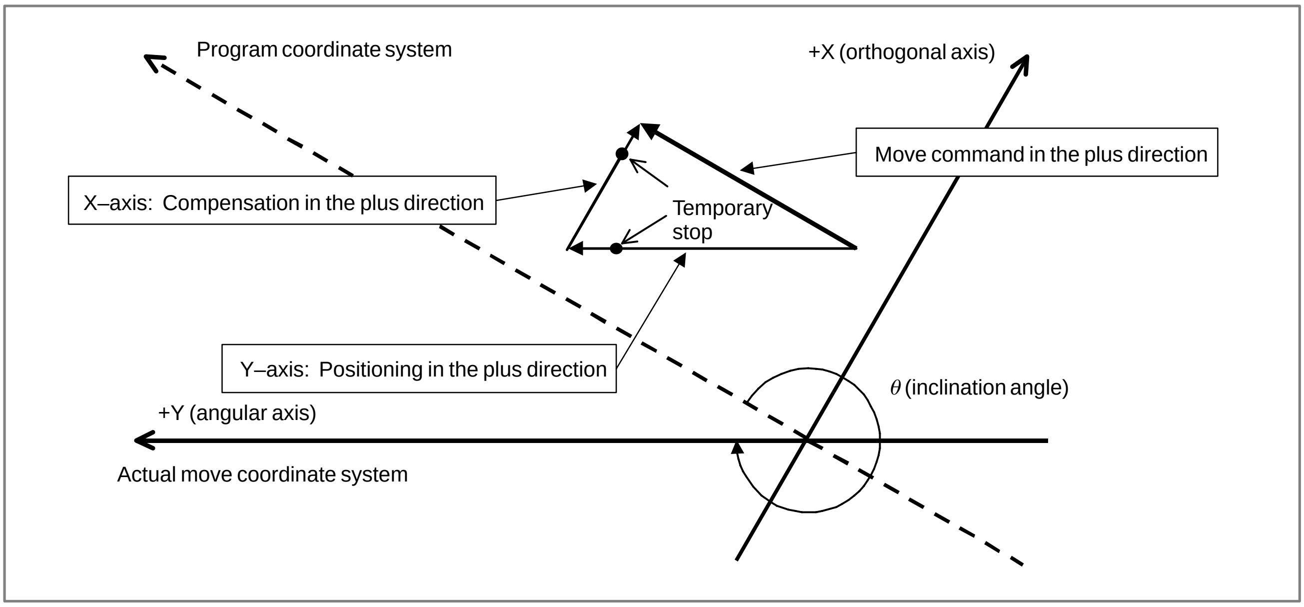
Move command in the plus direction



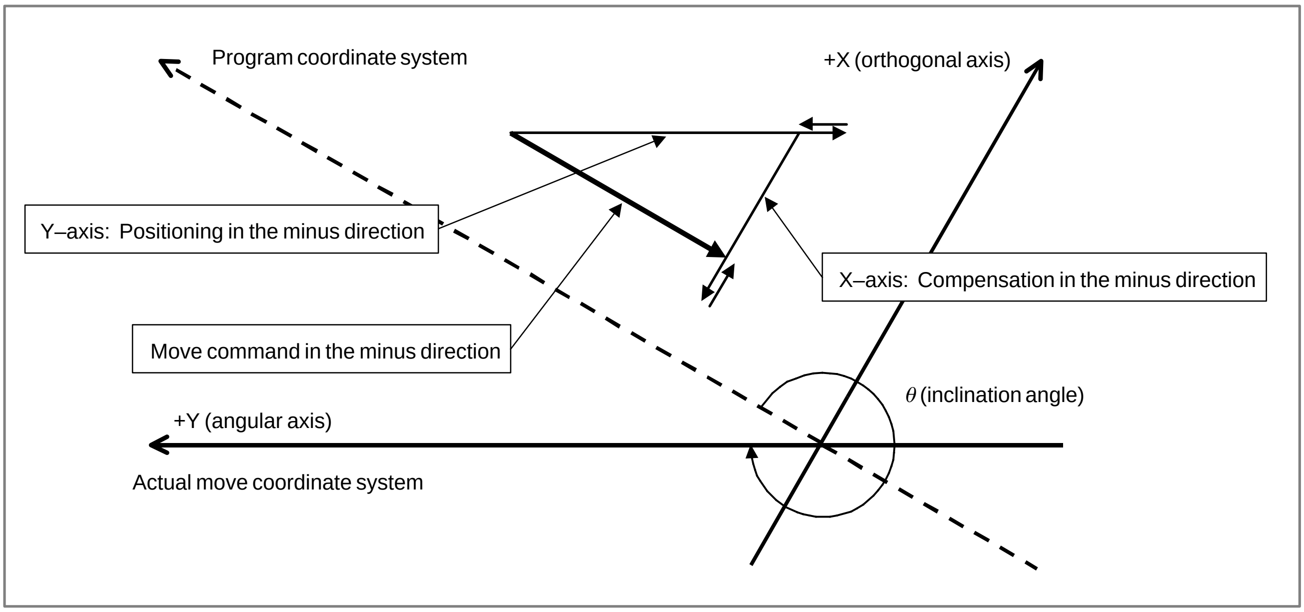
Move command in the minus direction

- **When the inclination angle of an angular axis (parameter No. 8210) is 91° to 179° or 271° to 359°**

Set the directions of single direction positioning (parameter No. 5440) for the angular axis and orthogonal axis so that the directions for the angular axis and for the orthogonal axis are the same (plus and plus, or minus and minus).
When both of the positioning directions for the X–axis (orthogonal axis) and for the Y–axis (angular axis) are plus, the operation shown below is performed.



Move command in the plus direction



Move command in the minus direction

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.2	Single direction positioning
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.2	Single direction positioning

6.6 HELICAL INTERPOLATION

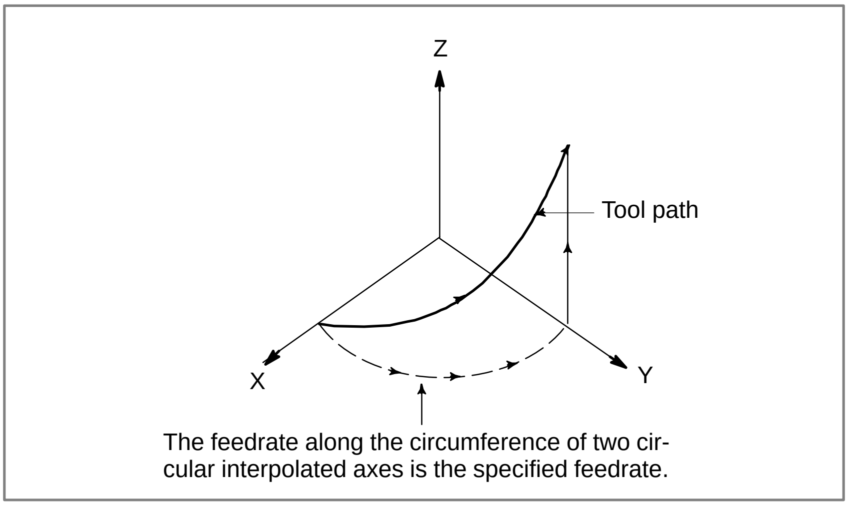
General

Helical interpolation is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

The command method is to simply add one or two move command axes which are not circular interpolation axes. An F command specifies a feedrate along a circular arc. Therefore, the feedrate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so that the linear axis feedrate does not exceed any of the various limit values.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1404								HFC

HFC The feedrate for helical interpolation is:

0 : Clamped so that the feedrates along an arc and linear axis do not exceed the maximum cutting feedrate specified by parameter.

1 : Clamped so that the composite feedrate along an arc and linear axis does not exceed the maximum cutting feedrate specified by parameter.

When HFC is 1, and two linear axes exist, the combined feedrate for the four axes (two axes (arc) + two axes (straight line)) is clamped so that it does not exceed the maximum cutting feedrate.

<Parameters used for clamping>

When HFC is 0

No. 1430: Maximum cutting feedrate for each axis

Since the cutting feedrate for the arc is clamped to the above parameter value, the feedrate along the linear axis is clamped to the smaller parameter value.

Example: No. 1430 X 1000
Y 1200
Z 1400

G17 G03 X0. Y100. R100. Z1000. F5000;

The feedrate along the linear axis is clamped to 1000.

No. 1422: Maximum cutting feedrate (common to all axes)

If parameter No. 1430 is set to 0, the feedrate is clamped to the value set in this parameter.

When HFC is 1

No. 1422: Maximum cutting feedrate (common to all axes)

The cutting feedrate is clamped to the value set in this parameter. The value set with parameter No. 1430 is ignored.

Alarm and message

If more than two axes are specified together with the two axes for circular interpolation in a block specifying a helical interpolation operation, P/S alarm No. 232 is issued.

No.	Message	Description
0232	TOO MANY HELICAL AXIS COMMANDS	Three or more axes are specified as helical axes.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.5	Helical Interpolation
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.4	Helical Interpolation
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.5	Helical Interpolation

6.7 POLAR COORDINATE INTERPOLATION

General

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

Explanations

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.7). Polar coordinate interpolation is performed on this plane.

• Polar coordinate interpolation plane

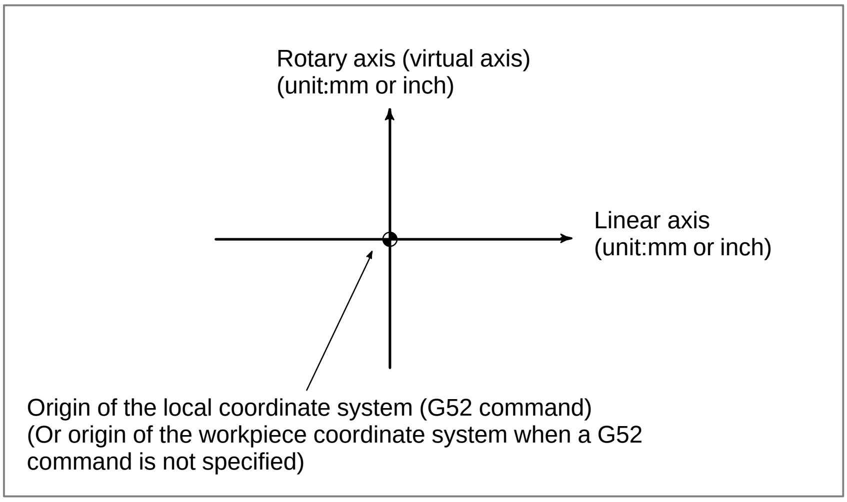


Fig. 6.7 Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
In M series, to specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

5460	Axis (linear axis) specification for polar coordinate interpolation
5461	Axis (rotary axis) specification for polar coordinate interpolarion

[Data type] Byte

[Valid data range] 1, 2, 3, ... control axes count

These parameters set control axis numbers of linear and rotary axes to execute polar interpolation.

5462	Maximum cutting feedrate during polar coordinate interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0, 6 – 240000	0, 6 – 100000
Inch machine	0.1 inch/min	0, 6 – 96000	0, 6 – 48000
Rotation axis	1 deg/min	0, 6 – 240000	0, 6 – 100000

This parameter sets the upper limit of the cutting feedrate that is effective during polar coordinate interpolation. If a feedrate greater than the maximum feedrate is specified during polar coordinate interpolation, it is clamped to the feedrate specified by the parameter. When the setting is 0, the feedrate during polar coordinate interpolation is clamped to the maximum cutting feedrate usually specified with parameter 1422.

Alarm and message

No.	Message	Description
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	II.4.5	Polar Coordinate Interpolation
Series 0*i* Mate–C	OPERATOR’S MANUAL (T series) (B–64134EN)	II.4.4	Polar Coordinate Interpolation

6.8 CYLINDRICAL INTERPOLATION

General

The amount of travel of a rotary axis specified by an angle is internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.
The cylindrical interpolation function allows the side of a cylinder to be developed for programming. So programs such as a program for cylindrical cam grooving can be created very easily.

Use parameter No. 1022 to specify whether the rotation axis is the X–, Y–, or Z–axis, or an axis parallel to one of these axes.
Only one rotation axis can be set for cylindrical interpolation.

Parameter

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

NOTE
When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is the rotation type (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is linear axis type (Is not rounded in 0 to 360°). Absolute coordinate values and relative coordinate values are rounded or not rounded by parameter No. 1008#0 and #2. · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axis roll over function and the index table indexing function (M series).

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

Alarm and message

Number	Message	Description
175	ILLEGAL G107 COMMAND	Conditions when performing cylindrical interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of “G07.1 rotation–axis name radius of cylinder.”
176	IMPROPER G–CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning, such as G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.6	Cylindrical Interpolation
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.6	Cylindrical Interpolation
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.5	Cylindrical Interpolation

6.9 POLYGONAL TURNING (T SERIES)

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

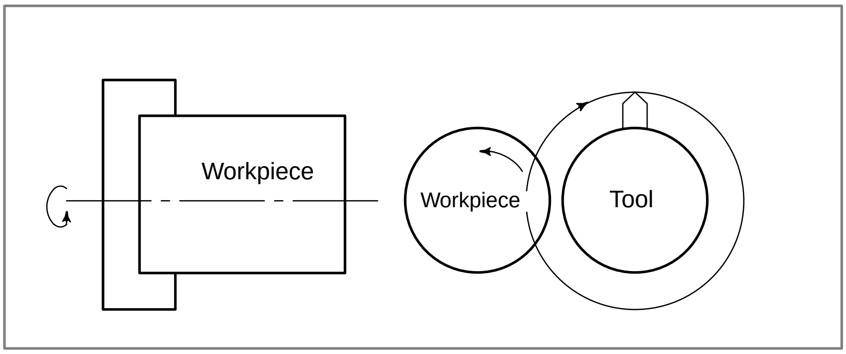


Fig. 6.9 (a) Polygonal turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters,the machining figure can be changed to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using C and X axes of the polar coordinate. The machined figure however, is not exactly polygonal. Generally, polygonal turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

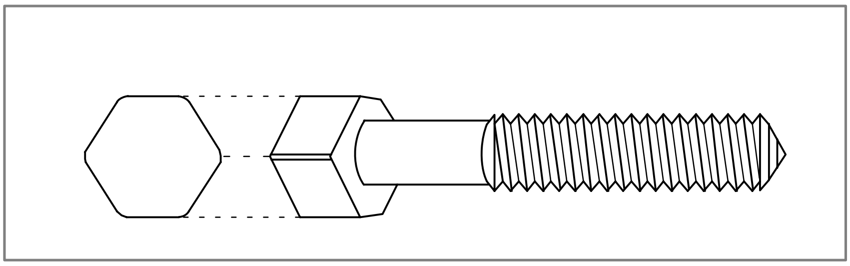


Fig. 6.9 (b) Hexagon bolt

This function controls the workpiece (spindle) and tool (rotation tool axis) so that the relationship between the spindle speed and tool speed is maintained at a constant ratio specified in a command given to the CNC.

(For the principle of polygonal turning, refer to Section 18.1, Part II of the “Operator’s Manual (For Lathe).”)

Following axis can be selected as the tool rotation axis:

· CNC controlled axis (servo axis)

In the following descriptions, the term polygonal turning refers to a turning operation in which a servo axis is used as the tool rotation axis (See Section 6.9.1.).

6.9.1 Polygonal Turning

General

One of the axes (servo axes) controlled by the CNC is assigned as a tool rotation axis. Either serial spindle or analog spindle can be used as a workpiece axis (spindle).

Polygonal turning using a servo axis is detailed in the operator's manual (for lathe).

This section focuses on supplementary information and examples for the connection.

• Spindle connection

A position coder must be mounted on the spindle. However, polygonal turning requires no additional changes to the spindle connection (See Section 9.3.).

Polygonal turning uses the position coder feedback signal to control the positional relationship (cutting position) between the spindle and tool rotation axis, and the ratio of speed.

• Tool rotation axis (servo axis) connection

Parameter No. 7610 specifies the controlled axis (servo axis) to be used as the tool rotation axis.

The same parameter setting as for ordinary servo axes applies to the servo axis connection for polygonal turning except for some parameters.

When the machine is not in the polygonal turning mode, the servo axis specified as the rotation tool axis functions as a feed axis. So, the servo axis can be:

· Used as a subspindle under PMC axis control

· Positioned by a move command from a machining program.

However, be careful about the angle to rotate through and feedrate. Read the operator's manual (for lathe) and the following examples.

• Examples of parameter setting

· The following descriptions exemplify typical parameter setting for polygonal turning using a serial pulse coder (with a million pulse capability).

→ The parameter setting described here is not a must for polygonal turning.

→ Specify typical values for parameters unless otherwise stated.

- Tool rotation axis setting

  This example uses the CNC's fourth axis (connected as the Y–axis) as a rotation tool axis for polygonal turning.

  Parameter No. 7610 = 4
  (controlled axis number for the tool rotation axis)

  The following description assumes that the axis type parameter is set to the fourth axis.

- Servo parameter setting

  Set the servo parameters as listed below:

  CMR = 1
  DMR = 36/1000

(With the above setting, the reference counter capacity is 36000.)

Parameter No. 1820 = 2 (CMR)
Parameter No. 1821 = 36000 (reference counter capacity)
Parameter No. 2084 = 36 (DMR numerator)
Parameter No. 2085 = 1000 (DMR denominator)

For the other servo parameters, specify typical values.

- Parameter setting for polygonal turning

The least command increment, detection unit, the angle to rotate through per rotation for the polygon axis are as follows:

$$\text{Least command increment} = \frac{L \times CMR}{Q \times DMR}$$

$$\text{Detection unit} = \frac{\text{least command increment}}{DMR} = \frac{L}{Q \times DMR}$$

Angle to rotate through per tool axis rotation

$$= \frac{360}{\text{least command increment}}$$

where

L: Tool axis rotation angle per motor rotation (degrees),
(360 × speed increment ratio)

When the servo motor is connected directly to the rotation tool, for example, L = 360. When the tool speed is doubled, L = 720.

Q: Number of pulses per pulse coder rotation
(For a serial pulse coder, Q = 1000000.)

The least command increment specified here is specific to the polygon axis. It is determined regardless of what is specified in parameter No. 1004 (ISA/ISC). However, both ISA and ISC must be set to 0 for IS–B setting.

If the servo motor is connected directly to the rotation tool:

$$\text{Least command increment} = \frac{360 \times 1}{1000000 \times \frac{36}{1000}} = 0.01 \text{ (degrees)}$$

Detection unit = 0.01 (degrees)

$$\text{Angle to rotate through per tool axis rotation} = \frac{360}{0.01} = 36000 \text{ (degrees)}$$

The upper limit to the tool rotation axis speed is:

Maximum servo motor speed × speed increment ratio

Therefore, if the maximum servo motor speed is 2000 min^{-1}, and the servo motor is directly connected to the servo motor:

Upper limit to the tool rotation axis speed = 2000 × 1 = 2000 (min^{-1})

This means the parameters must be set as follows:

No. 7620 = 36000 (angle to rotate through per tool axis rotation)
No. 7621 = 2000 (upper limit to tool rotation axis speed)

- Feedrate parameter setting

Because the least command increment is 0.01 degrees, the input unit for the feedrate is 10 degrees/min.

To obtain a rapid traverse speed of 2000 min^{-1}, for example, specify as follows:

$$\text{No. 1420} = 72000\ (= 2000 \times \frac{360}{10})$$

Also specify other feedrates in 10 degrees/min units.

- Commands from the NC program

When the machine is not performing polygonal turning, the machining program can issue move commands to the polygon axis.

Such commands can be issued in the same way as for ordinary axes. However, be careful about the angle to rotate through and feedrate.

Assuming the polygon axis is the Y–axis, the polygon axis rotates through 0.03 degrees by the following command:

V3;

Likewise, the polygon axis rotates through 10.00 degrees by the following command:

V1.0;

The feedrate unit is also increased by tenfold.

The current position of the polygon in the machine coordinate system is normalized according to the value specified by parameter No. 7620.

Typical values range from 0.000 to 35.999.

Signal

Polygon synchronization under way signal PSYN <F063#7>

[Classification] Output signal

[Function] Informs the PMC that the machine is in the polygon turning mode.

[Output condition] The polygon synchronization signal is set to logical “1” by the polygon turning mode command (G51.2) and stays at “1” during the polygonal turning mode.

The signal is reset to logical “0” by the polygon turning mode reset command (G50.2) or a reset. It stays at logical “0” when the machine is not in the polygonal turning mode.

· Other signals (related to the tool rotation axis)

→ Some signals related to the CNC controlled axis used as the tool rotation axis may be made ineffective depending on whether the machine is in the polygonal turning mode.

For these signals, read the note in operator’s manual (for lathe).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F063	PSYN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Data type] Bit

PLZ Synchronous axis using G28 command

0: Returns to the reference position in the same sequence as the manual reference position return.

1: Returns to the reference position by positioning at a rapid traverse. The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return–to–reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

[Data type] Byte

[Valid data range] 1, 2, 3, . . . number of control axes

This parameter sets the control axis number of a rotation tool axis used for polygon turning.

7620	Movement of tool rotation axis per revolution

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 1 to 9999999

This parameter sets the movement of a tool rotation axis per revolution.

7621	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] For polygonal turning using servo motors:

$$0 \text{ to } \frac{1.2 \times 10^8}{\text{set value of the parameter No. 7620}}$$

This parameter sets the upper–limit rotation speed of a tool rotation axis. The rotation speed of the tool rotation axis is clamped by the upper–limit rotation speed during polygon turning. The spindle and tool rotation axis go out of synchronization when the rotation speed is clamped (P/S alarm No. 5018).

Alarm and message

Number	Message	Description
217	DUPLICATE G251 (COMMANDS)	G51.2 (or G251) is further commanded in the polygonal turning mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G51.2 (or the G251) block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G51.2 (or G251) and G50.2 (or G250) are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHR–MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHR–MODE	Polygon machining synchronous operation and Cs contouring control or balance cutting are executed at a time. Modify the program.

Caution

CAUTION

1 Before issuing a G51.2, rotate the spindle. If it is not rotating when the G51.2 is issued, the program stops to wait for a one–rotation signal from the position coder on the spindle. This does not apply to a dry run.
2 A reset releases the polygonal turning mode.
3 Machine a workpiece at the same spindle speed until finish machining for the workpiece.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.19.1	POLYGONAL TURNING

6.10 NORMAL DIRECTION CONTROL (M SERIES)

General

When a tool with a rotation axis (C–axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C–axis is always perpendicular to the tool path (Fig. 6.10).

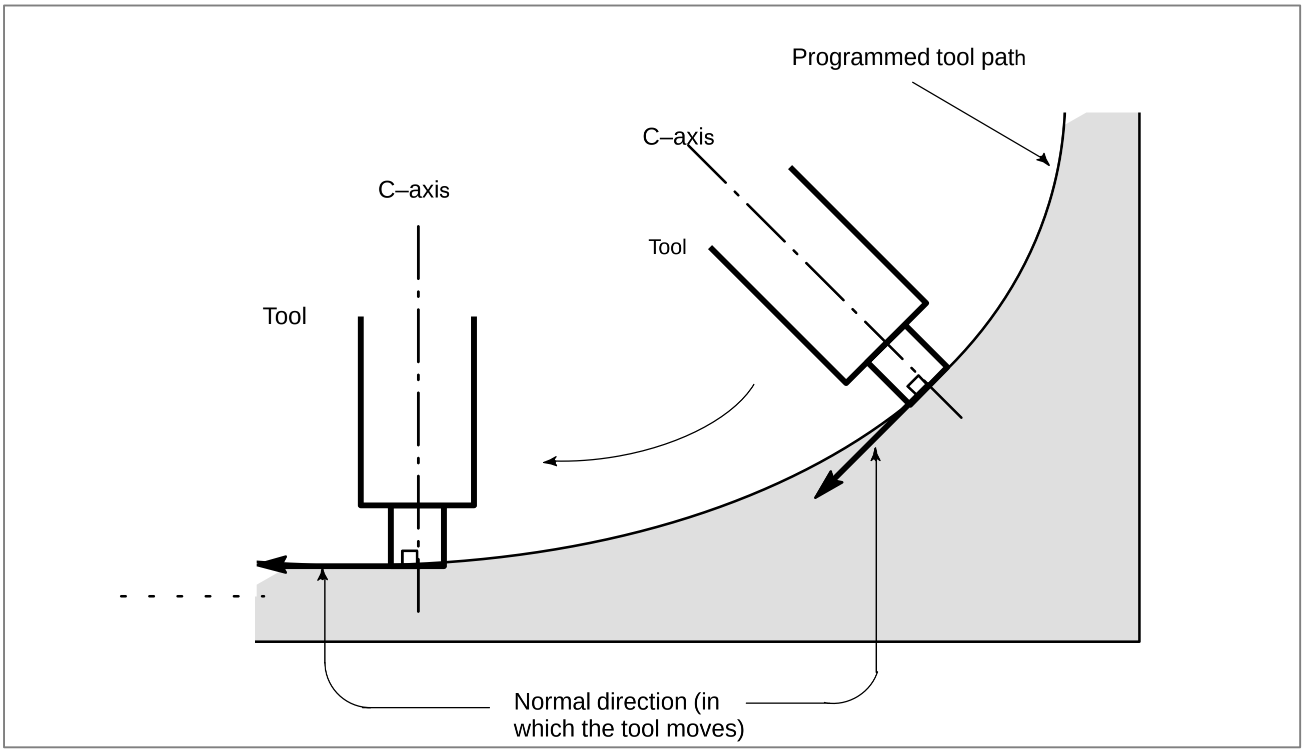


Fig. 6.10 Sample Movement of the tool

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X–and Y–axes in rapid traverse (G00) mode, the rapid traverse rate is applied.

If the federate of the C axis exceeds the maximum cutting feedrate of the C axis specified to parameter No. 1422, the federate of each of the other axes is clamped to keep the federate of the C axis below the maximum cutting feedrate of the C axis.

Parameter

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

> **NOTE**
> When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

[Valid data range] ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of the rotation type. (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is of linear axis type (i.e. not rounded in 0 to 360°). · Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axes roll over function and the index table indexing function (M series).

NOTE

The rotation axis must be set to the normal direction control axis.

5480	Number of the axis for controlling the normal direction

[Data type] Byte

[Valid data range] 1 to the maximum control axis number

This parameter sets the control axis number of the axis which controls the normal direction.

5481	Rotation feedrate of normal direction control axis

[Data type] Word

[Unit of data] 1 deg/min

[Valid data range] 1 to 15000

This parameter sets the feedrate of a normal direction control axis that is inserted at the start point of a block during normal direction control.

5482	Limit value that ignores the rotation insertion of normal direction control axis

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 1 to 99999999

The rotation block of a normal direction control axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting value. The ignored rotation angle is added to the next rotation insertion angle. The block insertion is then judged.

NOTE
1. No rotation block is inserted when 360 or more degrees are set.
2. If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation is 180 or more degrees.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

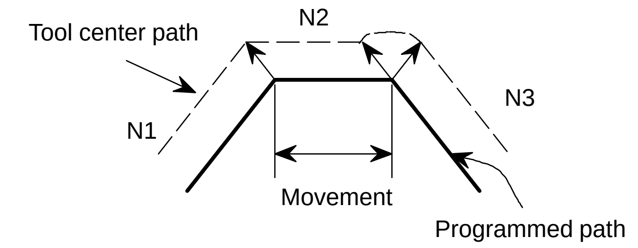
[Data type] Two–word

[Unit of data]

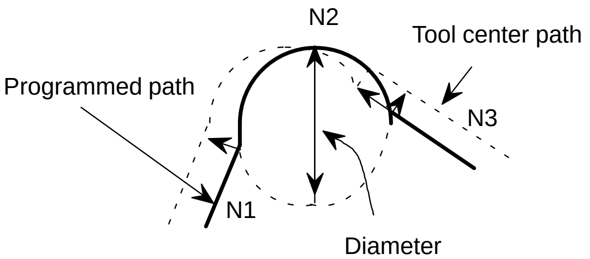
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the limit value of movement at the normal direction angle of a preceding block.



For straight line
Block N2 is machined with the tool being normal to block N1 when the movement of N2 in the figure on the left does not exceed the set value.



For arc
Arc N2 is machined with the tool being normal to block N1 when the arc diameter of N2 in the figure on the left does not exceed the setting value. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

	#7	#6	#5	#4	#3	#2	#1	#0
5484							CTI	SDC

[Data type] Bit

SDC In normal direction control:

0 : A C–axis movement is automatically inserted between blocks so that the C–axis is directed at right angles to the direction of motion at the start point of each block. (After movement on the C–axis, movement (along the X–axis and Y–axis) specified by the block is performed.)

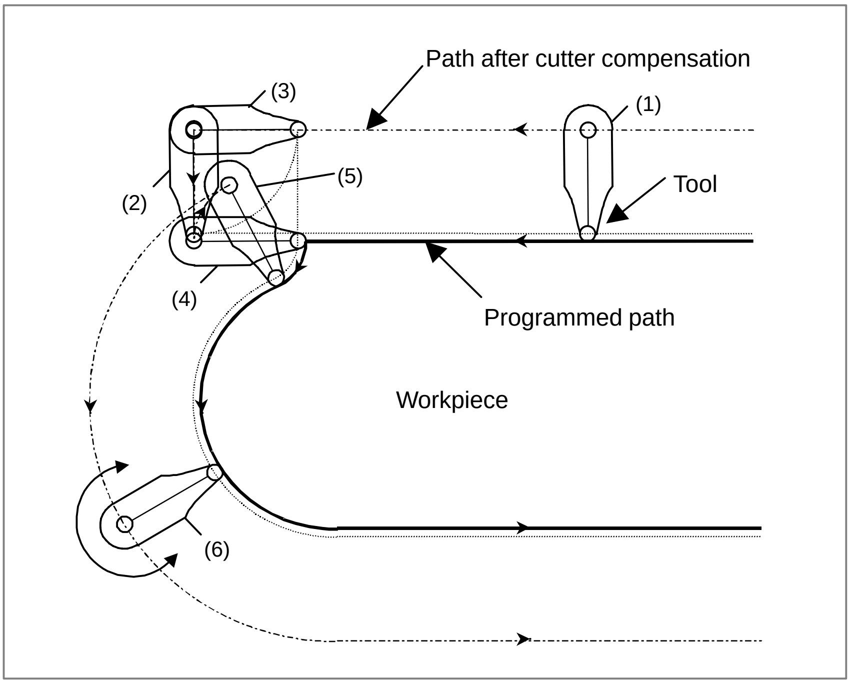
1 : If the amount of C–axis movement is smaller than the value set in parameter No.5485, a C–axis movement is not inserted before a block. Instead, it is performed together with movement along the X–axis and Y–axis.

CTI If such an arc that the vector from the center of the arc to a start point rotates in the reverse direction after cutter compensation is specified during normal direction control in the cutter compensation C mode:

0 : P/S 041 alarm is issued.

1 : The command is executed.

If this parameter is set to 1, and such an arc that the vector from the center of the arc to a start point rotates in the reverse direction after cutter compensation is specified during normal direction control in the cutter compensation C mode (see the tool path from (4) to (5) in the figure below), the tool is controlled so that the tool faces in the direction at right angles to the move direction (programmed path) before cutter compensation (see the tool path from (2) to (3) in the figure below).
Thus, as shown by the programmed path from (4) to (5) in the figure below, the inside of an arc where the radius of the workpiece is smaller than the compensation value of the tool can be cut.



NOTE
When this parameter is set to 1, no interference check is made in cutter compensation C.

5485	Limit imposed on the insertion of a single block for rotation about the normal direction control axis

[Data type] 2–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Units
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 1 to 99999999

When normal direction control is applied, the amount of movement (rotation angle) on the normal direction control axis (C–axis), calculated so that the C–axis is directed at right angles to the direction of motion at the start point of a block, may be smaller than the value specified in this parameter. In such a case, the C–axis movement is not inserted before the movement (along the X–axis and Y–axis) specified by the block. Instead, the C–axis movement is performed together with the movement specified by the block. If the amount of movement (rotation angle) on the C–axis is greater than or equal to the value specified with this parameter, the C–axis movement is inserted, and the movement specified by the block is made after the completion of the C–axis movement.

NOTE
This parameter is enabled when the SDC parameter (bit 0 of parameter No.5484) is set to 1. If a value equal to or greater than 180 degrees is specified, a C–axis movement is inserted only when circular interpolation involving a C–axis rotation of 180 degrees or more is performed.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.14.9	Normal Direction Control

6.11 LINEAR INTERPOLATION (G28, G30, G53)

General

When positioning operation of linear interpolation type is specified (bit 1 (LRP) of parameter No. 1401 = 1), the following operations can also be set as operations of linear interpolation type by setting bit 4 (ZLN) of parameter No. 1015 to 1:

- Movement from an intermediate point to a reference position in automatic reference position return operation (G28)
- Movement from an intermediate point to a reference position in second, third, or fourth reference position return operation (G30)
- Positioning by machine coordinate system selection (G53)

When any of the operations above is set as an operation of linear interpolation type, acceleration/deceleration control follows the setting of bit 4 (RCT) of parameter No. 1603.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1015				ZLN				

[Data type] Bit

ZLN When positioning of linear interpolation type is specified (bit 1 (LRP) of parameter No. 1401 = 1), automatic reference position return operation (G28), second to fourth reference position return operation, and machine coordinate system selection are set as:

0 : Positioning of non–linear interpolation type.
1 : Positioning of linear interpolation type.

This parameter is valid when bit 1 (LRP) of parameter No. 1401 = 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1603				RCT				

[Data type] Bit

RCT When positioning of linear interpolation type is specified (bit 1 (LRP) of parameter No. 1401 = 1), acceleration/deceleration control is based on:

0 : Constant acceleration/deceleration method.
1 : Constant time (time constant) method.

Note

1 Manual intervention

Positioning of non–linear interpolation type is performed if the automatic operation stop state is set by feed hold or mode switching during movement then the subsequent operation of the program is performed after the machine is moved by manual operation.

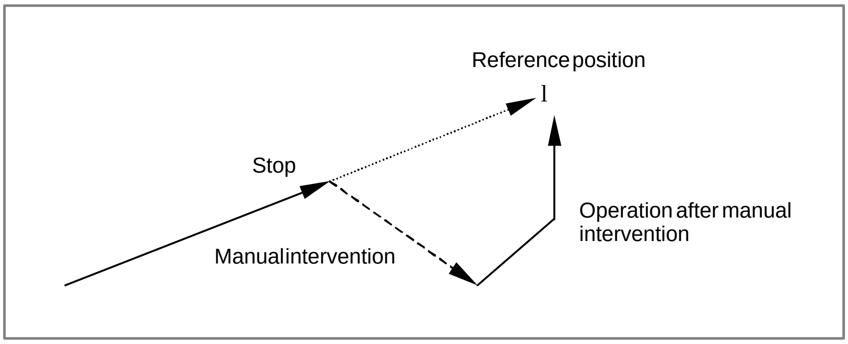


Fig. 6.11 Operation after manual intervention

If the machine position remains unchanged before and after manual intervention (if the machine is not moved in the automatic operation stop state or the machine is returned to the position before manual intervention), positioning of linear interpolation type is performed in the subsequent operation of the program.

2 Automatic reference position return operation of low–speed type (G28)

If reference position return operation is not performed for a specified axis even once after the power is turned on in automatic reference position return operation (G28), a reference position return operation of low–speed type is performed for the axis.

At this time, positioning of non–linear interpolation type is performed.

For example, if G28X0Y0Z0; is specified when reference position return operation is completed for the X–axis and Y–axis, and is not completed for the Z–axis, positioning of non–linear interpolation type is performed.

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL

7.1 FEEDRATE CONTROL

The feed functions control the feedrate of the tool. The following two feed functions are available:

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse rate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

Override can be applied to a rapid traverse rate or cutting feedrate using the override signal.

7.1.1 Rapid Traverse Rate

General

The positioning command (G00) positions the tool by rapid traverse.

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ ; Dimension word for the end point

In rapid traverse, the next block is executed after the specified rate becomes 0 and the servo motor reaches a certain range set by the parameter (No.1826) (in–position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse rate need be programmed.
The following overrides can be applied to a rapid traverse rate with the rapid traverse override signal:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
In addition, the use of the 1% rapid traverse override selection signal allows a rapid traverse override of between 0% and 100% to be applied in 1% steps.

Signal

Rapid traversing signal RPDO <F002#1>

[Function] This signal indicates that a move command is executed at rapid traverse.

[Output condition] “1” indicates that an axis starts moving after rapid traverse has been selected.
“0” indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

> **NOTE**
> 1 The rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. The manual rapid traverse also includes the rapid traverse in reference position return.
> 2 Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement is started.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002							RPDO	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR		RFO			LRP	

[Data type] Bit

LRP Positioning (G00)

0 : Positioning is performed with non–linear type positioning so that the tool moves along each axis independently at rapid traverse.
1 : Positioning is performed with linear interpolation so that the tool moves in a straight line.

RFO When cutting feedrate override is 0% during rapid traverse

0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

RDR Dry run for rapid traverse command

0 : Disabled
1 : Enabled

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rate of manual rapid traverse for each axis when the rapid traverse override is 100% for each axis.

NOTE
If 0 is set, the rate set in parameter 1420 is assumed.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.2	Rapid traverse
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.5.2	Rapid traverse
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.2	Rapid traverse
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.5.2	Rapid traverse

7.1.2 Cutting Feedrate Clamp

General

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit. For M series, the upper limit can be set on the cutting feedrate for each axis with parameter No. 1430.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

> **NOTE**
> To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead. (M series)

1430	
	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotation axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.

A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

NOTE
1. This parameter is effective only in linear and circular interpolation. In polar coordinate and cylindrical interpolation, the maximum feedrate for all axes specified in parameter No. 1422 is effective.
2. If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

Warning

WARNING
CNC calculation may involve a feedrate error of +2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.3	Cutting Feed
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.5.3	Cutting Feed
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.3	Cutting Feed
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.5.3	Cutting Feed

7.1.3
Feed Per Minute

General

- **Feed per minute (G94)**

After specifying G94 (G98 for T series) (in the feed per minute mode), the amount of feed of the tool per minute is specified by setting a number after F. G94 (G98 for T series) is a modal code. Once a G94 (G98 for T series) is specified, it is valid until G95 (G99 for T series) (feed per revolution) is specified. At power–on, the feed per minute mode (feed per revolution mode for T series) is set.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the feedrate override signal.

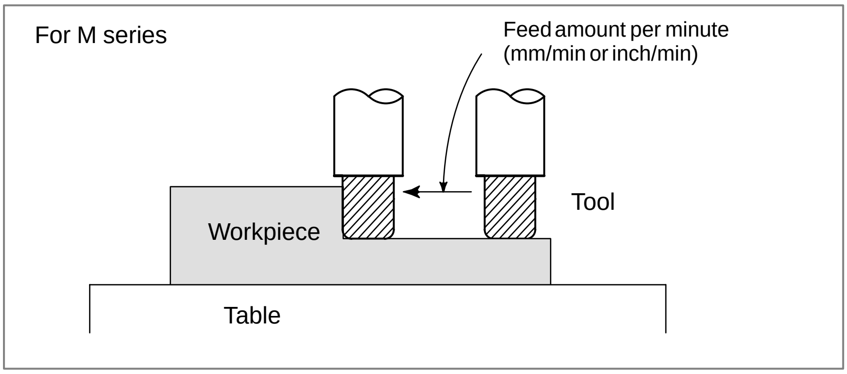


Fig. 7.1.3 Feed per minute

> **CAUTION**
> No override can be used for any commands such as for threading.

Format

```
For M series
        G94;    G code for feed per minute (Group 05)
        F_;     Feed rate (mm/min or inch/min)
```

```
For T series
        G98;    G code for feed per minute (Group 05)
        F_;     Feed rate (mm/min or inch/min)
```

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

> **NOTE**
> M series does not use this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

	#7	#6	#5	#4	#3	#2	#1	#0
3401							FCD	

[Data type] Bit

FCD When an F command and a G command (G98, G99) for feed per minute or feed per rotation are specified in the same block, and the G command (G98, G99) is specified after the F command, the F command is:

0 : Assumed to be specified in the mode (G98 or G99) when the F command is specified

1 : Assumed to be specified in the mode of the G command (G98 or G99) of the same block

> **NOTE**
> 1 When FCD = 1:
> If the block containing a G command (G98, G99) does not include an F command, the last F command specified is assumed to be specified in the G command mode of the block.
> **Example 1:** N1 G99 ;
> N2 Faaaa G98 ;
> - Faaaa is assumed to be specified in the G98 mode.
> N3 Fbbbb ;
> - Fbbbb is assumed to be specified in the G98 mode.
> N4 G99 ;
> - Fbbbb is assumed to be specified in the G99 mode.
> 2 In G code system B or C, G98 and G99 function are specified in G94 and G95.

	#7	#6	#5	#4	#3	#2	#1	#0
3402			FPM					

[Data type] Bit

FPM When the power is turned on:
0 : Feed per revolution mode is entered.
1 : Feed per minute mode is entered.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.5.3	Cutting feed
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.5.3	Cutting feed
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.5.3	Cutting feed
	OPERATOR’S MANUAL (T series) (B–64134EN)	II.5.3	Cutting feed

7.1.4 Feed Per Revolution/ Manual Feed Per Revolution

General

- **Feed per revolution**

After specifying G95 (G99 for T series) (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 (G99 for T series) is a modal code. Once a G95 is specified, it is valid until G94 (G98 for T series) (feed per minute) is specified.
An override of between 0 and 254% (in steps of 1%) can be applied to feed per rotation, using the feedrate override signals (*FV0 to *FV7). (See Subsection 7.1.6.2.)

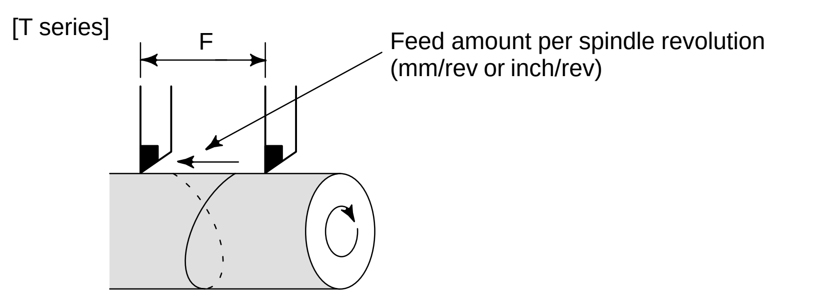


Fig. 7.1.4 Feed per revolution

- **Manual feed per revolution**

Jog feedrate can be specified by feed per revolution.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC
								NPC

[Data type] Bit

JRV Jog feed

0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

> **NOTE**
> Specify a feedrate in parameter No. 1423.

NPC Feed per revolution command

0 : The feed per revolution command is ignored when the position coder is not installed.
1 : The feed per revolution command is accepted even when the position coder is not installed. (The CNC automatically converts the feed per revolution command to a feed–per–minute operation.)

1423	Feedrate in jog feed for each axis

[Data type] Word axis

When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a feedrate in jog feed (feed per revolution) with an override of 100% applied to the jog feedrate.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 to 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

Caution

CAUTION
When the speed of the spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.5.3	Cutting feed
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.5.3	Cutting feed

7.1.5 F1-digit Feed (M series)

General

When a one–digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 1451 to 1459) is used. When F0 is specified, the rapid traverse rate is applied.

The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing F1–digit feedrate on the machine operator's panel, then by rotating the manual pulse generator.

The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{Fmax}{100X}$$

Fmax : feedrate upper limit for F1–F4 set by parameter 1460, or feedrate upper limit for F5–F9 set by parameter 1461

X : any value of 1–127 set by parameter 1450

The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the screen.

Signal

F1-digit feed select signal F1D <G016#7>

[Classification] Input signal

[Function] Increases or decreases F1-digit speed set by the parameters No. 1451 to 1459 using the manual pulse generator.
Since the manual pulse generator may also be used for axis feeding, signal F1D (G016#7) designates which function may be used.

[Operation] When the signal is "1", the F1–digit speed can be increased/decreased using the manual pulse generator.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016	F1D							

Parameter

1450	Number of revolution of manual pulse generator to reach maximum feedrate

[Data type] Byte

[Valid data range] 1 to 127

Set the constant that determines the change in feedrate as the manual pulse generator is rotated one graduation during F1-digit feed.

$$\Delta F = \frac{Fmaxi}{100n} \text{ (where, i=1 or 2)}$$

In the above equation, n is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi. Fmaxi refers to the upper limit of the feedrate for an F1-digit feed command, and set it in parameter 1460 or 1461.

Fmax1: Upper limit of the feedrate for F1 to F4 (parameter 1460)

Fmax2: Upper limit of the feedrate for F5 to F9 (parameter 1461)

1451	Feedrate for F1 digit command F1
1452	Feedrate for F1 digit command F2
1453	Feedrate for F1 digit command F3
1454	Feedrate for F1 digit command F4
1455	Feedrate for F1 digit command F5
1456	Feedrate for F1 digit command F6
1457	Feedrate for F1 digit command F7
1458	Feedrate for F1 digit command F8
1459	Feedrate for F1 digit command F9

These parameters can be set at “Setting screen”.

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	0.1 mm/min	6 – 150000	6 – 120000
Inch machine	0.01 inch/min	6 – 60000	6 – 48000
Rotation axis	0.1 deg/min	6 – 150000	6 – 120000

Set Feedrates for F1-digit feed commands F1 to F9.

When an F1-digit feed command is executed, as the feedrate is changed by turning the manual pulse generator, these parameter values also change accordingly.

1460	Upper limit of feedrate for the F1–digit feed command (F1 to F4)
1461	Upper limit of feedrate for the F1-digit feed command (F5 to F9)

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the upper limit of feedrate for the F1-digit feed command.

As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If an F1-digit feed command F1 to F4 is executed, the upper limit is that set in parameter 1460. If an F1-digit command F5 to F9 is executed, the upper limit is that set in parameter 1461.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.3	Cutting feed
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.3	Cutting feed

7.1.6 Feedrate Inverse Time Specification (M series)

General

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN).

- **Linear interpolation (G01)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Distance}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Distance: mm (metric input)
inch (inch input)

- **Circular interpolation (G02, G03)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Circle radius}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Circle radius:
mm (metric input)
inch (inch input)

CAUTION
In circular interpolation, the distance is not an actual distance of the block but the speed is calculated from the circle radius.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed, F0 was specified or the feedrate calculated (less than 0.001 mm/min, for metric input or less than 0.00001 inch for inch input) becomes less than an allowable range.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.3	Cutting feed

7.1.7 Override

7.1.7.1 Rapid traverse override

General

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).
Also, 1% rapid traverse override select signal allows rapid traverse override every 1% in the range of 0 to 100%.

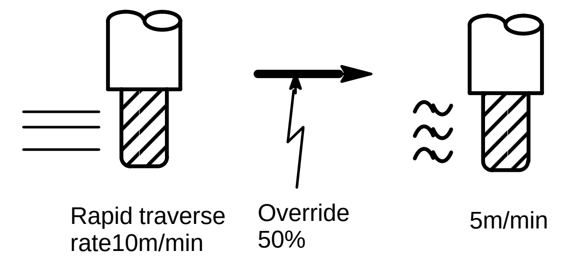


Fig.7.1.7.1 Rapid traverse override

- **Feedrate**

Actual feedrate is obtained by multiplying the rapid traverse rate preset by parameter no.1420 by the override value determined by this signal, whether in automatic or manual operation (including manual reference position return).

- **F0 rate**

For F0 value, an absolute value is set by parameter no.1421 within a range of 0 to rapid traverse rate (for each axis).

- **1% step rapid traverse override selection signal**

1% step rapid traverse override selection signal HROV determines whether rapid traverse override specified with rapid traverse override signals ROV1 and ROV2 is used or 1% step rapid traverse override is used.

When signal HROV is 0, override is applied to the rapid traverse rate using signals ROV1 and ROV2.

When signal HROV is 1, ROV1 and ROV2 are ignored, 1% step rapid traverse override signals *HROV0 to *HROV6 being used to override the rapid traverse rate.

- **PMC axis control**

These 1% step rapid traverse override signals are also effective to the rapid traverse rate for the PMC axis. When rapid traverse override is applied to the PMC axis (using signals ROV1E and ROV2E) with the setting of the OVE bit (bit 2 of parameter No. 8001) independently of the CNC, the 1% step rapid traverse override signals are ineffective.

Signal

Rapid traverse override signal ROV1,ROV2 <G014#0, #1>

[Classification] Input signal

[Function] These signals override the rapid traverse rate

[Operation] These code signals correspond to the rates as follows:

Rapid traverse override		Override value
ROV2	ROV1	
0	0	100 %
0	1	50 %
1	0	25 %
1	1	F0 %

Fo: Set in parameter No. 1421

1% step rapid traverse override selection signal HROV <G096 #7>

[Classification] Input signal

[Function] Selects the rapid traverse override signals or the 1% step rapid traverse override signals.

[Operation] When HROV is 1, signals *HROV0 to *HROV6 are effective and rapid traverse override with signals ROV1 and ROV2 is ignored.

When HROV is 0, signals *HROV0 to *HROV6 are ineffective, and rapid traverse override with signals ROV1 and ROV2 are effective.

1% step rapid traverse override signals *HROV0 to *HROV6 <G096 #0 to #6>

[Classification] Input signal

[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 1%.

[Operation] These seven signals give a binary code indicating an override applied to the rapid traverse rate.

· When a binary code corresponding to an override value of 101% to 127% is specified, the applied override is clamped at 100%.

· Signals *HROV0 to *HROV6 are inverted signals.
To set an override value of 1%, set signals *HROV0 to *HROV6 to 1111110, which corresponds to a binary code of 0000001.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G014							ROV2	ROV1
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0

Parameter

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 15000	6 – 12000
Inch machine	0.1 inch/min	30 – 6000	6 – 4800
Rotation axis	1 deg/min	30 – 15000	6 – 12000

Set the F0 rate of the rapid traverse override for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Data type] Bit

OVE Dry run and override signals during axis control by the PMC

0 : Use the same signals as CNC
(1) Feedrate override signal *FV0 to *FV7
(2) Override cancel signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT

1 : Use dedicated axis control signals by the PMC.
(1) Feedrate override signal *FV0E to *FV7E
(2) Override cancel signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.5.3	Rapid traverse override
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.5.3	Rapid traverse override
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.5.3	Rapid traverse override
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.5.3	Rapid traverse override

7.1.7.2
Feedrate override

General

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial.This feature is used to check a program. For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

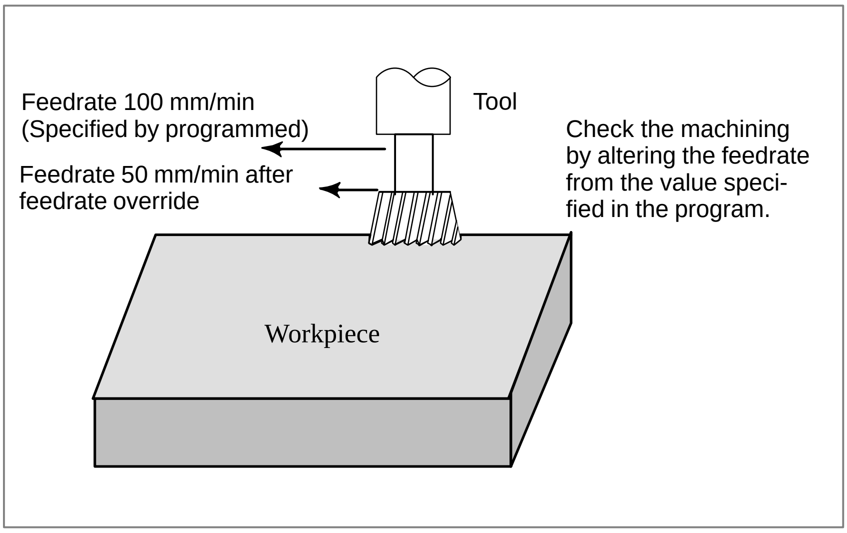


Fig. 7.1.7.2 Feedrate override

Signal

Feedrate Override signal *FV0 to *FV7 <G012>

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

$$\text{Override value} = \sum_{i=0}^{7} (2^i \times Vi)\ \%$$

Vi=0 when *FVi is "1" and
Vi=1 when *FVi is "0"

These signals have the following weight.

*FV0 : 1%	*FV1 : 2%
*FV2 : 4%	*FV3 : 8%
*FV4 : 16%	*FV5 : 32%
*FV6 : 64%	*FV7 : 128 %

When all signals are "0", they are regarded as overriding 0% in the same way as when all signals are "1" .
Thus, the override is selectable in steps over a range of 0 to 254%.

[Operation] Actual feedrate is obtained by multiplying the specified speed by the override value selected by this signal.
The override is regarded as 100%, regardless of this signal, in the following cases:

·Override cancel signal OVC is "1".
·During cutting in tap cycle of canned cycle;
·Tapping mode (63); or
·Thread cutting is in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RFO				

[Data type] Bit

RFO When cutting feedrate override is 0% during rapid traverse,

0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.5.3	Cutting feed
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.5.3	Cutting feed

7.1.7.3 Override cancel

General

The override cancel signal fixes the feedrate override to 100%.

Signal

Override cancel signal OVC <G006#4>

[Classification] Input signal

[Function] Feedrate override is fixed to 100%.

[Operation] When the signal is "1", the CNC operates as follows:

· Feedrate override is fixed to 100% irrespective of feedrate override signal.

· Rapid traverse override and spindle speed override are not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006				OVC				

7.1.8 Automatic Corner Override (M series)

General

- **Inner corner automatic override**

When G62 is specified, and the tool path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.
There are four types of inner corners (Fig. 7.1.8).
$2° \leqq \theta \leqq \theta p \leqq 178°$ in Fig. 7.1.8
θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

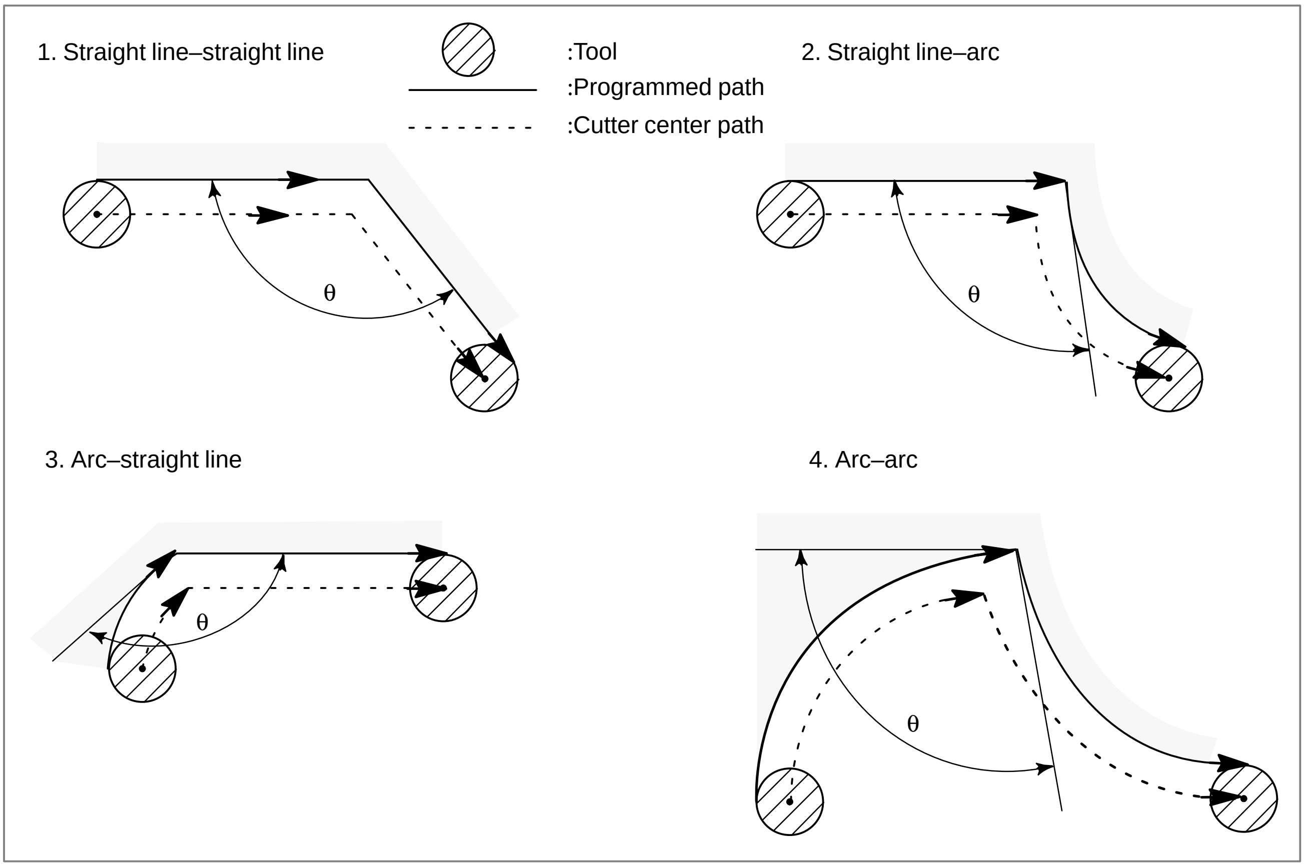


Fig. 7.1.8 Inner corner

> **WARNING**
> When the block before a corner is a start–up block, or the block after a corner includes G41 or G42, the feedrate is not overridden. The feedrate override function is disabled when the offset value is 0.

- **Override value**

An override value is set with parameter No. 1712. An override value is valid even for dry run and F1–digit feed specification.
In the feed per minute mode, the actual feedrate is as follows:

F × (inner corner automatic override) × (feedrate override)

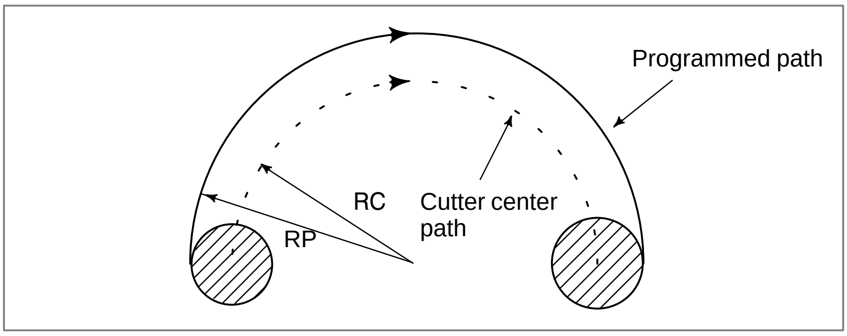
- **Internal circular cutting feedrate change**

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below. This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F \times \frac{Rc}{Rp}$$

Rc : Cutter center path radius
Rp : Programmed radius

It is also valid for the dry run and the F1–digit feed command.



Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp≐0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp≦MDR, the feedrate of the tool is (F×MDR).

> **CAUTION**
> When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times \text{(inner corner override)} \times \text{(feedrate override)}$$

Parameter

1710	Minimum deceleration ratio (MDR) of the inner circular cutting rate in automatic corner override

[Data type] Byte

[Unit of data] %

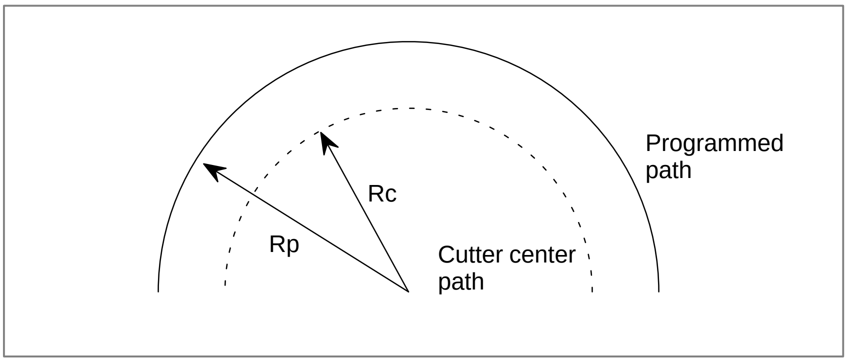
[Valid data range] 1 to 100

Set the minimum deceleration ratio (MDR) in changing the inner circular cutting feedrate by automatic corner override.

In circular cutting with an inward offset, the actual feedrate for a specified feedrate (F) becomes as follows:

$$F \times \frac{Rc}{Rp} \qquad \left(\begin{array}{l} \text{Rc: Radius of the path of the cutter's center} \\ \text{Rp: Programmed radius} \end{array} \right)$$

As the actual feedrate becomes the value obtained from the above equation, the specified rate F can be achieved on the program path.



If Rc is too samall in comparison with Rp so that $\frac{Rc}{Rp} \doteq 0$, the cutter will stop. To prevent this, the minimum deceleration ratio (MDR) is set.

When $\frac{Rc}{Rp} \doteq 0$,

the actural rate becomes as follows:

$F \times (MDR)$

1711	Angle (θp) to recognize the inner corner in automatic override

[Data type] Byte

[Unit of data] Degree

[Valid data range] 1 to 179 (standard value = 91)

Set the angle to recognize the inner corner when automatic corner override is performed for the inner corner.

1712	Amount of automatic override for an inner corner

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100

Set inner corner automatic override value when automatic corner override is performed.

1713	Distance Le from the starting point in inner corner automatic override

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

Set distance Le from the starting point in an inner comer for automatic corner override.

1714	Distance Ls up to the ending point in inner corner automatic override

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

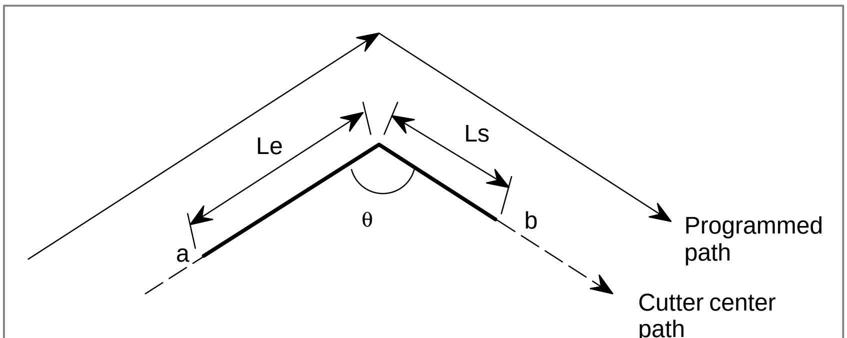
Set distance Ls up to the end point in an inner corner for automatic corner override.

If $\theta \leqq \theta$p, the inside of a comer is recognized. (θ is set in parameter 1711.)

When an inner corner is recognized, the feedrate is overridden in the range of Le in the block immediately before the intersection of the corner and Ls in the next block following the intersection.

Ls and Le are each a straight line connecting the intersection of the corner and a given point on the path of the cutter's center.

Ls and Le are set in parameters 1713 and 1714.



An override is applied from point a to b.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.5.4.2	Automatic Override for Inner Corners
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.5.4.2	Automatic Override for Inner Corners

7.1.9 External Deceleration

General

These signals decelerate the feedrate of the control axes down to the speed which has been set by parameters.
Three types of setting points are available for external deceleration.

Specifications of external deceleration

Three types of setting points such as the external deceleration feedrates (rapid traverse, cutting feed) and external deceleration signals for the external deceleration function are available.

External deceleration 2 can be enabled by setting bit 0 (EXD2) of parameter No. 1406.

If multiple deceleration conditions are specified by application of multiple external deceleration signals during machining, the lowest deceleration feedrate is employed.

Example)

Programmed cutting feedrate	2400 [mm/min]
Cutting feed external deceleration feedrate 1	800 [mm/min]
Cutting feed external deceleration feedrate 2	350 [mm/min]
Cutting feed external deceleration feedrate 3	400 [mm/min]

If external deceleration feedrates 1 through 3 above are applied, the machining feedrate is 350 [mm/min].

The feedrate in manual handle feed is usually clamped to the manual rapid traverse rate. With the HNDLF (G023#3) signal, however, the feedrate in manual handle feed can be clamped to the maximum feedrate (parameter Nos. 1434, 1442, and 1445) for manual handle feed.

External deceleration signal 1 <G118, G119> does not affect the maximum feedrate for manual handle feed. The HNDLF (G23#3) signal switches the maximum feedrate for manual handle feed from the manual rapid traverse rate to the value of parameter No. 1434.

External deceleration signal 2 <G101, G103> can switch the maximum feedrate for manual handle feed to the value of parameter No. 1442. External deceleration signal 3 <G107, G109> can switch the maximum feedrate for manual handle feed to the value of parameter No. 1445. In manual handle feed, when external deceleration signal 2 or 3 of the handle axis is set to 0 in the positive or negative direction, the feedrate is switched to the maximum feedrate for manual handle feed. If multiple deceleration conditions are specified by application of multiple external deceleration signals, the lowest deceleration feedrate is employed.

Signal

Handle feed maximum feedrate switch signal HNDLF <G023#3>

[Classification] Input signal

[Function] Chooses whether to clamp the feedrate in manual handle feed to the manual rapid traverse rate (parameter No. 1424) or the maximum feedrate (parameter Nos. 1434, 1442, and 1445) for manual handle feed.

[Operation] When this signal is set to 0, the feedrate in manual handle feed is clamped to the manual rapid traverse rate (parameter No. 1424).
When this signal is set to 1, the feedrate in manual handle feed is clamped to the maximum feedrate (parameter Nos. 1434, 1442, and 1445) for manual handle feed.
For the relationship with the external deceleration signals, see the specifications of external deceleration.

External deceleration signals 1 *+ED1 to *+ED4 <G118#0 to #3> *–ED1 to *–ED4 <G120#0 to #3>

External deceleration signals 2 *+ED21 to *+ED24 <G101#0 to #3> *–ED21 to *–ED24 <G103#0 to #3>

External deceleration signals 3 *+ED31 to *+ED34 <G107#0 to #3> *–ED31 to *–ED34 <G109#0 to #3>

[Classification] Input signal

[Function] These signals are used to apply deceleration, and provided for each direction of each control axis; The plus (+)/minus (–) sign of a signal name indicates the direction of deceleration. X, namely, the number after ED, indicates a type of external deceleration. Y, namely, the last number of a signal name, indicates a controlled axis number.

* + ED X Y

Y:
1 : The 1st axis is decelerated.
2 : The 2nd axis is decelerated.
3 : The 3rd axis is decelerated.
: :

X:
No number : External deceleration 1
2 : External deceleration 2
3 : External deceleration 3

+/–:
+ … The feed is decelerated in the plus (+) direction.
– … The feed is decelerated in the minus (–) direction.

[Operation] When a signal becomes “0”, the corresponding axis decelerates to stop in the specified direction.
For the relationship with the handle feed maximum feedrate switch signal, see the specifications of external deceleration.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G023					HNDLF			
G118					*+ED4	*+ED3	*+ED2	*+ED1
G120					*–ED4	*–ED3	*–ED2	*–ED1
G101					*+ED24	*+ED23	*+ED22	*+ED21
G103					*–ED24	*–ED23	*–ED22	*–ED21
G107					*+ED34	*+ED33	*+ED32	*+ED31
G109					*–ED34	*–ED33	*–ED32	*–ED31

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				

[Data type] Bit axis

EDPx External deceleration signal in the positive direction for each axis

0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

EDMx External deceleration signal in the negative direction for each axis

0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

	#7	#6	#5	#4	#3	#2	#1	#0
1405			EDR					

[Data type] Bit

EDR Selects a parameter for setting the external deceleration applied during interpolation type rapid traverse.

0 : Parameter No. 1426 is used for setting the external deceleration rate applied during interpolation type rapid traverse.
1 : The first axis of parameter No. 1427 is used for setting the external deceleration rate applied during interpolation type rapid traverse.

(Similarly, for external deceleration 2 and 3, the first axis of the external deceleration rate parameter for rapid traverse is used if EDR is set to 1.)

	#7	#6	#5	#4	#3	#2	#1	#0
1406							ED3	ED2

[Data type] Bit

ED2 External deceleration 2 is:

0 : Disabled.
1 : Enabled.

ED3 External deceleration 3 is:
0 : Disabled.
1 : Enabled.

1426	External deceleration rate 1 of cutting feed
1440	External deceleration rate 2 of cutting feed
1443	External deceleration rate 3 of cutting feed

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800

Set the external deceleration rate of cutting feed.

1427	External deceleration rate 1 of rapid traverse for each axis
1441	External deceleration rate 2 of rapid traverse for each axis
1444	External deceleration rate 3 of rapid traverse for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set the external deceleration rate of rapid traverse for each axis.

1434	Maximum feedrate 1 of manual handle feed for each axis
1442	Maximum feedrate 2 of manual handle feed for each axis
1445	Maximum feedrate 3 of manual handle feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set the maximum feedrate of manual handle feed for each axis.

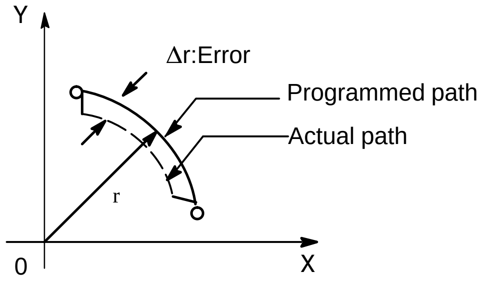
Notes

PMC axis control supports external deceleration 1 only.

7.1.10 Feedrate Clamping by Arc Radius (M series)

General

When an arc is cut at a high speed in circular interpolation, a radial error exists between the actual tool path and the programmed arc. An approximation of this error can be obtained from the following expression:



$$\Delta r = \frac{1}{2}\left(T_1^2+T_2^2\right)\frac{v^2}{r} \quad \ldots\ldots \text{(Expression 1)}$$

Δr: Maximum radial error (mm)
v: Feedrate (mm/s)
r: Arc radius (mm)
T_1: Time constant for exponential acceleration/deceleration of cutting feed (s)
T_2: Time constant of the servo motor (s)

When actual machining is performed, radius r of the arc to be machined and permissible error Δr are given. Then, maximum allowable feedrate v (mm/min) is determined from the above expression.

The function for clamping the feedrate by the arc radius automatically clamps the feedrate of arc cutting to the value set in a parameter. This function is effective when the specified feedrate may cause the radial error for an arc with a programmed radius to exceed the permissible degree of error.

When the permissible error Δr is determined, the maximum permissible speed V for the arc radius R is obtained from expression 2.

$$\Delta r = \frac{1}{2}\left(T_1^2+T_2^2\right)\frac{V^2}{R} \quad \ldots\ldots \text{(Expression 2)}$$

For the arc radius r, the maximum permissible speed v to set the permissible error to Δr is obtained from expression 1. From expressions 1 and 2, the following expression is obtained:

$$\frac{1}{2}\left(T_1^2+T_2^2\right)\frac{v^2}{r} = \frac{1}{2}\left(T_1^2+T_2^2\right)\frac{V^2}{R}$$

$$\therefore\ v = \sqrt{\frac{r}{R}}\,V \quad \ldots\ldots \text{(Expression 3)}$$

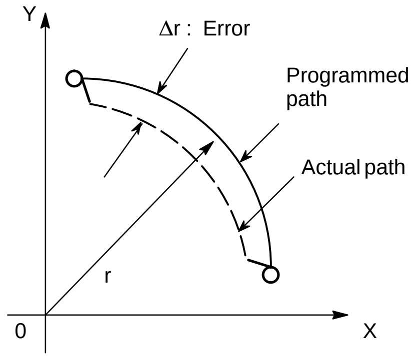
When a given arc radius R and the maximum permissible speed V for that arc radius are set as parameters, the maximum permissible speed v for an arc with a programmed radius r can be obtained from expression 3. Then, if a specified feedrate exceeds the speed v, the feedrate is automatically clamped to the speed v.

The maximum permissible speed v obtained from expression 3 decreases with the specified arc radius. To prevent the maximum permissible speed from decreasing excessively, the lower limit imposed on the maximum permissible speed v can be set in parameter 1732.

Provided the specified feedrate does not exceed the maximum permissible speed v obtained from expression 3, arc cutting is performed at the specified feedrate.

- **Cutting feed linear acceleration/deceleration**

When the cutting feed linear acceleration/deceleration function is used, an approximate error in arc cutting can be obtained from expression 4.



Δr	:	Maximum radius error (mm)
v	:	Feedrate (mm/s)
r	:	Arc radius (mm)
T_1	:	Time constant for cutting feed linear acceleration/deceleration (s)
T_2	:	Servo motor time constant (s)

$$\Delta r = \left(\frac{1}{24}T_1^{\,2} + \frac{1}{2}T_2^{\,2}\right)\frac{V^2}{r} \quad \cdots\cdots\cdots \text{(Expression 4)}$$

As can be seen from expression 4, expression 3 also holds for linear acceleration/deceleration after interpolation. Therefore, feedrate clamping by the arc radius is enabled.

- **Actual error**

Expressions 1, 2, and 4 logically denote an approximate error in the CNC. They do not denote an actual error in machining.

Let the actual error in machining be Δr all. Then, it is expressed as follows:

$$\Delta r\ \mathit{all} = \Delta r\ \mathit{NC} + \Delta r\ \mathit{machine} \quad \cdots\cdots\cdots \text{(Expression 5)}$$

Δr *machine*: Error caused by the machine

Δr *NC*: Error caused by the acceleration/deceleration time constant (T_1) and servo motor time constant (T_2)

This function controls only the first term on the right side of expression 5. In other words, this function suppresses only the error caused by the acceleration/deceleration time constant (T_1) and servo motor time constant (T_2) to a certain level. This function does not control the error caused by the machine.

Expressions 1, 2, and 4 are approximate expressions. This means that, as the arc radius becomes smaller, the approximate precision lowers. Therefore, even when the feedrate is clamped to the maximum permissible speed v obtained from expression 3, the permissible error may be exceeded.

Parameter

1730	Maximum feedrate for arc radius R

[Data type] Word

[Unit of data]

[Valid data range]

.Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 – 15000	0 – 12000
Inch machine	0.1 inch/min	8 – 6000	0 – 4800

Set a maximum feedrate for the arc radius set in parameter No. 1731.

1731	Arc radius value corresponding to a maximum feedrate

[Data type] Two–word

[Unit of data]

Unit	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter machine)	0.01	0.001	0.0001	mm
Linear axis (inch machine)	0.001	0.0001	0.00001	inch

[Valid data range] 1000 to 99999999

Set the arc radius corresponding to the maximum feedrate set in parameter No. 1730.

1732	Minimum value (RV min) for arc radius–based feedrate clamp

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 15000	0 – 12000
Inch machine	0.1 inch/min	0 – 6000	0 – 4800

The arc radius–based feedrate clamping function reduces the maximum feedrate as the arc radius decreases. When the specified maximum feedrate is not greater than RV min (minimum value for arc radius–based feedrate clamping), RV min is used as the maximum feedrate.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.19.1	Feedrate clamp by circle radius
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.19.1	Feedrate clamp by circle radius

7.1.11
Automatic Corner Deceleration

General

This function automatically controls the feedrate during corner machining according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis.

This function is enabled when G64 (machining) mode (M series) is selected and deceleration of the first of two consecutive cutting feed blocks is executed.

Feedrate control can be performed according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis. The desired method is selected by specifying the corresponding value in the CSD bit (bit 4 of parameter No. 1602).

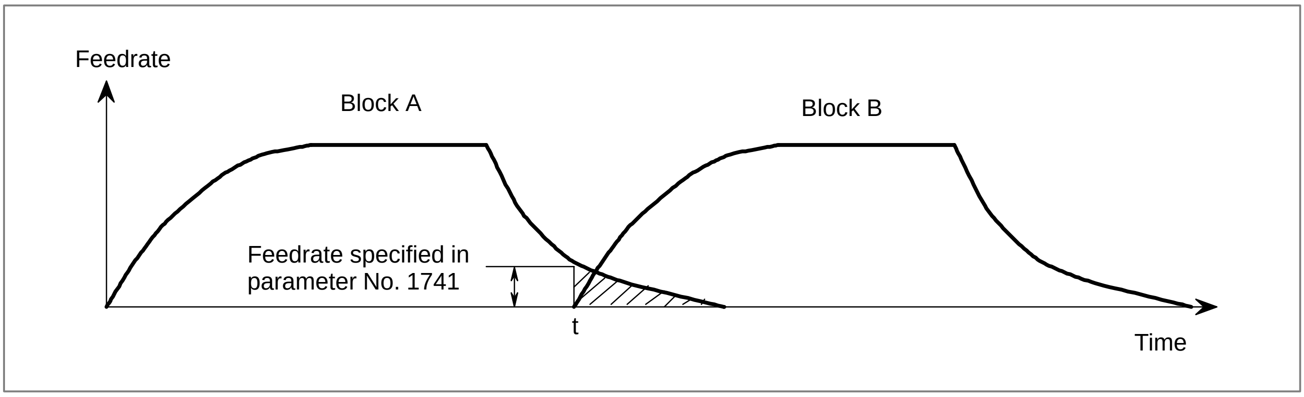
Feedrate control according to corner angle

- **Overview**

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates along the first and second axes on that plane are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

The figure shows the relationship between feedrate and time when a corner angle is smaller than the angle specified in the parameter.

At time t, some accumulated pulses remain, as indicated by the shaded part. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in the parameter.



- **Parameter**

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner deceleration function)

0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1740	Critical angle subtended by two blocks for automatic corner deceleration

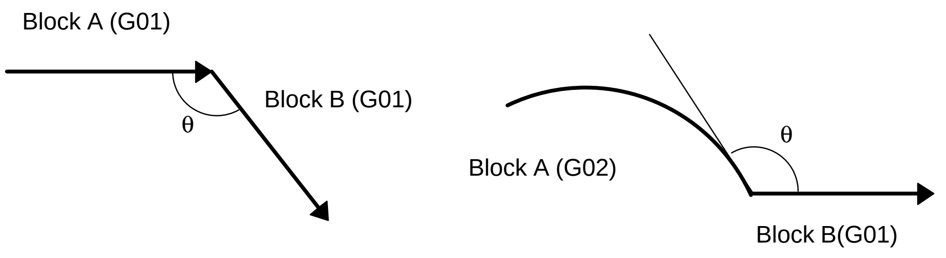
[Data type] Two–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 180000

Set a critical angle to be subtended by two blocks for corner deceleration when the angle–based automatic corner deceleration function is used.

The angle subtended by two blocks is defined as θ in the examples shown below.



Angle made by two lines

If a circular path is included, the angle between the tangent of the arc and another line is considered.

1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

Caution

CAUTION

1 The angle of the machining tool path is compared with that specified in parameter No. 1740 only for the selected plane. The actual feedrate and that specified in parameter No. 1741 are compared only for the first and second axes of the selected plane. Even if simultaneous movement is performed along three or more axes, the feedrates of only the first and second axes are compared for the selected plane.
2 The roundness of a corner is determined by the angle and feedrate specified in parameter Nos. 1740 and 1741, respectively. If a sharp corner is always required, set a feedrate of zero and an angle of 180000 (180 degrees).
3 If a G09 (exact stop) command is executed, an exact stop is performed, and Automatic Corner Deceleration is not executed.
4 This function is disabled in single block and dry run mode.

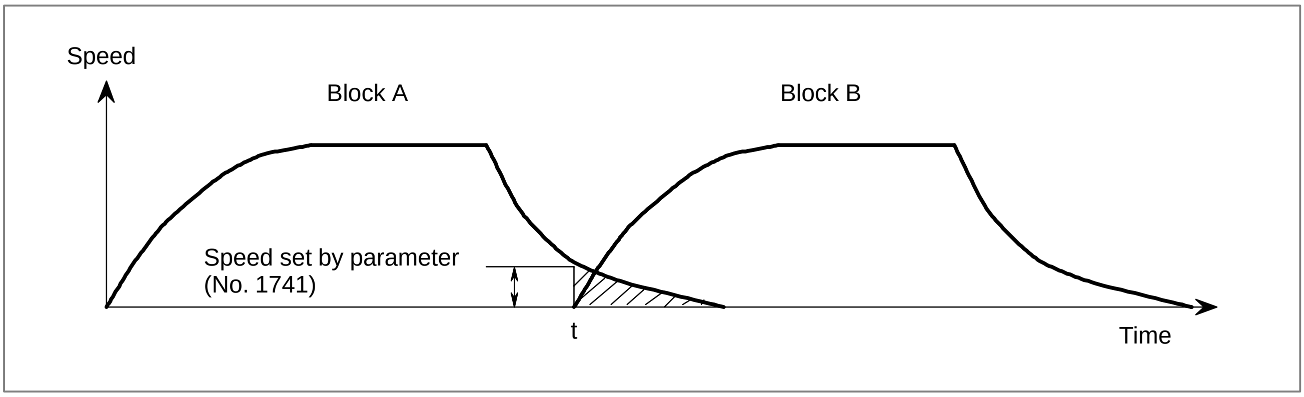
Feedrate control according to the feedrate difference for each axis

• Overview

If the difference between the programmed feedrates at the end of block A and at the beginning of block B for each axis exceeds the value specified in parameter No. 1781, and if the feedrates for all axes are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

The figure shows the relationship between the feedrate and time when the feedrate difference for each axis exceeds the value specified in parameter No. 1781.

At time t, some accumulated pulses remain, as indicated by the shaded section. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in parameter No. 1741.



- **Parameter**

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner deceleration function)

0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

1781	Allowable speed difference for the speed difference–based corner deceleration function (linear acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set speed difference for the speed difference–based automatic corner deceleration function when linear acceleration/deceleration after interpolation used.

Caution

CAUTION
1 Even during dry run or external deceleration, the feedrate difference is checked according to the F command in the program.
2 If the G09 (exact stop) command is executed, an exact stop is performed, and Automatic Corner Deceleration is not executed.
3 This function is invalid for the feed per rotation command, F1–digit feed command (M series), and rigid tapping command, as well as in single block mode.
4 If the override is changed during operation, the feedrate difference cannot be checked correctly.

7.1.12 Advanced Preview Control

General

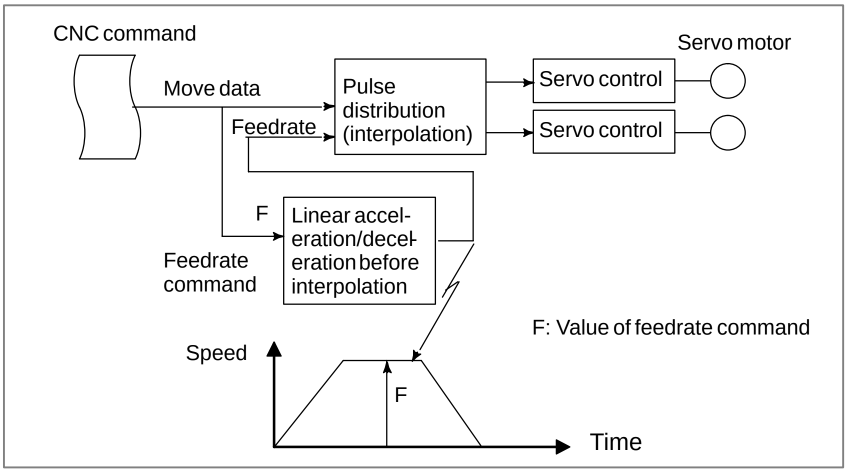
This function is designed for high–speed precise machining. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase with the feedrate can be suppressed.
The tool can then follow specified path accurately and errors in the machining profile can be reduced.
This function becomes effective when advanced preview control mode is activated by G08P1 command.

• Available functions

In advanced preview control mode, the following functions are available:
(1) Linear acceleration/deceleration before interpolation for cutting feed
(2) Automatic corner deceleration function

Linear Acceleration/ Deceleration before Cutting Feed Interpolation

A specified cutting feedrate can be linearly increased or decreased before interpolation. This function eliminates machining profile errors caused by the delay occurring in acceleration or deceleration. The time required for acceleration or deceleration by this function is significantly shorter than that by the function of exponential acceleration/deceleration.



The function of linear acceleration/deceleration before interpolation increases or decreases the feedrate specified in the tangential direction.

If the feedrate command is changed

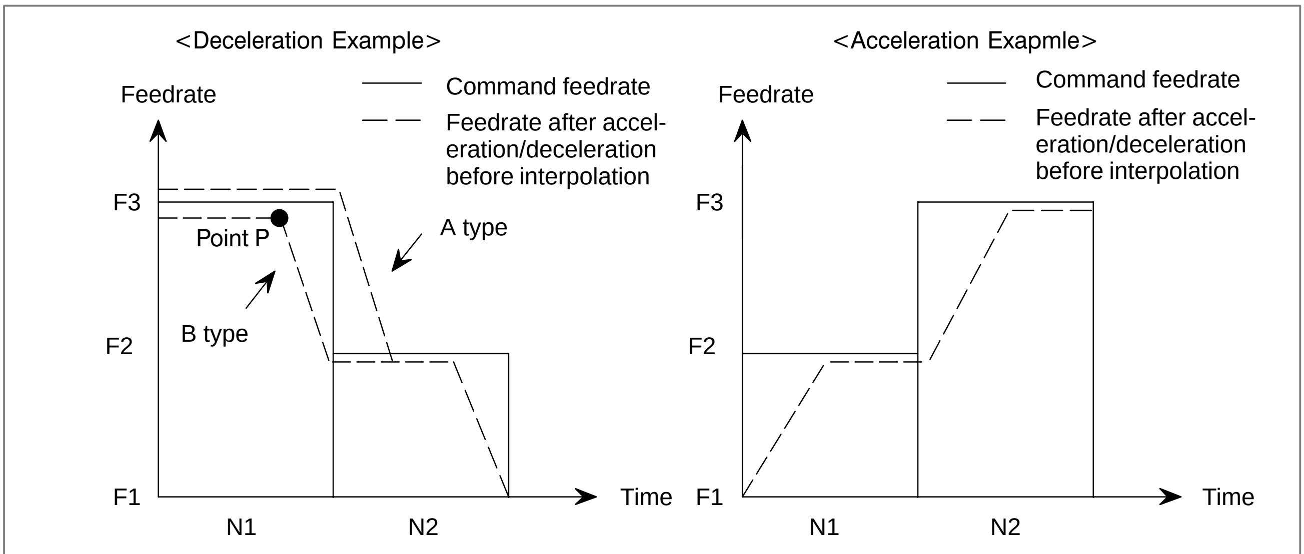
- Type A

Acceleration/deceleration is started in the block in which a new feedrate command is specified.

- Type B (Set the FWB bit (bit 0 of parameter No. 1602) to 1.)

Deceleration: Deceleration is started in a prior block such that deceleration is completed before the beginning of the block in which a new feedrate command is specified.

Acceleration: Acceleration is started in the block in which a new feedrate command is specified.



With B type, to decelerate from F3 to F2, deceleration must start with point P.

If an overtravel alarm occurs during linear acceleration/deceleration before interpolation, the movement is decelerated and stopped. As deceleration and stop are performed after the alarm occurs, the tool will overrun by an amount equal to the distance required for the deceleration. The actual overrun depends on the feedrate when the overtravel alarm occurs.

The distance can be minimized by starting deceleration in advance, such that the feedrate has fallen to the value specified in parameter No. 1784 when an overtravel alarm occurs. Because deceleration is executed such that the feedrate at the time an overtravel alarm occurs does not exceed the feedrate specified in the corresponding parameter, deceleration may be completed earlier. After deceleration is completed, the feedrate specified in the parameter is maintained.

Deceleration is performed when the following condition is satisfied:

Distance to stored stroke limit 1 for each axis	<	Distance needed to reduce the current feedrate (tangential feedrate) to that specified in parameter No. 1784

The overrun is calculated as follows:

$$\text{Overrun} \leqq \frac{[\text{FIX}\ (\frac{F_{OT}}{F} \times \frac{T}{8})\ +\ 1.5]^2 \times \frac{F}{T}}{1875}$$

F : Maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1770)

T : Time needed to attain the maximum cutting feedrate in linear acceleration/deceleration before interpolation

For a linear axis (command not involving a rotation axis):
Parameter No. 1771
For a rotation axis (command involving a rotation axis):
Parameter No. 1786

For a linear axis (command not involving a rotation axis):
Parameter No. 1772
For a rotation axis (command involving a rotation axis):
Parameter No. 1787

F_{OT}: Feedrate at the time an overtravel alarm occurs during linear acceleration/deceleration before interpolation (parameter No. 1784)

FIX: Any fractional part is truncated.

NOTE
A time required to reach the maximum machining feedrate based on acceleration/deceleration before interpolation in the advanced preview control mode can be set separately for a linear axis and rotation axis. Note, however, that the tool decelerates and stops between a block specifying a command not involving a rotation axis and a block specifying a command involving a rotation axis.

Note

NOTE

1 If a block without a move command is found during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in the previous block.
2 If a one–shot G code is specified during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in the previous block.
3 If an M, S, or T code is specified in a block containing a move command during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in that block.
4 During acceleration/deceleration before interpolation, a G31 block (skip function) is not subjected to acceleration/deceleration.
5 If the machine lock signal (MLK1 to MLK8) for an axis is set on or off during acceleration/deceleration before interpolation, the axis for which machine lock is performed is not subjected to acceleration/deceleration.
6 During acceleration/deceleration before interpolation, automatic corner override is enabled only when the internal circular cutting feedrate is changed.
7 During acceleration/deceleration before interpolation, acceleration/deceleration after interpolation can be executed. So that acceleration/deceleration is executed only before interpolation, set the time constant for acceleration/deceleration after interpolation to zero.
8 In acceleration/deceleration before interpolation of type B, deceleration is started if preprocessing for the next block has not been completed before the remaining distance of the current block becomes less than that needed to decelerate and stop the movement.
9 If an F1–digit command is executed in the inch input system, avoid specifying a command for simultaneous movement on two axes, including a rotation axis during acceleration/deceleration before interpolation (M series).
10 The error detect signal (SMZ) is invalid during acceleration/deceleration before interpolation (T series).
11 If an overtravel alarm occurs during acceleration/deceleration before interpolation, the movement is decelerated and stopped. As deceleration and stop are performed after the alarm occurs, the tool will overrun by the distance required for the deceleration.
12 When feed per revolution is specified during acceleration/deceleration before interpolation, the spindle speed can be set to up to 30000 min^{-1} in theory.
13 If switching between feed per revolution and feed per minute is performed at the interface of two blocks during acceleration/deceleration before interpolation, the movement is decelerated and temporarily stopped in the previous block.

Automatic Corner Deceleration

This function automatically controls the feedrate during corner machining according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis.

This function is enabled when deceleration of the first of two consecutive cutting feed blocks is executed.

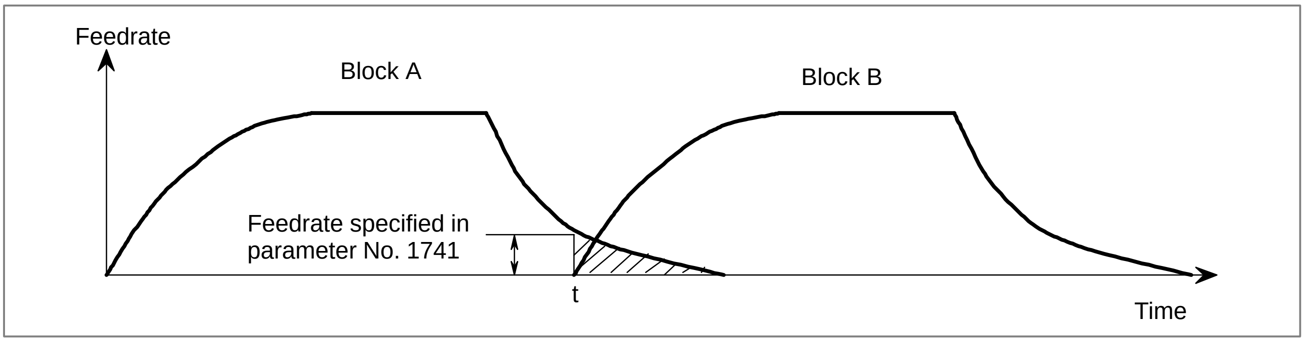
Feedrate control can be performed according to the angle of a corner made by machining blocks or according to the feedrate difference for each axis. The desired method is selected by specifying the corresponding value in the CSD bit (bit 4 of parameter No. 1602).

• Feedrate control according to corner angle

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates along the first and second axes on that plane are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

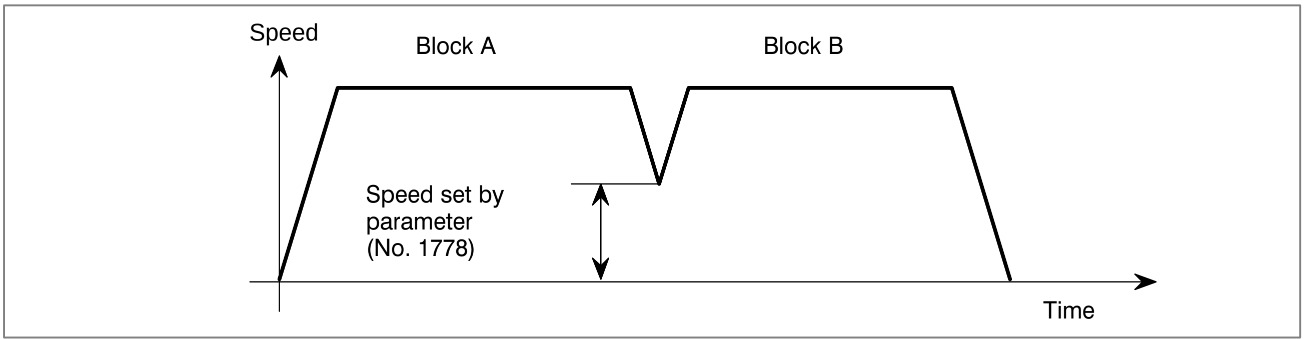
The figure shows the relationship between feedrate and time when a corner angle is smaller than the angle specified in the parameter.

At time t, some accumulated pulses remain, as indicated by the shaded part. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in the parameter.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates programmed for blocks A and B are higher than the value set in parameter No. 1778, the feedrate is reduced to the value specified in the parameter in block A. In block B, the feedrate is increased to the programmed feedrate. The rate of acceleration depends on the parameter for linear acceleration/deceleration before interpolation for cutting feed.



Caution

CAUTION

1 The angle of the machining tool path is compared with that specified in parameter No. 1779 only for the selected plane. The actual feedrate and that specified in parameter No. 1741 are compared only for the first and second axes of the selected plane. Even if simultaneous movement is performed along three or more axes, the feedrates of only the first and second axes are compared for the selected plane.

2 The roundness of a corner is determined by the angle and feedrate specified in parameter Nos. 1740 and 1741, respectively. If a sharp corner is always required, set a feedrate of zero and an angle of 180000 (180 degrees).

3 If a G09 (exact stop) command is executed, an exact stop is performed, and Automatic Corner Deceleration is not executed.

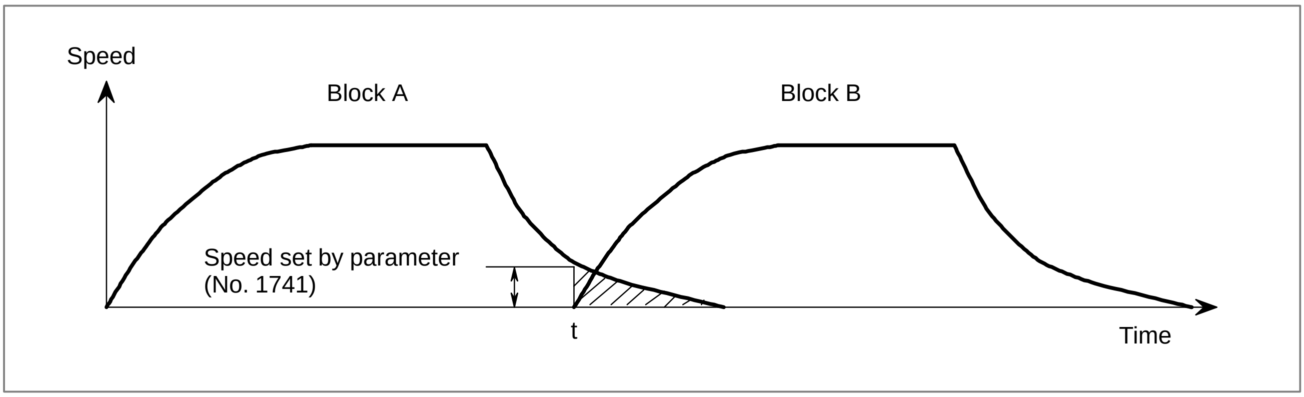
4 This function is disabled in single block and dry run mode.

- **Feedrate control according to the feedrate difference for each axis**

If the difference between the programmed feedrates at the end of block A and at the beginning of block B for each axis exceeds the value specified in parameter No. 1781, and if the feedrates for all axes are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

The figure shows the relationship between the feedrate and time when the feedrate difference for each axis exceeds the value specified in parameter No. 1781.

At time t, some accumulated pulses remain, as indicated by the shaded section. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in parameter No. 1741.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

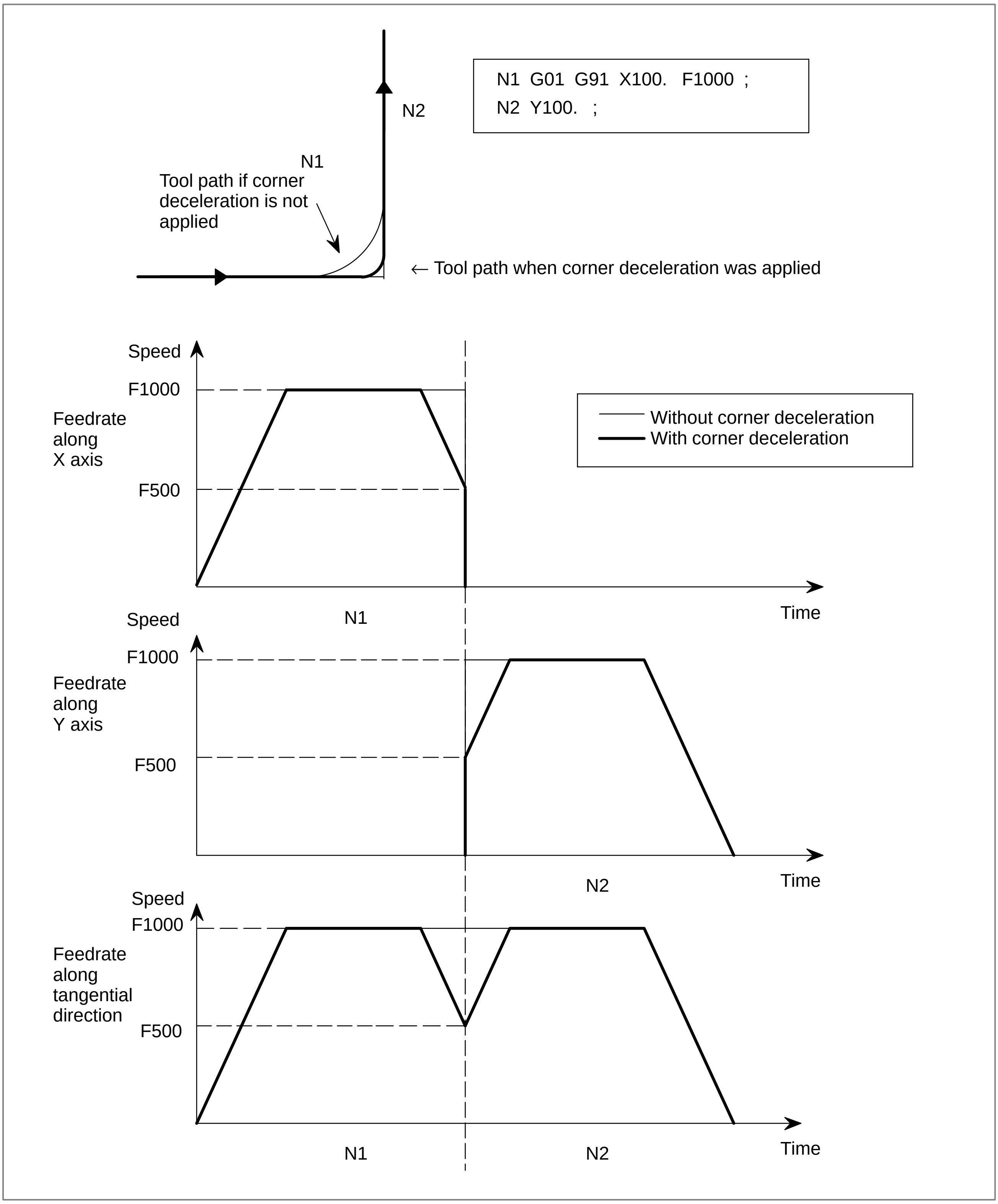
If the difference between the feedrates of blocks A and B for each axis exceeds the value specified in parameter No. 1780, the feedrate at the corner is calculated from the difference for each axis, as shown below. The feedrate is reduced to the calculated value in block A.

The feedrate change for each axis (Vc[X], Vc[Y], ...), caused by the movement at programmed feedrate F, is compared with Vmax specified in parameter No. 1780. If an feedrate change exceeding Vmax is detected, the target feedrate after deceleration Fc is calculated, using maximum comparison value Rmax.

$$R = \frac{Vc}{Vmax}$$

$$Fc = \frac{F}{Rmax}$$

If, for example, the direction of movement is changed from the X–axis to the Y–axis, that is through 90 degrees, and if the programmed feedrate is 1000 mm/min and the permissible feedrate difference specified in parameter No. 1780 is 500 mm/min, the deceleration shown below is performed:



Different permissible feedrate differences can be specified for different axes. If a value is specified in parameter No. 1783, the permissible feedrate difference for each axis becomes valid. Deceleration at a corner is calculated for the axis for which the permissible feedrate difference is exceeded with the highest ratio of actual feedrate difference to permissible feedrate difference.

Caution

CAUTION

1 Even during dry run or external deceleration, the feedrate difference is checked according to the F command in the program.
2 If the G09 (exact stop) command is executed, an exact stop is performed, and Automatic Corner Deceleration is not executed.
3 This function is invalid for the feed per rotation command, F1–digit feed command (M series), and rigid tapping command, as well as in single block mode.
4 If the override is changed during operation, the feedrate difference cannot be checked correctly.

Signal

advanced preview control mode signal G08MD <F066#0>

[Classification] Output signal

[Function] Informs that the control is in the advanced preview control mode.

[Output condition] The signal is “1” in the following case:
·In the advanced preview control mode
The signal is “0” in the following case:
·It is not the advanced preview control mode

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F066								G08MD

Parameter

1431	Maximum cutting feedrate for all axes in the advanced preview control mode

[Data type] 2–words

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min	0 to 96000	0 to 48000
Rotation axis	1 deg/min	0 to 240000	0 to 100000

Specify the maximum cutting feedrate for all axes in the advanced preview control mode.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
1 To specify the maximum cutting feedrate for each axis, use parameter No.1432 instead.
2 In a mode other than the look–ahead mode, the maximum cutting feedrate specified in parameter No.1422 or No.1430 is applied and the feedrate is clamped at the maximum feedrate.

1432	Maximum cutting feedrate for each axis in the AI advanced preview control/ AI contour control mode or advanced preview control mode

[Data type] 2–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min	0 to 96000	0 to 48000
Rotation axis	1 deg/min	0 to 240000	0 to 100000

Specify the maximum cutting feedrate for each axis in the AI advanced preview control/AI contour control mode or advanced preview control mode.

A feedrate for each axis is clamped during cutting feed so that it does not exceed the maximum cutting feedrate specified for each axis.

NOTE
1 This parameter is effective only in linear and circular interpolation. In polar coordinate and cylindrical interpolation, the maximum feedrate for all axes specified in parameter No.1431 is effective.
2 If a setting for each axis is 0, the maximum feedrate specified in parameter No.1431 is applied to all axes and the feedrate is clamped at the maximum feedrate.
3 In a mode other than the AI advanced preview control / AI contour control mode or advanced preview mode, the maximum cutting feedrate specified in parameter No.1422 or No.1430 is applied and the feedrate is clamped at the maximum feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
1601						OVB		

[Data type] Bit

OVB Block overlap in cutting feed
0: Blocks are not overlapped in cutting feed.
1: Blocks are overlapped in cutting feed.

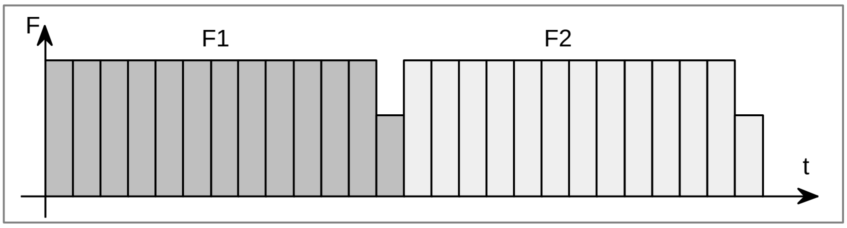
Block overlap outputs the pulses remaining at the end of pulse distribution in a block together with distribution pulses in the next block. This eliminates changes in feedrates between blocks.

Block overlap is enabled when blocks containing G01, G02, or G03 are consecutively specified in G64 mode. If minute blocks, however, are specified consecutively, overlap may not be performed.

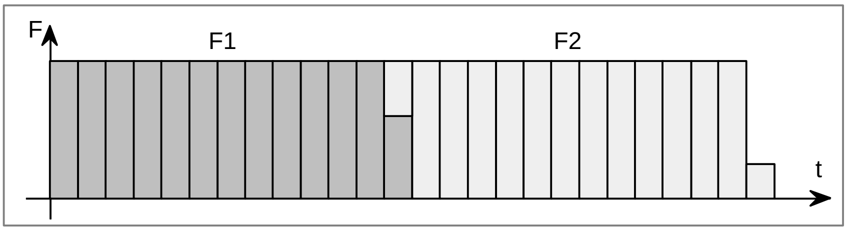
The following pulses in block F2 are added to the pulses remaining at the end of pulse distribution in block F1.

$$\textbf{(Number of pulses to be added)} = \textbf{F2} \times \frac{\textbf{(Number of pulses required at the end of block F1)}}{\textbf{F1}}$$

When F1 = F2



When block overlap is disabled



When block overlap is enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2		CSD	BS2			FWB

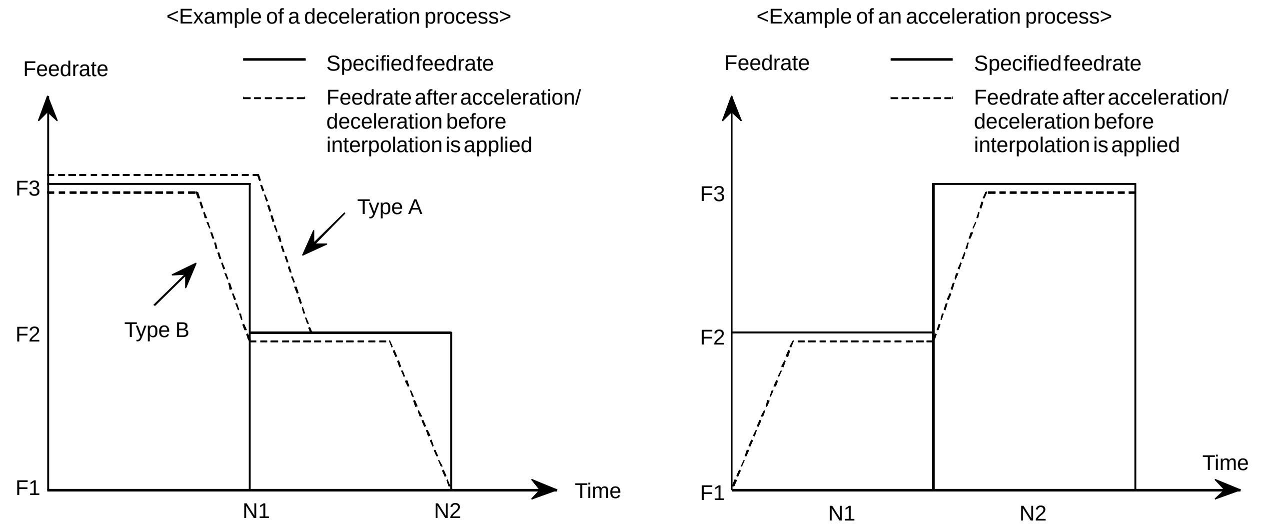
[Data type] Bit

FWB Cutting feed acceleration/deceleration before interpolation

0 : Type A of acceleration/deceleration before interpolation is used.
1 : Type B of acceleration/deceleration before interpolation is used.

Type A: When a feedrate is to be changed by a command, acceleration/deceleration starts after the program enters the block in which the command is specified.

Type B: When a feedrate is to be changed by a command, deceleration starts and terminates at the block before the block in which the command is specified.
When a feedrate is to be changed by a command, acceleration starts after the program enters the block in which the command is specified.



To change the feedrate from F3 to F2, it is necessary to start reducing the feedrate at point 1.

BS2 The type of acceleration/deceleration after interpolation in cutting feed in advanced preview control mode is:

0 : Specified by bit 6 (LS2) of parameter No. 1602.
1 : Bell–shaped acceleration/deceleration.

BS2	LS2	Acceleration/deceleration
0	0	Exponential acceleration/deceleration after interpolation
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation. (The option for bell–shaped acceleration/deceleration after interpolation for cutting feed is required.)

CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

LS2 Type of acceleration/deceleration after interpolation in cutting feed in advanced preview control, AI advanced preview control, or AI contour control mode

0 : Exponential acceleration/deceleration is applied (advanced preview control), or no acceleration/deceleration is applied (AI advanced preview control and AI contour control).
1 : Linear acceleration/deceleration is applied.

1762	Exponential acceleration/deceleration time constant for cutting feed in the advanced preview control mode

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set an exponential acceleration/deceleration time constant for cutting feed in the advanced preview control mode.

1763	Minimum speed in exponential acceleration/deceleration for cutting feed in the advanced preview control mode

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set minimum speed (FL) in exponential acceleration/deceleration for cutting feed in the advanced preview control mode.

1768	Time constant of linear acceleration/deceleration or bell–shaped acceleration/deceleration after interpolation in cutting feed in advanced preview control, AI advanced preview control, or AI contour control mode

[Data type] Word

[Unit of data] msec

[Valid data range]

Advanced preview control, AI advanced preview control	0, 8 to 512
AI contour control	0, 4 to 256

Set the time constant to be used for linear or bell–shaped acceleration/deceleration after interpolation in cutting feed in advanced preview control, AI advanced preview control, or AI contour control mode.

NOTE
For bell–shaped acceleration/deceleration, the function of bell–shaped acceleration/deceleration after cutting feed interpolation is required.

1769	Time constant of linear acceleration/deceleration or bell–shaped acceleration/deceleration after interpolation in cutting feed for each axis in advanced preview control, AI advanced preview control, or AI contour control mode

[Data type] Word axis

[Unit of data] msec

[Valid data range]

Advanced preview control, AI advanced prevoew control	0, 8 to 512
AI contour control	0, 4 to 256

Set the time constant to be used for linear or bell–shaped acceleration/deceleration after interpolation in cutting feed in advanced preview control, AI advanced preview control, or AI contour control mode for each axis. Which acceleration/deceleration type, the linear or bell–shaped type, is to be used is specified by bit 3 (BS2) and bit 6 (LS2) of parameter No. 1602.

NOTE

1 If 0 is set in parameter No. 1769 for all axes, the value set in parameter No. 1768 is used. For other than special purposes, set a time constant in parameter No. 1768, which is common to all axes.

2 If a different time constant is set in parameter No. 1769, a correct straight line or arc shape cannot be obtained.

1770	Parameter 1 for setting the acceleration rate of linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode (maximum machining speed during linear acceleration/deceleration before interpolation)

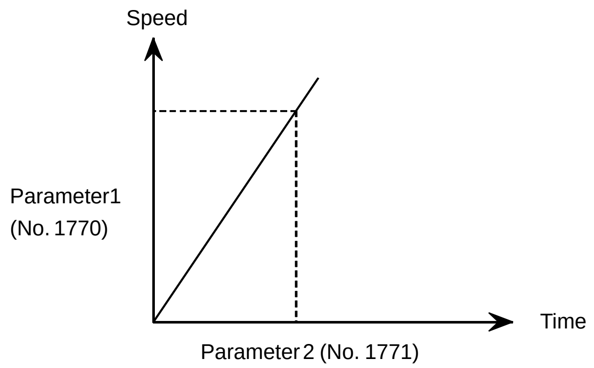
[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 48000

This parameter is used to set the acceleration rate of linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode. In this parameter, set the maximum machining speed during linear acceleration/deceleration before interpolation. Set the time used to reach the maximum machining speed in parameter No.1771.



NOTE

When 0 is set in parameter No.1770 or parameter No.1771, linear acceleration/deceleration before interpolation is disabled.

1771	Parameter 2 for setting the acceleration rate of linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode (time until the maximum machining speed is reached during linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 4000

This parameter is used to set the acceleration rate of linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode. In this parameter, set the maximum machining speed during linear acceleration/deceleration before interpolation. In this parameter, set the time (time constant) used to reach the speed set in parameter No.1770.

NOTE

1 When 0 is set in parameter No.1770 or parameter No.1771, linear acceleration/deceleration before interpolation is disabled.
2 In parameter Nos. 1770 and 1771, set values that satisfy the following:
Parameter No.1770/Parameter No.1771 $\geq$ 5
3 If 0 is set in parameter No. 1770 or 1771 in AI advanced preview control or AI contour control, P/S alarm 5157 is issued.

1777	Minimum speed for the automatic corner deceleration function (for advanced preview control, AI advanced preview control, or AI contour control)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800

Set a speed at which the number of buffered pulses in deceleration is assumed to be 0 when linear acceleration/deceleration before interpolation is used.

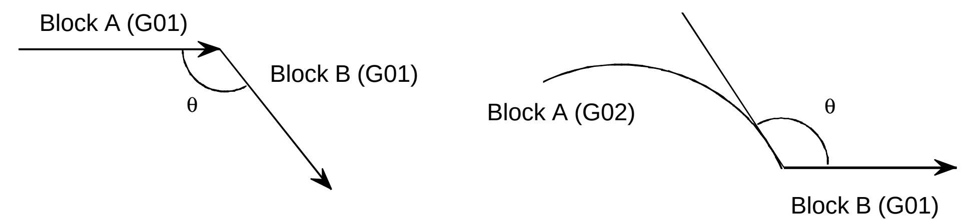
1779	Critical angle subtended by two blocks for automatic corner deceleration (for advanced preview control, AI advanced preview control, or AI contour control)

[Data type] 2–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 180000

Set a critical angle to be subtended by two blocks for corner deceleration when the angle–based automatic corner deceleration function is used. The angle subtended by two blocks is defined as q in the examples shown below.



Angle subtended by two straight lines Angle subtended by an arc and its tangent

1780	Allowable speed difference for the speed difference–based corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800

Set the speed difference for the speed difference–based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

1783	Allowable speed difference for the speed difference based corner deceleration function (linear acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

A separate allowable feedrate difference can be set for each axis. The allowable feedrate difference is set for each axis with this parameter. Among the axes that exceed the specified allowable feedrate difference, the axis with the greatest ratio of the actual feedrate difference to the allowable feedrate difference is used as the reference to calculate the reduced feedrate at the corner.

1784	Speed when overtravel alarm has generated during acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Deceleration is started beforehand to reach the feedrate set in the parameter when an overtravel alarm is issued (when a limit is reached) during linear acceleration/deceleration before interpolation. By using this parameter, the overrun distance that occurs when an overtravel alarm is output can be reduced.

This setting can be applied also to rapid traverse blocks by setting bit 0 (OTR) of parameter No.7057. (M series)

NOTE
1 When 0 is set in this parameter, the control described above is not exercised.
2 Use type–B linear acceleration/deceleration before interpolation (by setting bit 0 (FWB) of parameter No.1602 to 1).
3 The control described above is applicable only to stored stroke check 1.
4 The control described above is performed for the axes specified in the current block and next block. By setting bit 5 (ODA) of parameter No. 7055, the control can be performed just for the axis specified in the current block.

1786	Time (time constant) until the maximum machining speed is reached during linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode (for rotation axes)

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 4000

This parameter is used to set the acceleration rate (for rotation axes) of linear acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control mode.
This parameter sets the time (time constant) required to reach the speed set in parameter No. 1770. The acceleration rate set by this parameter applies to commands containing rotation axes. (The acceleration rate set by parameter No. 1771 applies to commands not containing rotation axes.)

NOTE
If 0 is set in this parameter, the acceleration rate set in parameter No. 1771 applies also to commands containing rotation axes.

	#7	#6	#5	#4	#3	#2	#1	#0
6901					PSF			

[Data type] Bit

PSF In advanced preview control mode, AI advanced preview control mode, or AI contour control mode, position switches are:
0 : Not used.
1 : Used.

NOTE
The position switch signals are output considering acceleration/deceleration after interpolation and servo delay. Acceleration/deceleration after interpolation and servo delay are considered even for position switch signal output in a mode other than advanced preview control mode, AI advanced preview control mode, and AI contour control mode. When this parameter is set to 1, however, signals are output from the position switches at different times from the specified ones.

Parameters for advanced preview control mode and normal mode

· **Parameters for the cutting feed acceleration/deceleration before interpolation**

Parameter description	Parameter No.	
	Normal mode	Advanced preview control mode
Acceleration/decelerationtype (A type/B type)	FWB (1602#0)	FWB (1602#0)
Acceleration (Parameter 1)	1630	1770
Acceleration (Parameter 2)	1631	1771
Speed when overtravel alarm has generated	1784	1784

· **Parameters for automatic corner deceleration**

Parameter description	Parameter No.	
	Normal mode	Advanced preview control mode
Automatic corner deceleration according to the corner angle or the speed difference	CSD (1602#4)	CSD (1602#4)
Minimum speed (according to the corner angle)	1778	1777
Critical angle (according to the corner angle)	1740	1779
Allowable speed difference for all axes (according to speed difference)	1780	1780
Allowable speed difference for each axis (according to speed difference)	1783	1783

Alarm and message

Number	Message	Description
109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.

Note

NOTE

The functions usable in the advanced preview control mode are listed below. When using an function other than those listed below, turn off the advanced preview control mode before using the function, and turn on the advanced preview control mode upon completion of using the function.

- PMC–based axis control (usable in the advanced preview control mode by setting bits 4 (G8R) and 3 (G8C) of parameter No. 8004)
- Single direction positioning (M series)
- Polar coordinate command (M series)
- Helical cutting
- Rigid tapping (usable in the advanced preview control mode by setting bit 5 (G8S) of parameter No. 1602. The serial spindle parameter also needs to be set.)
- Program restart
- External deceleration
- Simple synchronization control
- Sequence number check stop
- Position switch (usable in the advanced preview control mode by setting bit 3 (PSF) of parameter No. 6901)
- Cs contour control (usable in the advanced preview control mode by setting bit 5 (G8S) of parameter No. 1602. The serial spindle parameter also needs to be set.)
- Constant surface speed control
- Spindle speed fluctuation detection
- Spindle synchronization control
- Custom macro B
- Optional–angle chamfering/corner rounding (M series)
- Direct drawing dimension input (T series)
- Inch/metric switching
- Programmable mirror image (M series)
- Mirror image for double turret (T series)
- Canned cycle (M series)
- Multiple repetitive canned cycle (T series)
- Multiple repetitive canned cycle 2 (T series)
- Hole machining canned cycle (T series)
- Automatic corner override (valid only for changing the inside circular cutting feedrate)
- Scaling (M series)
- Coordinate system rotation
- Workpiece coordinate system
- Workpiece coordinate system preset
- Cutter compensation C (M series)
- Tool–nose radius compensation (T series)
- Corner arc
- Tool offset (M series)
- Y–axis offset (T series)
- Offset measurement value direct input B (T series)

NOTE

- Tool life management
- Tool length measurement (M series)
- Graphic display
- Dynamic graphic display (M series)
- Feed per revolution

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.19.2	Advanced preview control (G08)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.18.1	Advanced preview control (G08)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.19.2	Advanced preview control (G08)

7.1.13 AI Advanced Preview Control Function/AI Contour Control Function (M Series)

Overview

The AI advanced preview control/AI contour control function is provided for high–speed, high–precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.
Look–ahead acceleration/deceleration before interpolation is enabled for up to 20 blocks (0*i*) or 12 blocks (0*i* Mate) in AI advanced preview control or for up to 40 blocks in AI contour control. This enables execution of smooth acceleration/ deceleration extending over multiple blocks and higher machining.

Explanation

This function is enabled by setting the AI advanced preview control or AI contour control mode.

- **Format**

G05.1 Q _ ;

Q1 : AI advanced preview control/AI contour control mode on
Q0 : AI advanced preview control/AI contour control mode off

NOTE
1 Always specify G05.1 in an independent block.
2 The AI advanced preview control/AI contour control mode is also canceled by a reset.
3 When the AI contour control option is installed, the AI contour control mode is enabled.

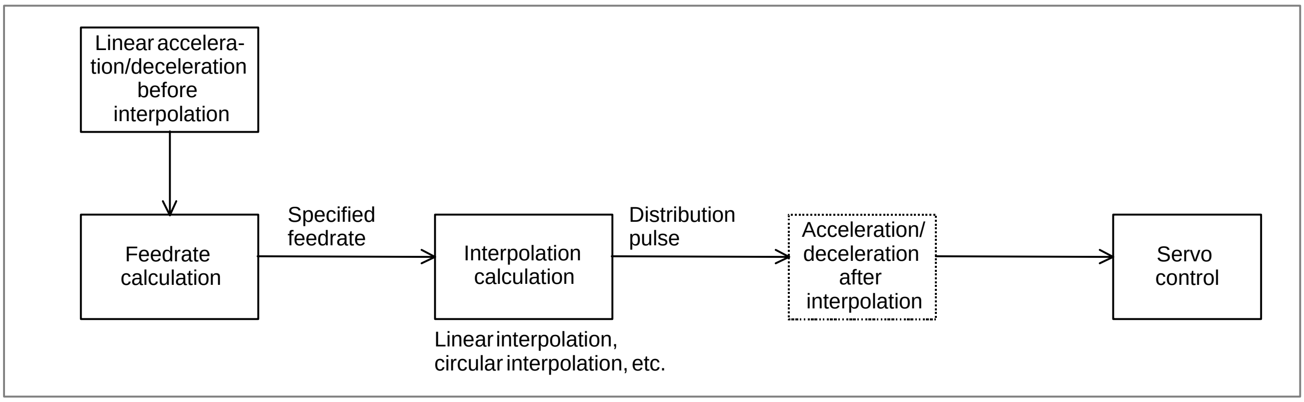
- **Functions valid in the AI advanced preview control/AI contour control mode**

The functions listed below are valid in the AI advanced preview control/AI contour control mode:

· Look–ahead linear acceleration/deceleration before interpolation
· Look–ahead bell–shaped acceleration/deceleration before interpolation (This function can not be used in AI advanced preview control) (The option of look–ahead bell–shaped acceleration/deceleration before interpolation is required.)
· Automatic corner deceleration
· Feedrate clamping by acceleration
· Feedrate clamping by arc radius
· Block overlap
· Advanced preview feed forward

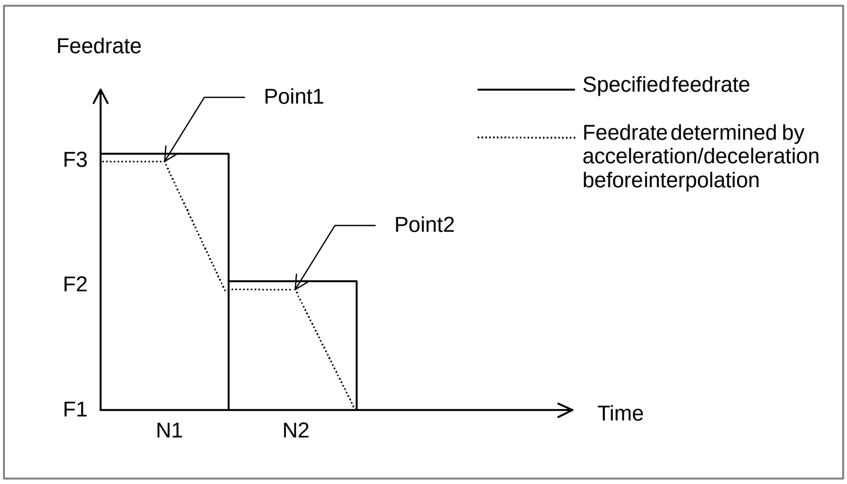
- **Look–ahead linear acceleration/deceleration before interpolation**

For a cutting feed command in the feed per minute mode, linear acceleration/deceleration can be applied before interpolation, that is, for the specified feedrate by reading up to 20 blocks (0*i*) or 12 blocks (0*i* Mate) (in the AI advanced preview control mode) or 40 blocks (in the AI contour control mode) in advance. With acceleration/deceleration after interpolation, the interpolated data is changed because acceleration/deceleration is applied to the data. With acceleration/deceleration before interpolation, the interpolated data cannot be changed by acceleration/deceleration because acceleration/deceleration is applied to the feedrate data before interpolation. For this reason, the interpolated data can always be applied to the specified straight line or curve to eliminate machining profile errors caused by acceleration/deceleration delays.



(Example of deceleration)

Deceleration is started in a prior block so that the feedrate specified for the target block is reached at the execution.

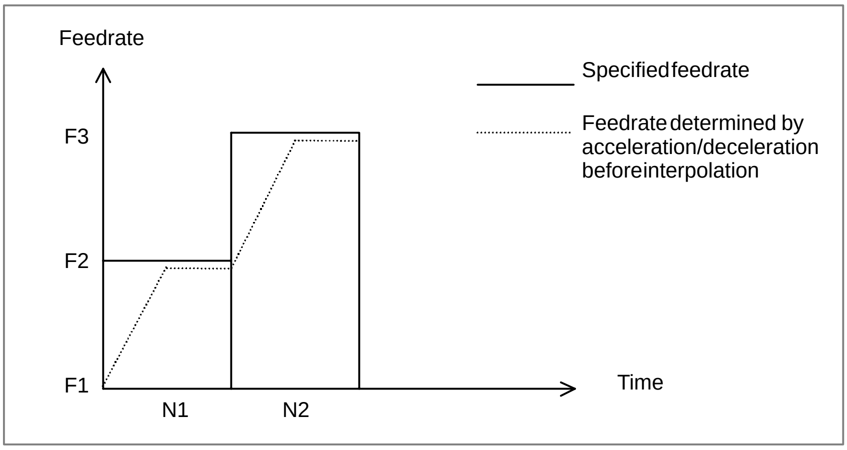


To decelerate from feedrate F3 to F2, deceleration must start with point 1. To decelerate from feedrate F2 to F1, deceleration must start with point 2.

Multiple blocks can be read in advance to perform deceleration extending over several blocks.

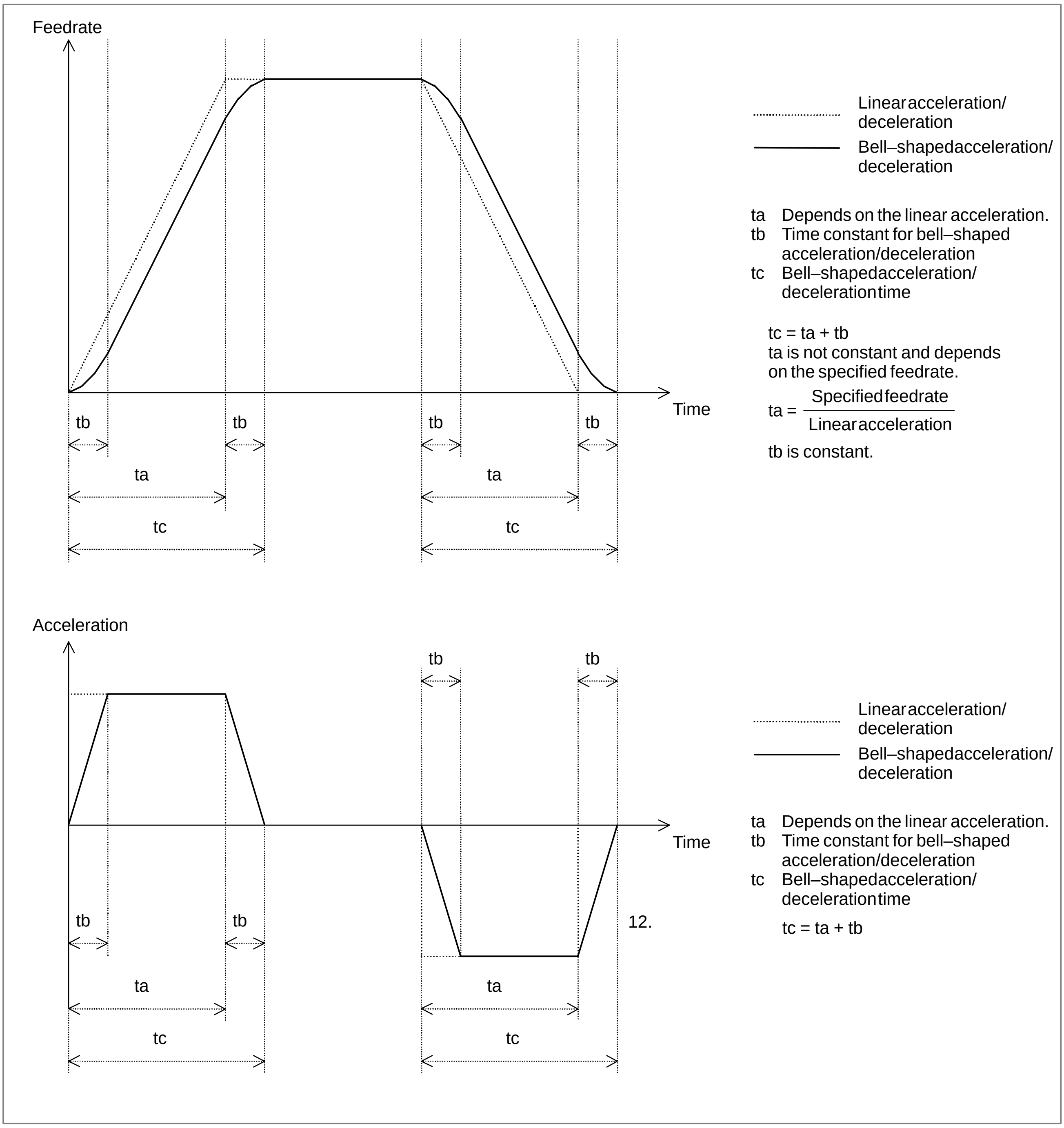
(Example of acceleration)

Acceleration is performed so that the feedrate specified for the target block is reached at the execution.



- **Look–ahead bell–shaped acceleration/deceleration before interpolation**

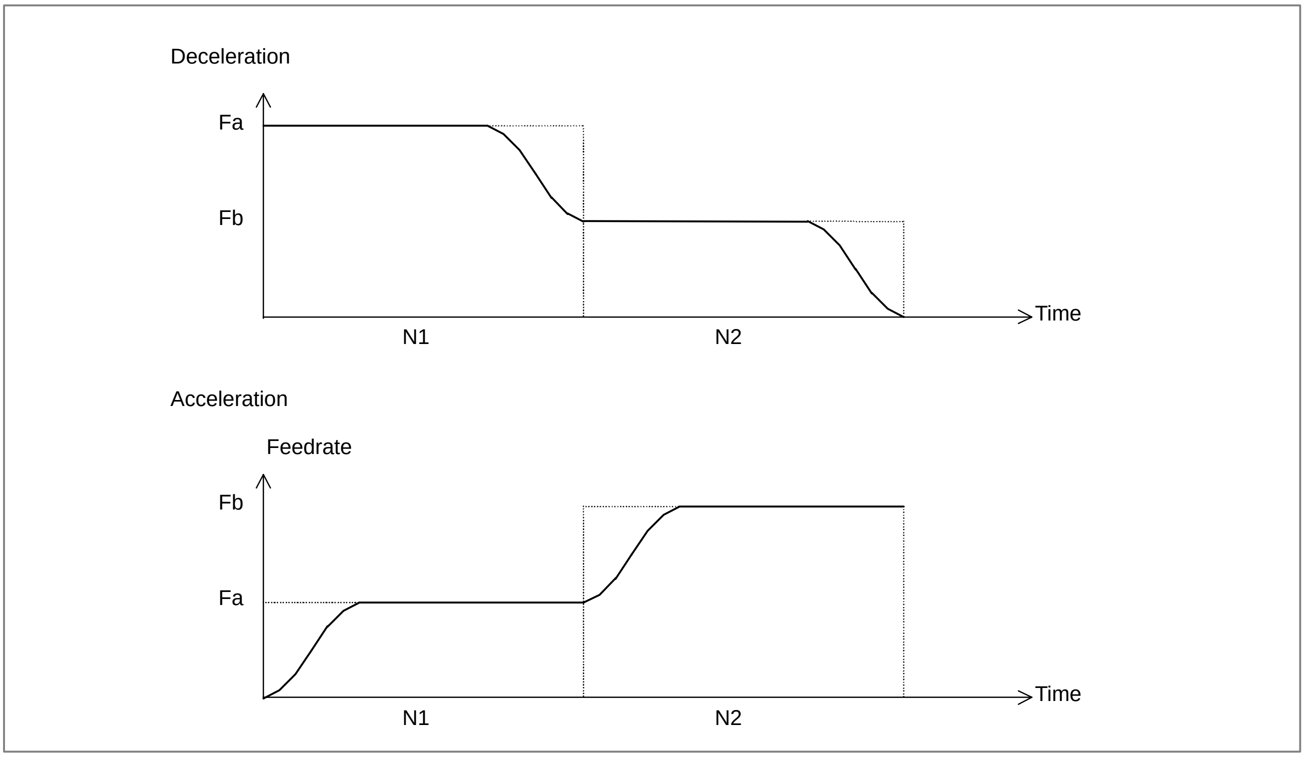
Linear acceleration/deceleration before interpolation for cutting feed in the AI contour control mode can be changed to bell–shaped acceleration/deceleration before interpolation. With bell–shaped acceleration/deceleration before interpolation, smooth acceleration/deceleration can be applied to the cutting feedrate to reduce the shock on the machine by fluctuations in acceleration that are involved in changes in the cutting feedrate.
To use this function, the option of look–ahead bell–shaped acceleration/deceleration before interpolation is required.



When the feedrate is changed, deceleration and acceleration are performed as follows:

For deceleration: Bell–shaped deceleration is started in the preceding block so that deceleration terminates by the beginning of the block in which the feedrate is changed.

For acceleration: Bell–shaped acceleration is started after the beginning of the block in which the feedrate is changed.



- **Automatic corner deceleration**

Between contiguous two blocks, the feedrate difference for an axis may exceed the setting (parameter No. 1783). In this case, the feedrate at the corner is calculated as follows based on the axis for which the ratio of the actual feedrate difference to the allowable feedrate difference is the largest. Deceleration is performed so that the feedrate is reached at the interface of the blocks.

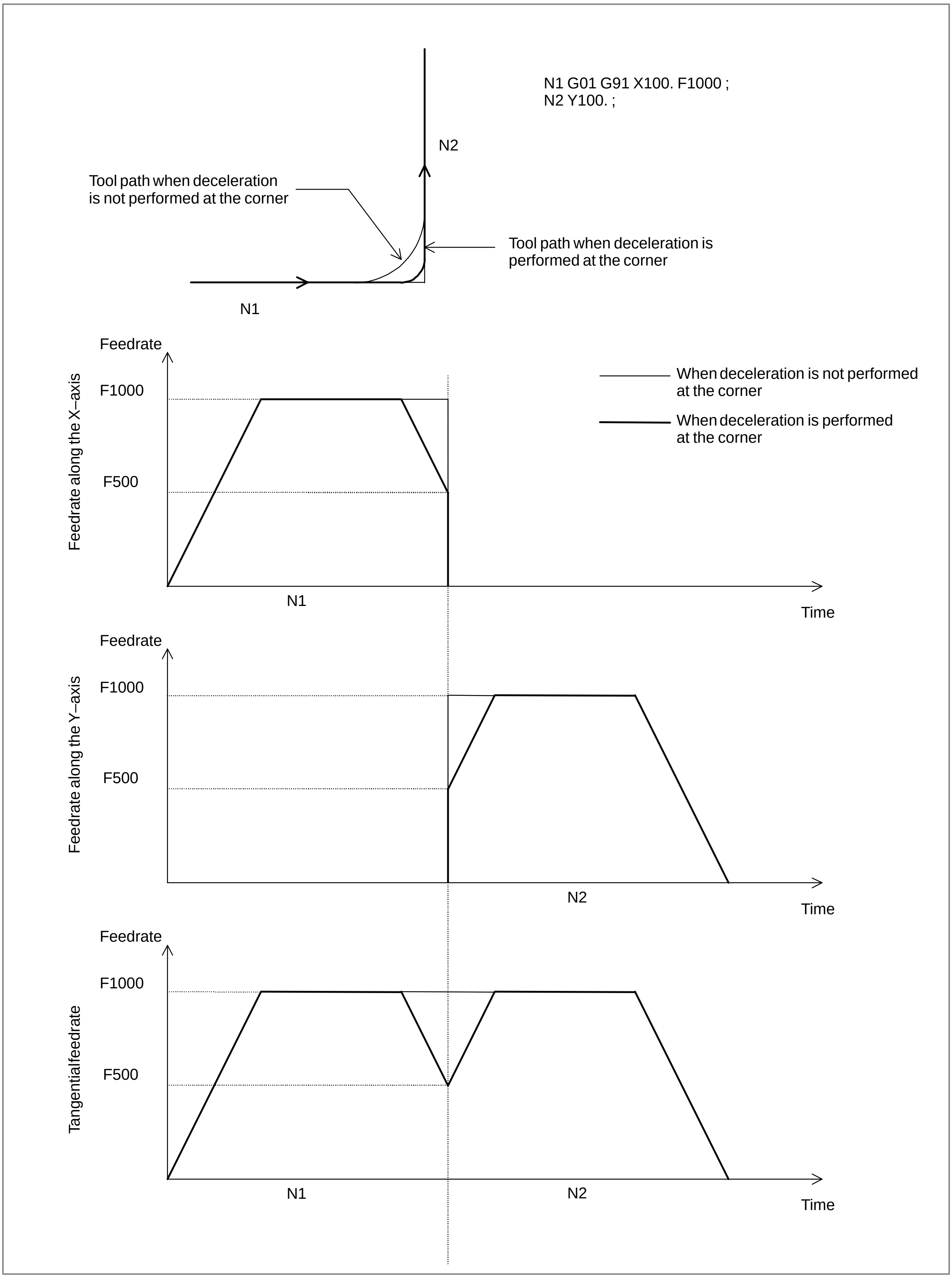
The change in the feedrate for each axis (Vx, Vy, ...) during movement at the specified feedrate F is compared with the setting of parameter No. 1783 (Vprm–x, Vprm–y, ...). If the change in the feedrate for any axis exceeds the setting of the parameter, the following expression is used:

$$Rmax = \max\left[\frac{Vx}{Vprm - x}, \frac{Vy}{Vprm - y}, , ,\right]$$

The required feedrate (Fc) is obtained as follows and deceleration is performed at the corner:

$$Fc = F \times \frac{1}{Rmax}$$

For example, assume that the tool move direction changes by 90 degrees from movement along the X–axis to that along the Y–axis. Also assume that the specified feedrate is 1000 mm/min and the allowable feedrate difference (parameter No. 1783) is 500 mm/min. Deceleration is performed as shown in the figure below:


N1 G01 G91 X100. F1000 ;
N2 Y100. ;
N2
Tool path when deceleration
is not performed at the corner
Tool path when deceleration is
performed at the corner
N1
Feedrate
Feedrate along the X–axis
F1000
F500
When deceleration is not performed
at the corner
When deceleration is performed
at the corner
N1
Time
Feedrate
Feedrate along the Y–axis
F1000
F500
N2
Time
Feedrate
Tangentialfeedrate
F1000
F500
N1
N2
Time

- **Feedrate clamping by acceleration**

When continuous minute straight lines form curves as shown in the example in the figure below, the feedrate difference for each axis at each corner is not so large. For this reason, deceleration according to the feedrate difference is not effective. Continuous small feedrate differences make a large acceleration for each axis as a whole, however.

In this case, deceleration is performed to suppress the shock on the machine and machining errors caused by too large acceleration. The feedrate is decreased so that the acceleration for each axis that is obtained using the expression below does not exceed the allowable acceleration setting for all axes.

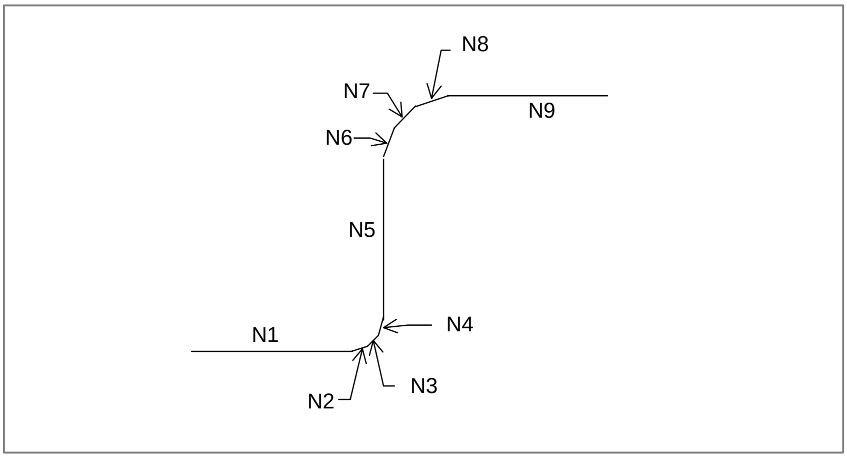
The allowable acceleration is set based on the maximum cutting feedrate (parameter No. 1432) and time required to reach the feedrate (parameter No. 1785).

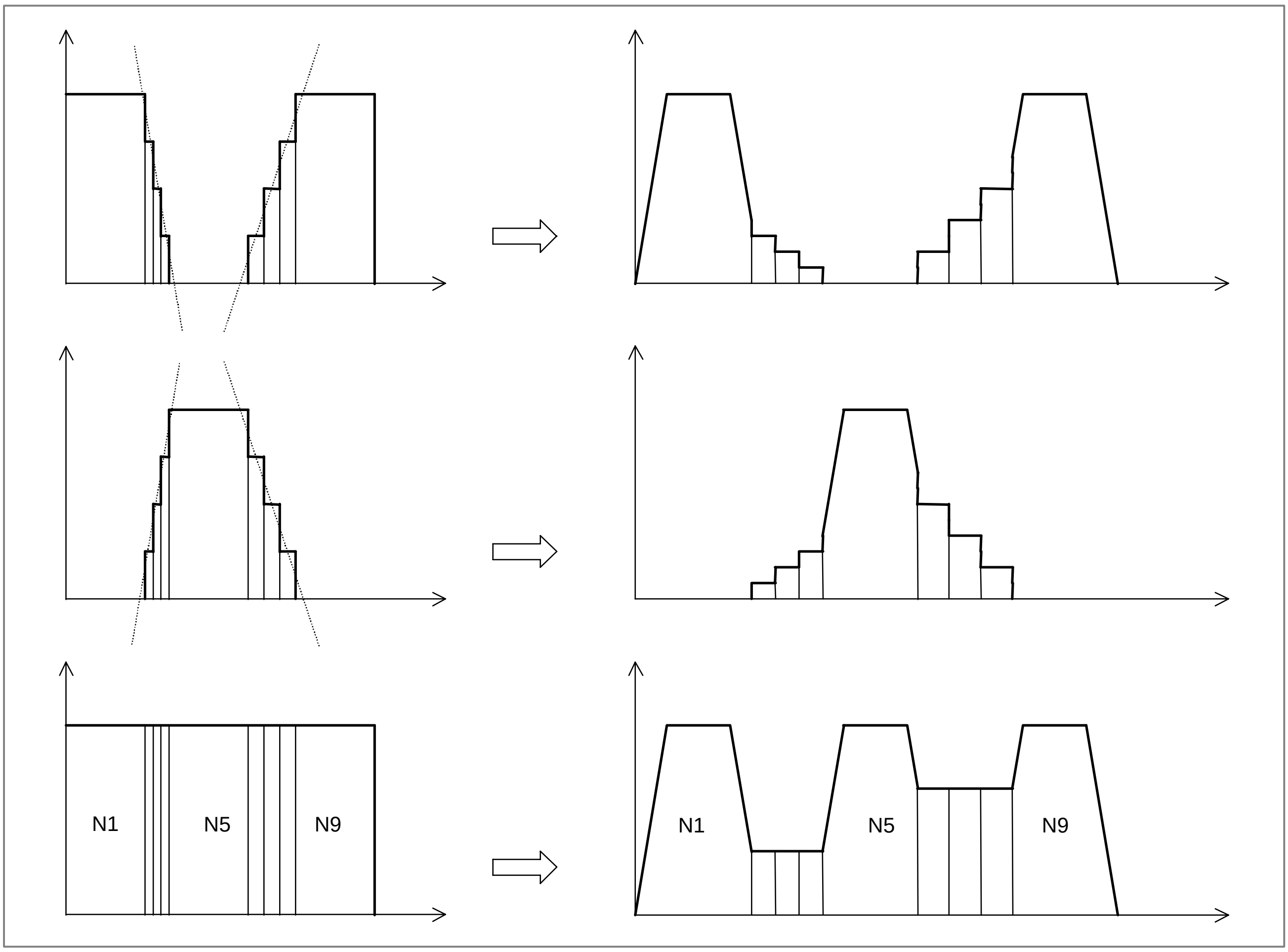
Acceleration for each axis =

$$\frac{\text{Feedrate difference for each axis at each corner}}{\max\left[\dfrac{\text{Travel distance in preceding block}}{F}, \dfrac{\text{Travel distance in following block}}{F}\right]}$$

The decreased feedrate is obtained for each corner. The decreased feedrate obtained at the start or end point of the block, whichever is lower, is used as the actual feedrate.

(Example) In the example below, deceleration is performed because the acceleration (gradient of each dotted line in the feedrate graphs) from N2 to N4 and from N6 to N8 is too large.





- **Feedrate clamping by arc radius**

The maximum allowable feedrate v for an arc of radius r specified in a program is calculated using the arc radius R and maximum allowable feedrate V (setting of a parameter) for the radius as follows so that the acceleration in an arc block does not exceed the allowable value. If the specified feedrate exceeds the feedrate v, the feedrate is automatically clamped to the feedrate v.

$$\text{Maximum allowable feedrate} = \frac{V^2}{R}$$

R : Arc radius V : Feedrate for arc radius R

then, the maximum allowable feedrate v for an arc of radius r can be obtained using the following expression:

$$v = \sqrt{(r/R) \times V}$$

NOTE

As the specified arc radius becomes smaller, the maximum allowable feedrate v becomes lower. If the maximum allowable feedrate v is lower than the setting of parameter No. 1732 (lower feedrate limit for feedrate clamping by arc radius), it can be assumed to be the setting of the parameter to prevent the maximum allowable feedrate from becoming too low.

- **Rapid traverse**

By setting the corresponding parameter, the linear or non–linear interpolation type can be selected. (In the AI nano contour control mode, the non–linear interpolation type cannot be selected.)
When the linear interpolation type is selected, acceleration/deceleration is performed before interpolation and linear interpolation type positioning is used for movement. For acceleration/deceleration, linear or bell–shaped acceleration/deceleration can be selected by setting the corresponding parameter.
The feedrate during movement and acceleration for acceleration/deceleration before interpolation are obtained as follows:

(1) Feedrate during movement

The minimum value obtained using the following expression for each axis along which movement is done is used as the feedrate during movement:

$$\text{Rapid traverse rate for each axis (parameter No. 1420)} = \frac{\text{Amount of travel in block}}{\text{Amount of travel for each axis}}$$

(2) Acceleration for acceleration/deceleration before interpolation

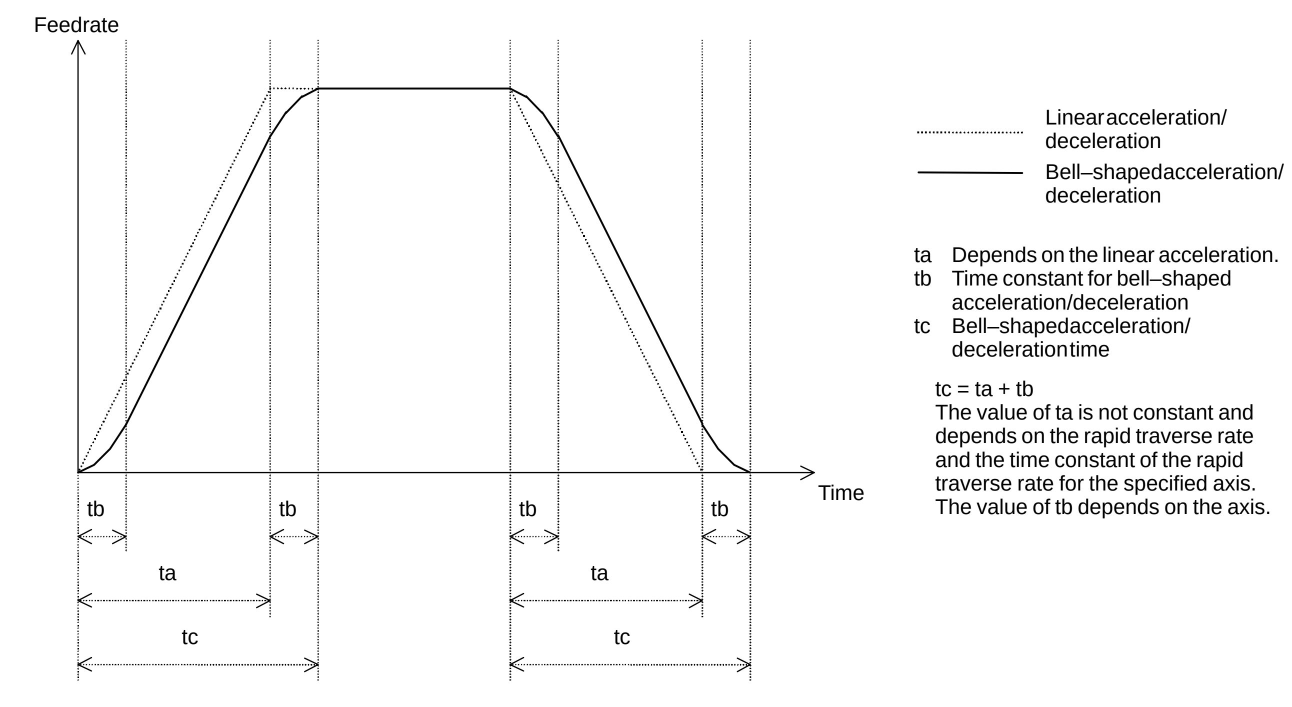
- For linear acceleration/deceleration

The minimum value obtained using the following expression for each axis along which movement is done is used as the acceleration for linear acceleration/deceleration before interpolation during movement:

$$\frac{\text{Rapid traverse rate for each axis (parameter No. 1420)}}{\text{Time constant for each axis (parameter No. 1620)}} \times \frac{\text{Amount of travel in block}}{\text{Amount of travel for each axis}}$$

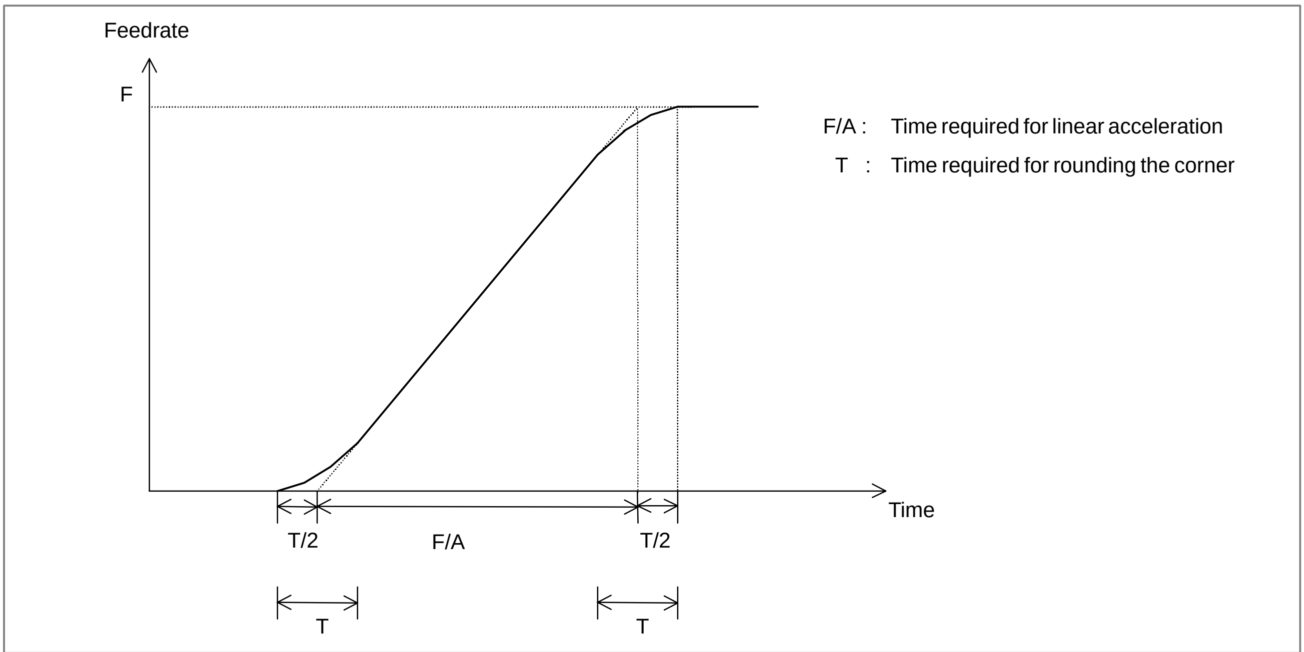
- For bell–shaped acceleration/deceleration

The time constant set in parameter No. 1621 (time constant for bell–shaped acceleration/deceleration for rapid traverse for each axis) for the axis for which the minimum value is obtained using the above expression is applied to the feedrate obtained using the above acceleration.



If the feedrate during movement is F, the acceleration for linear acceleration/deceleration is A, the time constant for bell–shaped acceleration/deceleration is T, the time required for acceleration/ deceleration can be obtained as follows:

Time required for acceleration/deceleration
= F/A (linear acceleration/deceleration)
= F/A+T (bell–shaped acceleration/deceleration)



When the non–linear interpolation type is selected, movement is performed at the feedrate set in parameter No. 1420 with acceleration/deceleration set in parameter No. 1620. The corresponding value can be set in parameter No. 1621 to select bell–shaped acceleration/deceleration.

> **NOTE**
> Rapid traverse block overlap is disabled.

- **Specifications**

Axis control ◯ : Can be specified. × : Cannot be specified.

Name	Function
Number of controlled axes	3 to 4
Number of simultaneously controlled axes	Up to 4
Axis name	The basic three axes are fixed to X, Y, and Z. Other axes are any of U, V, W, A, B, and C.
Least input increment	0.001 mm, 0.001 deg, 0.0001 inch
One–tenth input increment	0.0001 mm, 0.0001 deg, 0.00001 inch One–tenth input increment for each axis cannot be used.
Simple synchronous control	◯ However, switching between synchronous and normal operation cannot be performed during automatic operation (when the automatic operation signal (OP) is set to 1) regardless of whether the AI advanced preview control/AI contour control mode is set. In this case, switching causes a PS213 alarm.
Angular axis control	×
Inch/metric conversion (G20, G21)	◯ (*1)
Interlock	◯
Interlock for each axis	◯ Movement along all axes stops. To stop movement only along the interlock axis in non–linear interpolation type positioning, set bit 5 (AIL) of parameter No. 7054 to 1 and bit 4 (XIK) of parameter No. 1002 to 0.
Machine lock	◯ When the machine lock signal for each axis (MLK1 to MLK8) is turned on or off, acceleration/deceleration is not applied to the axis for which machine lock is performed.
Mirror image	◯
Stored pitch error compensation	◯
Position switch	◯ Set bit 3 (PSF) of parameter No. 6901 to 1. When this parameter is set to 1, the signal output timing changes.
Abnormal load detection	◯
Manual handle interruption	◯ Manual handle interruption is disabled during switching to the AI advanced preview control/AI contour control mode.
External pulse synchronization	×

Interpolation functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Positioning (G00)	○
Single direction positioning (G60)	○ To perform single direction positioning in the AI advanced preview control/AI contour control mode, set bit 4 (ADP) of parameter No. 7055 to 1.
Exact stop (G09)	○
Exact stop mode (G61)	○
Tapping mode (G63)	○
Linear interpolation (G01)	○
Circular interpolation (G02, G03)	○ (Circular interpolation for multiple quadrants is enabled.)
Dwell (G04)	○ (Dwell with the time in seconds or speed specified) For dwell with the speed specified, another option is required.
Cylindrical interpolation (G07.1)	×
Helical interpolation (G02, G03)	○ (Circular interpolation + linear interpolation for up to two axes) Specify the feedrate including the helical axis in the feedrate command.
Threading and synchronous feed (G33)	×
Skip function (G31)	○ (*2)
High–speed skip function (G31)	○ (*2)
Multistage skip function (G31 Px)	○ (*2)
Reference position return (G28)	○ (*2) To execute G28 in the status in which the reference position is not established, set bit 2 (ALZ) of parameter No. 7055 to 1.
Reference position return check (G27)	○ (*2)
2nd, 3rd, and 4th reference position return (G30)	○ (*2)
Normal direction control (G41.1, G42.1)	○ Set bit 2 (ANM) of parameter No. 5484 to 1.
Continuous dressing	×
In–feed control (G161)	×
Index table indexing	○ (*2) To set follow–up of the index table indexing axis (fourth axis), set bit 7 (NAH4) of parameter No. 1819 and bit 0 (NMI4) of parameter No. 7052 to 1.

Feed functions ○ : Can be specified. × : Cannot be specified.

Name	Function
Rapid traverse rate	Up to 240 m/min (0.001 mm)
	Up to 100 m/min (0.0001 mm)
Rapid traverse rate override	F0, 25, 50, 100 %
Rapid traverse rate override in increments of 1%	0 to 100 %
Feed per minute (G94)	○
Feed per revolution (G95)	×
Cutting feedrate clamp	○
Bell–shapedacceleration/deceleration for rapid traverse	○
Linearacceleration/deceleration after cutting feed interpolation	○

Name	Function
Linearacceleration/deceleration before cutting feed interpolation	○ (When the Series 0*i* is used, up to 20 blocks are read in advance in the AI advanced preview control mode.) (When the Series 0*i* Mate is used, up to 12 blocks are read in advance in the AI advanced preview control mode.) (Up to 40 blocks are read in advance in the AI contour control mode.)
Feedrate override	0 to 254 %
Single–digit F code feed	○ To enable feedrate change using a manual handle, set bit 1 (AF1) of parameter No. 7055 to 1.
Override cancel	○
External deceleration	○
Look–ahead bell–shaped acceleration/deceleration before interpolation	○

Program input ○ : Can be specified. × : Cannot be specified.

Name	Function
Control in/control out command ()	○
Optional block skip command (/n: n is a number.)	○
Absolute command (G90)/ incremental command (G91)	○
Decimal point programming/pocket calculator type decimal point programming	○
10–fold input unit	○
Plane selection (G17, G18, G19)	○
Rotation axis specification	○
Rotation axis roll over	○
Polar coordinate command (G16)	×
Local coordinate system (G52)	○ (*2)
Machine coordinate system (G53)	○ (*2)
Workpiece coordinate system (G54 to G59) (G54.1Pxx)	○
Workpiece coordinate system (G92)	○ (*2)
Workpiece coordinate system preset (G92.1)	○ (*2)
Arbitrary angle chamfering/corner rounding	×
Programmable data input (G10)	○ (*2) Only the tool offset value, workpiece origin offset, and parameter can be changed.
Custom macro B	○ See the description of ”Notes on using custom macros.”
Addition of custom macro common variables	○
Pattern data input	×
Interruption type custom macro	×
Canned cycle (G73 to G89)	○ (*2)
Initial level return (G98)/ point R level return (G99)	○ (*2)
Small hole peck drilling cycle (G83)	×
Arc radius R programming	○

Name	Function
Automatic corner override (G62)	○ Set bit 0 (ACO) of parameter No. 7055 to 1.
Automatic corner deceleration	○
Feedrate clamping by arc radius	○
Scaling (G51)	○
Coordinate system rotation (G68)	○
Programmable mirror image (G51.1)	○
F10/11 tape format	○

Auxiliary functions/spindle–speed functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Miscellaneous function (Mxxxx)	○ The function code and function strobe signals are output only.
Second auxiliary function (Bxxxx)	○ The function code and function strobe signals are output only.
High–speed M/S/T/B interface	○
Multiple miscellaneous function specification	○
Spindle–speed function (Sxxxx)	○
Spindle synchronous control	○
Rigid tapping	○ (*2) Set bit 5 (G8S) of parameter No. 1602 or bit 3 (ACR) of parameter No. 7051 to 1.

Tool compensation functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Tool function (Txxxx)	○ The function code and function strobe signals are output only.
Tool offset memory C	○
Tool length compensation (G43, G44, G49)	○
Tool offset (G45 to G48)	×
Cutter compensation C (G40, G41, G42)	○
Tool life management	×
Automatic tool length measurement	×

Other functions ○ : Can be specified. × : Cannot be specified.

Name	Function
Cycle start/feed hold	○
Dry run	○
Single block	○
Sequence number comparison and stop	○
Program restart	○ For the time constant for acceleration/deceleration during movement to the restart position, the following parameters are used: When exponential acceleration/deceleration is used: Parameter Nos. 1624 and 1625 When linear/bell–shaped acceleration/deceleration is used: Parameter No. 1622 To set the acceleration/deceleration type, use bits 0 and 1 of parameter No. 1610.

Name	Function
Rigid tapping return	×
Macro executor (execution macro)	×
MDI operation	○
Manual intervention	○

NOTE
*1 The above tables include a function that another option is required for specifying.
*2 Multiple blocks are not read in advance.

- **Conditions for setting the AI advanced preview control/AI contour control mode**

When G05.1 Q1 is specified, the modal G codes must be set as listed below. If one of these conditions is satisfied, a PS5111 alarm occurs.

G code(s)	Description
G00 G01 G02 G03	Positioning Linear interpolation Circular interpolation (CW) Circular interpolation (CCW)
G15	Polar coordinate command cancel
G25	Spindle speed fluctuation detection off
G40	Cutter compensation cancel
G40.1	Normal direction control cancel mode
G49	Tool length compensation cancel
G50	Scaling cancel
G50.1	Programmable mirror image cancel
G64	Cutting mode
G67	Macro modal call cancel
G69	Coordinate system rotation cancel
G80	Canned cycle cancel
G94	Feed per minute
G97	Constant surface speed control cancel
G160	In–feed control function cancel

Signal

AI advanced preview control or AI contour control mode signal AICC <F062#0>

[Classification] Output signal

[Function] This signal indicates that the system is in AI advanced preview control or AI contour control mode.

[Output condition] The signal is set to 1 when:

- The system is in AI advanced preview control or AI contour control mode.

The signal is set to 0 when:

- The system is in other than AI advanced preview control or AI contour control mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F062								AICC

Parameter

- **Parameters related to linear acceleration/ deceleration before interpolation**

1770	Maximum machining feedrate during linear acceleration/deceleration before interpolation

[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.01 inch/min	6 to 9600	6 to 4800

This parameter sets the maximum machining feedrate for linear acceleration/deceleration before interpolation. (Parameter 1 for setting the acceleration in linear acceleration/deceleration before interpolation)

1771	Time required to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation (time constant)

[Data type] Word

[Unit of data] 1 ms

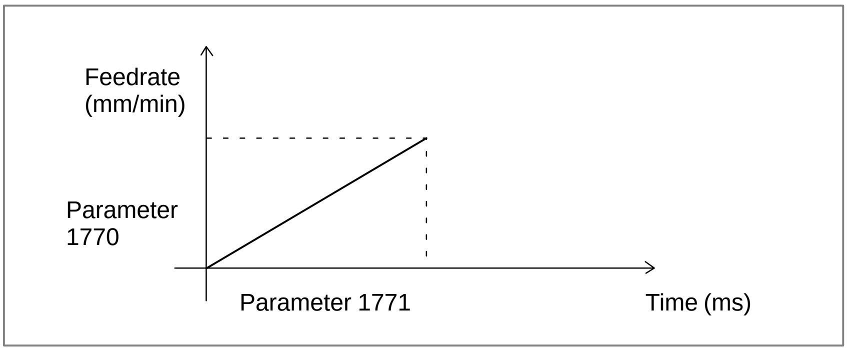
[Valid data range] 0 to 4000

This parameter sets the time (time constant) required to reach the feedrate set in parameter 1. (Parameter 2 for setting the acceleration in linear acceleration/deceleration before interpolation)

NOTE

1 When parameter 1770 or 1771 is set to 0, linear acceleration/deceleration before interpolation is not performed.

2 Set these parameters so that parameter 1770/parameter 1771 = 5 or more.



1784	Speed when an overtravel alarm is issued during linear acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.01 inch/min	6 to 6000	6 to 4800

This parameter sets the speed to be reached when an overtravel alarm is issued during linear acceleration/deceleration during interpolation. If an overtravel alarm is issued during linear acceleration/deceleration before interpolation, the movement is decelerated and halted after the issue of the alarm. Therefore, the tool overruns by an amount equal to the distance traveled during deceleration. The overrun varies depending on the feedrate observed when the overtravel alarm is issued. The overrun can be reduced by performing deceleration to the speed set in parameter 1784 in advance when an overtravel alarm is issued. In this case, deceleration is performed so that the feedrate at the instant when the overtravel alarm is issued does not exceed the parameter–set speed. So, deceleration may be completed earlier. Upon the completion of deceleration, the feedrate is set to the parameter–set speed.

NOTE
This parameter is invalid for rapid traverse blocks.

If the following condition is satisfied, deceleration is performed:

Distance to the stored stroke limit on an axis	<	Distance required for decelerating the current speed (tangent direction feedrate) to the speed set in parameter 1784

The overrun is expressed as follows:

$$\text{Overrun distance} \leqq \frac{\left[\text{FIX}\left(\frac{F_{OT}}{F} \times \frac{T}{8}\right) + 1.5\right]^2}{1875} \times \frac{F}{T}$$

F : Maximum machining feedrate during linear acceleration/deceleration before interpolation (parameter 1770)

T : Time required to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation (parameter 1771)

F_{OT} : Speed when an overtravel alarm is issued during linear acceleration/deceleration before interpolation (parameter 1784)

FIX : Any fractional part is truncated.

NOTE
1 When 0 is set, the above control is not performed.
2 When stroke check is invalid, the above control is also invalid.
3 The above control is valid only for stored stroke check 1.
4 The above control is exercised on those axes that are specified in the current block and the next block.

- **Parameter related to automatic corner deceleration**

1783	Allowable feedrate difference for each axis in the corner deceleration function by feedrate difference (for acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

This parameter sets the allowable difference in feedrate on each axis in the automatic corner deceleration function by the difference in feedrate when liner acceleration/deceleration before interpolation is used.

- **Parameter related to feedrate clamping by acceleration**

1785	Parameter for determining the allowable acceleration in feedrate clamping by acceleration

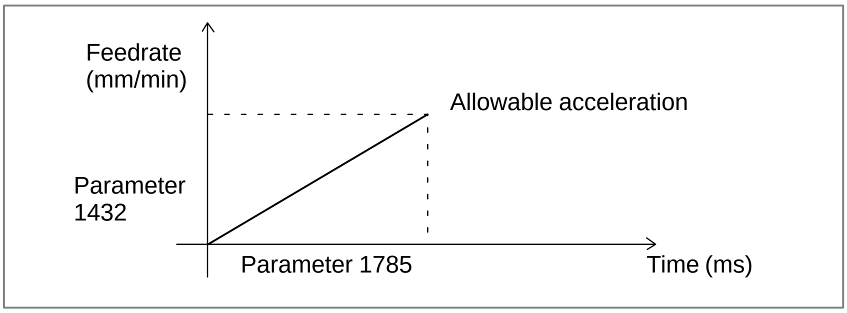
[Data type] Word axis

[Unit of data] 1 ms

[Valid data range] 0 to 32767

This parameter is used to set the time required to reach the maximum cutting feedrate and determine the allowable acceleration when feedrate clamping by acceleration is performed.

The allowable acceleration is determined from the maximum cutting feedrate and the data set in this parameter. Parameter 1432 (maximum cutting feedrate in AI advanced preview control or AI contour control mode) is used as the maximum cutting feedrate.



- **Parameters related to feedrate clamping by arc radius**

1731	Arc radius for the upper limit imposed on feedrate

[Data type] 2–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1000 to 99999999

This parameter sets the arc radius for the upper limit imposed on the feedrate set in parameter 1730.

1730	Upper limit imposed on feedrate for arc radius R

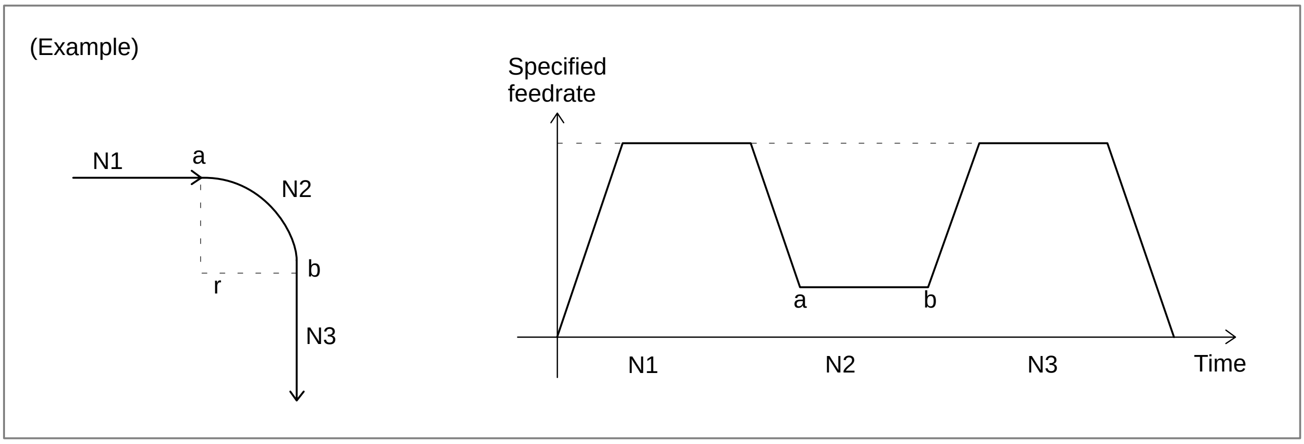
[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 to 15000	8 to 12000
Inch machine	0.1 inch/min	8 to 6000	8 to 4800

This parameter sets the upper limit imposed on the feed rate for the arc radius set in parameter 1731.



1732	Lower limit RVmin for feedrate clamping by arc radius

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 to 15000	0 to 12000
Inch machine	0.1 inch/min	0 to 6000	0 to 4800

When the function for clamping the feedrate by arc radius is used, the upper limit imposed on the feedrate falls with the arc radius. If the upper limit imposed on the feedrate is less than the lower limit imposed on the feedrate clamping RVmin, the upper limit imposed on the feedrate is set as RVmin.

- **Other parameters**

1422	Upper limit imposed on cutting feedrate in AI advanced preview control or AI contour control

[Data type] 2–word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 4800

This parameter is used to set the upper limit on the cutting feedrate in AI advanced preview control or AI contour control mode.

1432	Maximum cutting feedrate in AI advanced preview control or AI contour control mode (for each axis)

[Data type] 2–word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min	0 to 96000	0 to 48000
Rotation axis	1 deg/min	0 to 240000	0 to 100000

This parameter sets the maximum cutting feedrate for each axis in AI advanced preview control or AI contour control mode.

NOTE
Be sure to set a maximum cutting feedrate in both of parameter No. 1422 and No. 1432.

	#7	#6	#5	#4	#3	#2	#1	#0
1603	BEL	RBL						

[Data type] Bit

RBL In the AI advanced preview control or AI contour control mode, acceleration/deceleration of rapid traverse is:
0: Linear acceleration/deceleration.
1: Bell–shaped acceleration/deceleration.

NOTE
To select bell–shaped acceleration/deceleration, the option for rapid traverse bell–shaped acceleration/deceleration is required.

BEL In AI advanced preview control or AI contour control mode:
0 : Linear acceleration/deceleration before look–ahead interpolation is used.
1 : Bell–shaped acceleration/deceleration before look–ahead interpolation is used.

> **NOTE**
> To select look–ahead bell–shaped acceleration/deceleration before interpolation, the option for look–ahead bell–shaped acceleration/deceleration before interpolation is required.

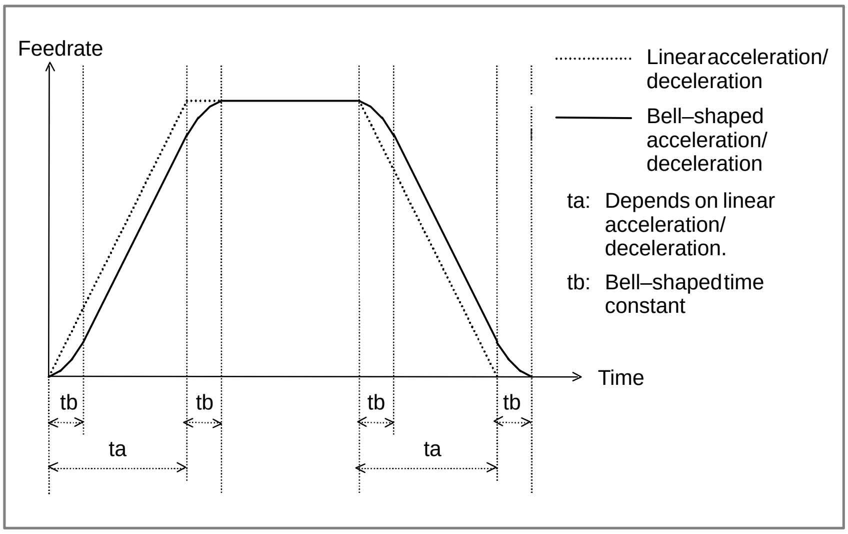
1621	Time constant for axis–by–axis rapid traverse bell–shaped acceleration/deceleration

[Data type] Word axis

[Unit of data] 1 ms

[Valid data range] 0 to 512

Set tb in the figure below for each axis. When 0 is set, linear acceleration/deceleration is assumed.



	#7	#6	#5	#4	#3	#2	#1	#0
6901					PSF			

[Data type] Bit

PSF In AI contour control mode (M series), AI advanced preview control (M series), or advanced preview control mode, position switches are:
0 : Not used.
1 : Used.

NOTE

The position switch signals are output considering acceleration/deceleration after interpolation and servo delay. Acceleration/deceleration after interpolation and servo delay are considered even for position switch signal output in a mode other than the AI contour control (M series), AI advanced preview control (M series), and advanced preview control modes. When this parameter is set to 1, however, signals are output from the position switches at different times from the specified ones.

	#7	#6	#5	#4	#3	#2	#1	#0
7050		MI1	MI0					

[Data type] Bit

MI1, MI0 Set the following values

	MI1	MI0
Setting	0	1

NOTE

This parameter is valid only with AI contour control. This parameter need not be set with AI advanced preview control.

	#7	#6	#5	#4	#3	#2	#1	#0
7051					ACR			

[Data type] Bit

ACR When rigid tapping is specified in AI advanced preview control or AI contour control mode, the mode is:

0 : Not turned off.
1 : Turned off.

When the serial spindle does not support advanced preview control of rigid tapping, AI advanced preview control or AI contour control mode must be turned off in rigid tapping.

Setting this parameter and satisfying the following conditions can automatically turn AI advanced preview control or AI contour control mode off only during execution of rigid tapping when rigid tapping is specified in AI advanced preview control or AI contour control mode.

Conditions

- To specify rigid mode, use “the method for specifying M29 S**** prior to the tapping command.”

  If a method other than the above is used, P/S alarm No. 5110 is issued.
- The interval between M29 (rigid mode specification M code) and the completion signal (FIN) must be at least 32 msec.

- The rigid mode cancel command and cutting feed move command cannot be specified simultaneously. If they are specified simultaneously, P/S alarm No. 5110 is issued.

  (Additional information: The rigid mode cancel command and rapid traverse move command can be specified in the same block.)

- Set bit 2 (CRG) of parameter No. 5200 to 0.

  (This setting specifies that rigid tapping mode is canceled when the rigid tapping signal RGTAP is set to “0”.)

	#7	#6	#5	#4	#3	#2	#1	#0
7052								NMI

[Data type] Bit axis

For the PMC–controlled axes and Cs axis, set 1.
Set this bit to 1 for the index table indexing axis (fourth axis) set for follow–up.
Set this bit to 0 when using the servo FAD function.

> **NOTE**
> This parameter is valid only with AI contour control. This parameter need not be set with AI advanced preview control.

	#7	#6	#5	#4	#3	#2	#1	#0
7054			AIL		AZR		AIR	

[Data type] Bit

AIR In AI advanced preview control or AI contour control mode, the rapid traverse type is:
0 : Linear interpolation type.
1 : According to the setting of bit 1 (LRP) of parameter No. 1401.

AZR In AI advanced preview control or AI contour control mode, the G27, G28, G30, G30.1, and G53 commands are executed:
0 : In normal mode. (advanced preview feed forward is valid.)
1 : In AI advanced preview control or AI contour control mode.

> **NOTE**
> When G27, G28, G30, G30.1, or G53 is executed when AI advanced preview control or AI contour control is on, a choice between linear interpolation type and non–linear interpolation type can be made by parameter setting.
> However, while G28, G30, G30.1, or G53 is being executed with linear interpolation type selected, automatic operation cannot be restarted at a position other than the stop position through manual intervention. If such an attempt is made, an alarm (No. 5114) is issued.

AI advanced preview control or AI contour control	Bit 1 (AIR) of parameter No. 7054	Bit 1 (LRP) of parameter No. 1401	G27, G28, G30, G30.1, G53	G00
Off	0	0	Non–linear interpolation type	Non–linear interpolation type
	0	1	Non–linear interpolation type	Linear interpolation type
	1	0	Non–linear interpolation type	Non–linear interpolation type
	1	1	Non–linear interpolation type	Linear interpolation type
On	0	0	Linear interpolation type	Linear interpolation type
	0	1	Linear interpolation type	Linear interpolation type
	1	0	Non–linear interpolation type	Non–linear interpolation type
	1	1	Linear interpolation type	Linear interpolation type

NOTE
When an index table indexing axis is specified, G27, G28, G30, G30.1, or G53 is executed with AI advanced preview control or AI contour control turned off, regardless of the setting of bit 3 (AZR) of parameter No. 7054.

AIL When non–linear type positioning is specified in AI advanced preview control or AI contour control mode and an axis–by–axis interlock signal is input:
0 : The tool stops along all axes.
1 : The setting of bit 4 (XIF) of parameter No. 1002 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
7055				ADP		ALZ	AF1	ACO

[Data type] Bit

ACO In AI advanced preview control or AI contour control mode:
0 : Automatic corner override and changing both internal and external circular feedrates are disabled.
1 : Automatic corner override and changing the internal circular feedrate are enabled, and whether to enable changing the external circular feedrate depends on the setting of bit 2 (COV) of parameter No. 1602.

AF1 During one–digit F code feed in AI advanced preview control or AI contour control mode, changing the feedrate by the manual handle is:
0 : Disabled.
1 : Enabled.

ALZ If no reference position has been established and G28 is specified in AI advanced preview control or AI contour control mode:
0 : P/S alarm No. 090 is issued.
1 : AI advanced preview control or AI contour control mode is turned off and the command is executed.

NOTE

1 If an axis for which a reference position is established and an axis for which no reference position is established are simultaneously specified with G28 when bit 2 (ALZ) of parameter No. 7055 is set to 1, G28 is executed after turning off the AI advanced preview control or AI contour control mode, regardless of the setting of bit 3 (AZR) of parameter No. 7054.

2 If the serial spindle is switched to the Cs contour control mode then G00 is specified for the Cs contour control axis without performing a reference position return operation even once during AI advanced preview control or AI contour control when bit 1 (NRF) of parameter No. 3700 is set to 0, a P/S alarm (No. 090) is issued, regardless of the setting of bit 2 (ALZ) of parameter No. 7055.

ADP Single direction positioning in the AI advanced preview control or AI contour control mode is executed:
0 : In the normal mode.
1 : In the AI advanced preview control or AI contour contro mode.

3241	Character blinking in the AI advanced preview control or AI contour control mode (first character)
to	
3247	Character blinking in the AI advanced preview control or AI contour control mode (seventh character)

[Data type] Byte

[Valid data range] 0 to 225

Set the character codes of characters blinking in the AI advanced preview control or AI contour control mode.
Set character codes according to the character code list in Appendix A.

NOTE

When 0 is set, “AICC” (AI contour control option is used) or “AI APC” (AI contour control option is not used) blinks.

- **Parameter numbers in standard mode, advanced preview control mode, AI advanced preview control, and AI contour control mode**

(1) Parameters related to linear acceleration/deceleration before interpolation

Parameter	Parameter No.		
	Standard mode	Advanced preview control	AI advanced preview control or AI contour control
Acceleration/deceleration type (type A/B)	FWB/1602#0		None
Parameter 1 for setting acceleration	1630	1770	
Parameter 2 for setting acceleration	1631	1771	
Speed when overtravel alarm is issued	1784		

(2) Parameters related to automatic corner deceleration

Parameter	Parameter No.		
	Standard mode	Advanced preview control	AI advanced preview control or AI contour control
Method for determining automatic corner deceleration (angle/feedrate difference)	CSD/1602#4		None
Lower limit imposed on feedrate (control based on angle)	1778	1777	None
Angle to be determined (control based on angle)	1740	1779	None
Allowable feedrate difference for all axes (control based on feedrate difference)	1780		None
Allowable feedrate difference for each axis (control based on feedrate difference)	1783		

(3) Parameters related to feedrate clamping by acceleration/deceleration

Parameter	Parameter No.		
	Standard mode	Advanced preview control	AI advanced preview control or AI contour control
Parameter for determining acceleration/deceleration	None		1785

(4) Parameters related to feedrate clamping by arc radius

Parameter	Parameter No.		
	Standard mode	**Advanced preview control**	**AI advanced preview control or AI contour control**
Arc radius for the upper limit of feedrate	1731		
Upper limit imposed on feedrate for arc radius R	1730		
Lower limit imposed on clamp feedrate	1732		

(5) Parameters related to involute interpolation

Parameter	Parameter No.		
	Standard mode	**Advanced preview control**	**AI advanced preview control or AI contour control**
Initial angle error limit	5610		
Basic circle neighborhood override: Radius of curvature	None		5611 to 5615
Basic circle neighborhood override: Override value	None		5616 to 5619
Lower override limit	None		5620

(6) Other parameters

Parameter	Parameter No.		
	Standard mode	**Advanced preview control**	**AI advanced preview control or AI contour control**
Precision of radius error in circular interpolation	PCIR1/3403#0		None
Maximum cutting feedrate (for all axes)	1422	1431	1422
Maximum cutting feedrate (for each axis)	1430	1432	
Rapid traverse type	LRP/1402#1		AIR/7054#1 LRP/1401#1
Rapid traverse bell–shaped acceleration/deceleration time constant	1621		

Alarm and message

Number	Message	Description
5110	IMPROPER G–CODE (G05.1 G1 MODE) (M series)	An invalid G code is specified in AI advanced preview control or AI contour control mode.
5111	IMPROPER MODAL G–CODE (G05.1 G1) (M series)	When AI advanced preview control or AI contour control mode is specified, a G code that cannot be used is placed in the model state.
5112	G08 CAN NOT BE COMMANDED (G05.1 G1) (M series)	An advanced preview control command (G08) is specified in AI advanced preview control or AI contour control mode.
5114	NOT STOP POSITION (G05.1 Q1) (M series)	Since manual intervention was performed while executing the G28, G30, G30.1, and G53 commands (liner interpolation type) with AI advanced preview control or AI contour control turned on, automatic operation is restarted at a position other than the stop position.
5156	SPL: ERROR (M series)	The controlled axis selection signal (PMC axis control) changes in AI advanced preview control or AI contour control mode. The simple synchronous axis selection signal changes in AI advanced preview control or AI contour control mode.
5157	Feedrate 0 (AICC)	The parameter for the maximum cutting feedrate (parameter No. 1422 or 1432) is set to 0. The parameter for acceleration/deceleration before interpolation (parameter No. 1770 or 1771) is set to 0.

Caution

1 When the total distance for the blocks under advanced preview control is equal to or less than the deceleration distance from the current feedrate, deceleration starts. When advanced preview control has proceeded upon the completion of deceleration, and the total distance for the blocks increases,acceleration starts. Especially, when a series of blocks containing very small amounts of travel are specified, deceleration and acceleration may be alternated, which prevents the feedrate from becoming constant. In such a case, specify a lower feedrate.

2 When the dry run signal is inverted from 0 to 1 or from 1 to 0 during movement along an axis, the speed of movement is increased or reduced to a specified speed without first being reduced to zero.

3 When a no–movement block or a one–shot G code such as G04 is encountered in AI advanced preview control or AI contour control mode, the movement is decelerated and halted in the preceding block.

4 As acceleration after interpolation, use linear or bell–shaped acceleration. Exponential acceleration/deceleration cannot be used.

Notes on using custom macros

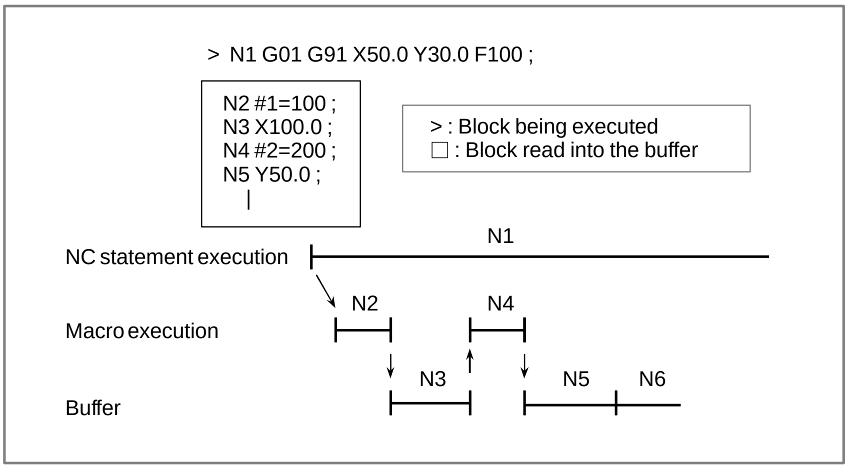
Macro statement processing

For smooth machining, the CNC reads the next NC statement in advance. This operation is referred to as buffering. In the AI advanced preview control mode, not only the next statement but also multiple blocks are buffered.

However, a macro statement such as an expression and conditional branch is processed immediately after it is buffered (read into the buffer). So, the timing of macro statement execution does not necessarily follow the specified sequence.

On the contrary, a block where M00, M01, M02, M30, or an M code set in a parameter (No. 3411 to 3432) for suppressing buffering is specified, or a block where a G code such as G53 for suppressing buffering is specified does not read subsequent blocks in advance. So, it is guaranteed that until the execution of such an M code or G code is completed, no subsequent macro statement is executed.

Details of NC statement and macro statement execution



In the AI advanced preview control or AI contour control mode, multiple blocks are read in advance. So, during execution of N1, up to 40 blocks/20 blocks (0*i*) or 12 blocks (0*i* Mate) of NC statements are read into the buffer. The macro statements (N2, N4) are processed while N1 is being executed.

So, the sequence of NC statement and macro statement execution differs from the programmed sequence.

Notes on using system variables

When a system variable listed in the table below is used in a macro program, and the macro program needs to be executed after the block immediately preceding the macro program is executed, an M code (parameter No. 3411 to No. 3432) for suppressing buffering or G53 must be inserted immediately before the macro program.

Meaning	Read/ write	Variable number	Remarks (when buffering is not suppressed)
Interface signal	Read	#1000 to #1015, #1032	Data can be read when a macro is buffered.
	Write	#1100 to #1115, #1132	Data is written when a macro is buffered.
Tool offset value	Write	#10000 to	A tool offset value is written when a macro is buffered.
Time information	Read	#3001, #3002, #3011, #3004	Time information is read when a macro is buffered.
Automatic operation control	Write	#3003, #3004	Automatic operation control is enabled starting with up to the third block ahead.
Setting data	Write	#3005	Setting data is written when a macro is buffered.
Mirror image	Read	#3007	Mirror image state can be read when a macro is buffered.
Additional workpiece coordinate system number currently selected	Read	#4130(P) #4014 (G54 to G59)	The information of up to 3 blocks ahead can be read.
Block end (workpiece coordinate system)	Read	#5001 to #5008	The block end of up to the third block ahead can be read.
Machine coordinate system	Read	#5021 to #5028	An undefined position during travel can be read.
Current position (workpiece coordinate system)	Read	#5041 to #5048	An undefined position during travel can be read.
Tool length compensation value	Read	#5081 to #5088	The compensation value used by the block currently being executed can be read.
Servo positional deviation amount	Read	#5101 to #5108	An undefined positional deviation amount during travel can be read.

Example)

O0001	O2000
N1 X10.Y10.;	(Mxx;) Inserts an M code block for suppressing buffering.
N2 M98P2000;	N100 #1=#5041; (Reads the current position along the X–axis.)
N3 Y200.0;	N101 #2=#5042; (Read the current position along the Y–axis.)
:	:
	M99;

In the example above, while the N1 block of the main program O0001 is being executed, the N2 block is buffered, and the macro program O2000 is read and executed. This means that the current position is read during travel along the axes in the N1 block. So, undefined position information during travel is read into #1 and #2. In such a case, insert a block specifying an M code (Mxx;) for suppressing buffering or G53; immediately before the N100 block of O2000. With this block insertion, the execution of O2000 starts when the execution of the N1 block of O0001 is completed. So, position information after the completion of execution of the N1 block can be read into #1 and #2.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.19.3	AI advanced preview control/AI contour control (G05.1)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.19.3	AI advanced preview control (G05.1)

7.2 ACCELERATION/ DECELERATION CONTROL

7.2.1 Automatic Acceleration/ Deceleration

7.2.1.1 Automatic acceleration/ deceleration

General

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 7.2.1.1).

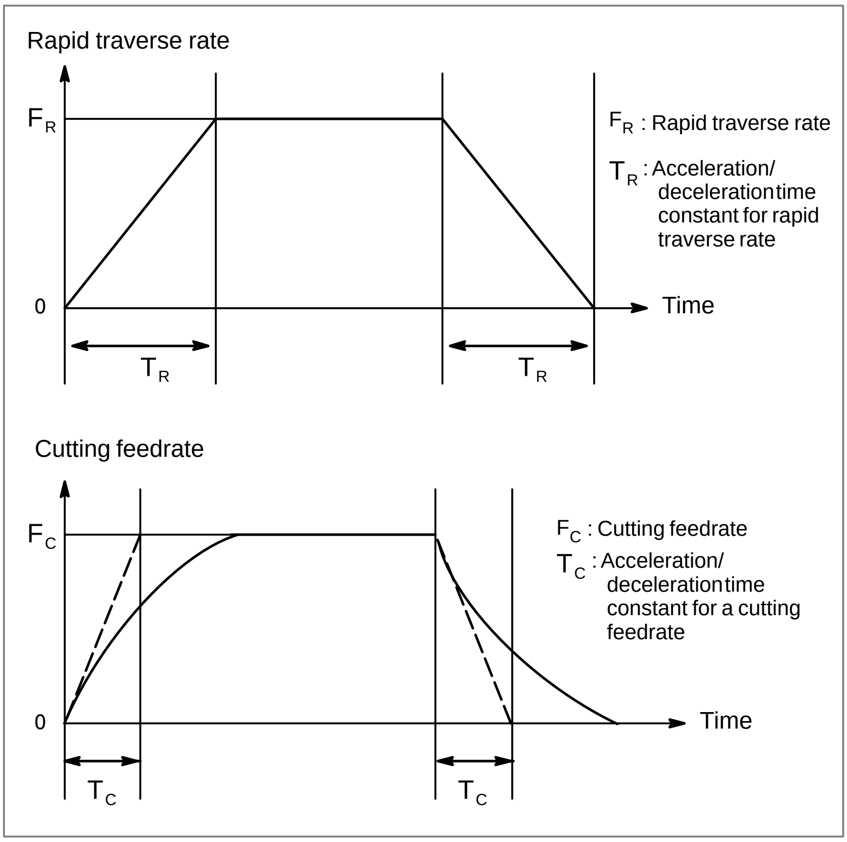


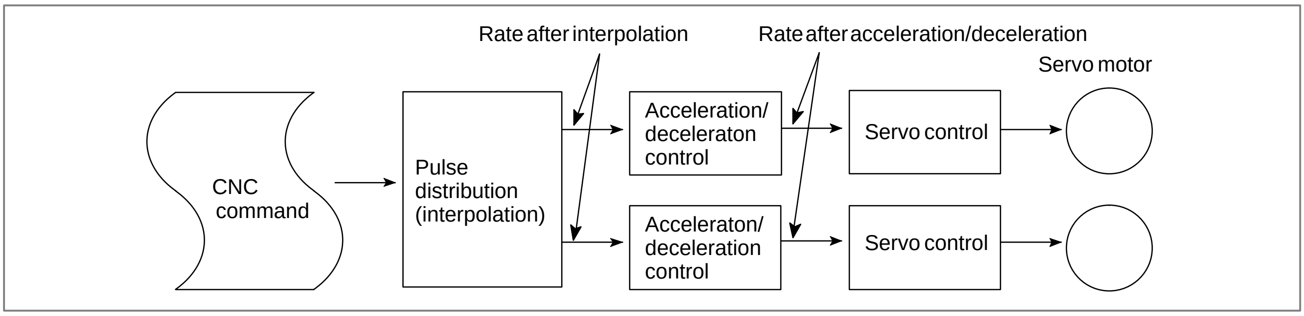
Fig. 7.2.1.1 Automatic acceleration/deceleration (example)

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so the change in speed is also smooth.
It is not necessary to take acceleration/deceleration into consideration when programming.

Rapid traverse: Linear acceleration/deceleration (time constant per axis is set by parameter 1620)

Cutting feed: Exponential acceleration/deceleration (time constant per axis is set by parameter 1622)

Jog feed : Exponential acceleration/deceleration (time constant per axis is set by parameter 1624)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

1620	Time constant used for linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.

(1) When the function is used, set this parameter to time constant T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.

(2) When the function is not used, specify a time constant used in linear acceleration/deceleration.

NOTE
When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even. In this case, this parameter stands for a time constant used in linear acceleration/deceleration in rapid traverse.

1622	Time constant of exponential acceleration/deceleration or linear acceleration/ deceleration after interpolation, in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration / deceleration)
0 to 512 (For linear acceleration / deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1623	FL rate of exponential acceleration/deceleration in cutting feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0, 6 – 15000	0, 6 – 12000
Inch machine	0.1 inch/min	0, 6 – 6000	0, 6 – 4800
Rotaion axis	1 deg/min	0, 6 – 15000	0, 6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed for each axis. Except for special applications, this parameter must be set to 0 for all axes. If a value other than 0 is specified, proper straight lines and arcs cannot be obtained.

1624	Time constant of exponential acceleration/deceleration or linear acceleration/ deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration / deceleration)
0 to 512 (For linear acceleration / deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

1626	Time constant of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

1627	FL rate of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

7.2.1.2 Rapid traverse block overlap

General

Rapid traverse blocks may be arranged successively or a rapid traverse block may be followed by a block that does not cause movement. In this case, execution of the next block can be started when the feedrate for each axis of the rapid traverse block is decreased to the deceleration ratio specified in the parameter.

Signal

Rapid traverse block overlap disable signal ROVLP <G0053#5>

[Classification] Input signal

[Function] This signal disables rapid traverse block overlap.

Siganl address

	#7	#6	#5	#4	#3	#2	#1	#0
G053		ROVLP						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

[Data type] Bit

RTO Rapid traverse block overlap is:

0 : Not performed.
1 : Performed.

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Data type] Byte axis

[Unit of data] %

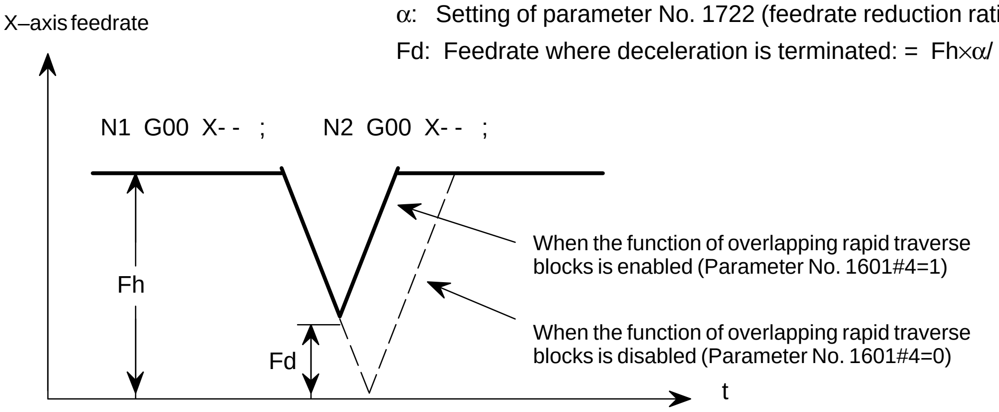
[Valid data range] 0 to 100

This parameter is used when rapid traverse blocks are arranged successively, or when a rapid traverse block is followed by a block that does not cause movement. When the feedrate for each axis of a block is reduced to the ratio set in this parameter, the execution of the next block is started.

> **NOTE**
> The parameter No. 1722 is effective when parameter No. 1601 #4 (RTO) is set to 1.

Example

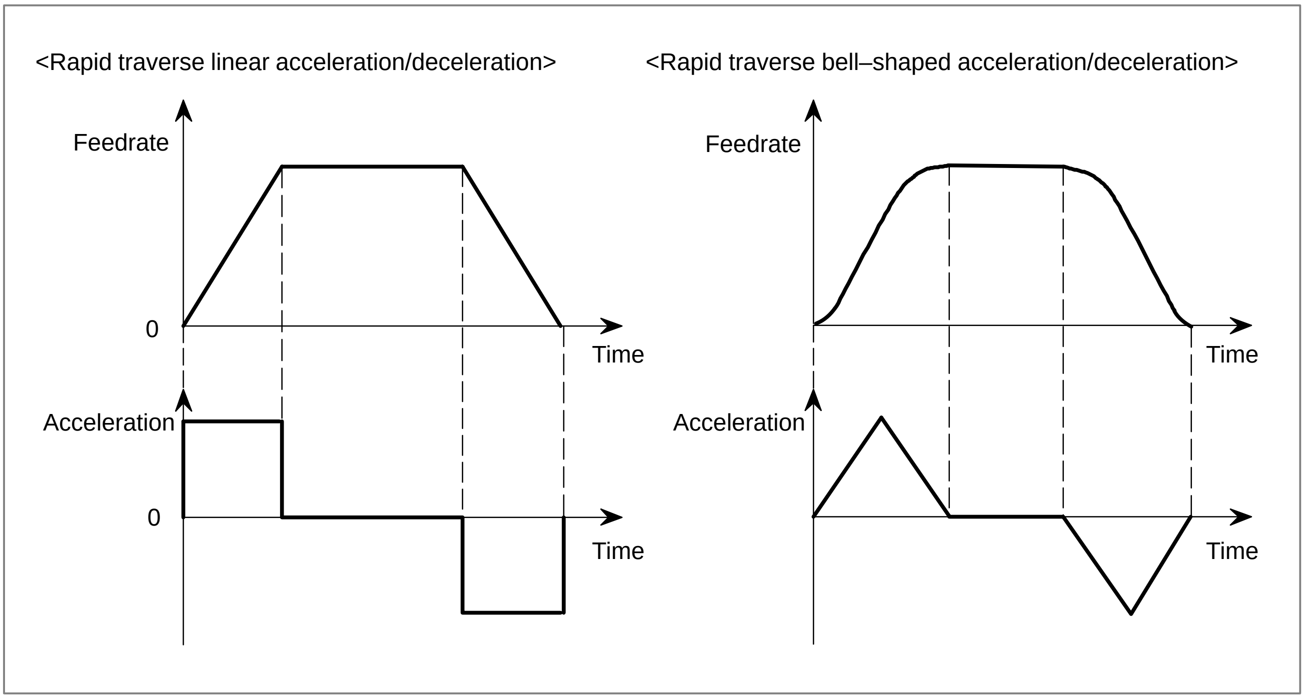
Fh: Rapid traverse feedrate
α: Setting of parameter No. 1722 (feedrate reduction ratio)
Fd: Feedrate where deceleration is terminated: = Fh×α/ 100



7.2.2 Rapid Traverse Bell–shaped Acceleration/ Deceleration

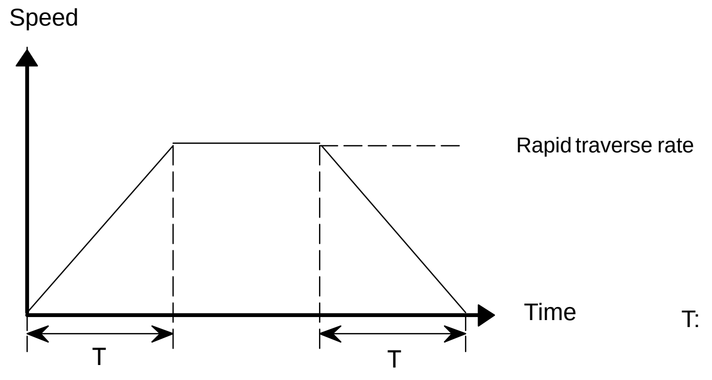
General

Rapid traverse bell–shaped acceleration/deceleration smoothly increases or decreases the rapid traverse rate, reducing the stress and strain imposed on the machine due to the variation in the acceleration with changes in the feedrate. As the time constant for bell–shaped acceleration/deceleration can be smaller than that for linear acceleration/deceleration, the time needed for acceleration/deceleration can be reduced.



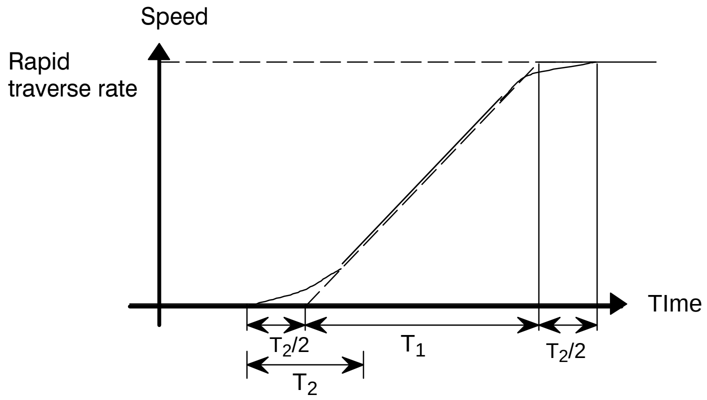
This function is enabled when the time constants for rapid traverse bell–shaped acceleration/deceleration T_1 and T_2 are specified in parameter Nos. 1620 and 1621, respectively.

<Rapid traverse linear acceleration/deceleration>



T: Time constant for linear acceleration/deceleration

<Rapid traverse bell shaped acceleration/deceleration>



T_1: Set a time constant used for linear acceleration/deceleration

T_2: Set a time for rounding.

Total time=$T_1 + T_2$
Time for linear=$T_1 – T_2$
Time for rounding part=T_2

Set a time when rapid traverse override is 100% . When it is less than 100%, the total time is reduced (constant acceleration method).

Value of T_1 is determined from motor torque. Set a value of T_2 to 24 ms or 32 ms.

Parameter

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse.

- When the function is used, set this parameter to time constant T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.
- When the function is not used, specify a time constant used for linear acceleration/deceleration.

> **NOTE**
> When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even. In this case, this parameter stands for a time constant used for linear acceleration/ deceleration in rapid traverse.

1621	Time constant t T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 512

Specify time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis.

> **NOTE**
> 1 Set parameter No. 1620 to time constant T1 used for bell–shaped acceleration/deceleration in rapid traverse, and set this parameter to time constant T2.
> For details of time constants T1 and T2, see the general description in this section.
> 2 When this parameter is set to 0, linear acceleration/ deceleration is applied in rapid traverse. The setting in parameter No. 1620 is used as a time constant in linear acceleration/deceleration.

Reference item

CONNECTION MANUAL (This manual)	7.2.1	AutomaticAcceleration/Deceleration

7.2.3 Linear Acceleration/ Deceleration after Cutting Feed Interpolation

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 0 of parameter No. 1610, CTL), acceleration/deceleration is performed as follows:

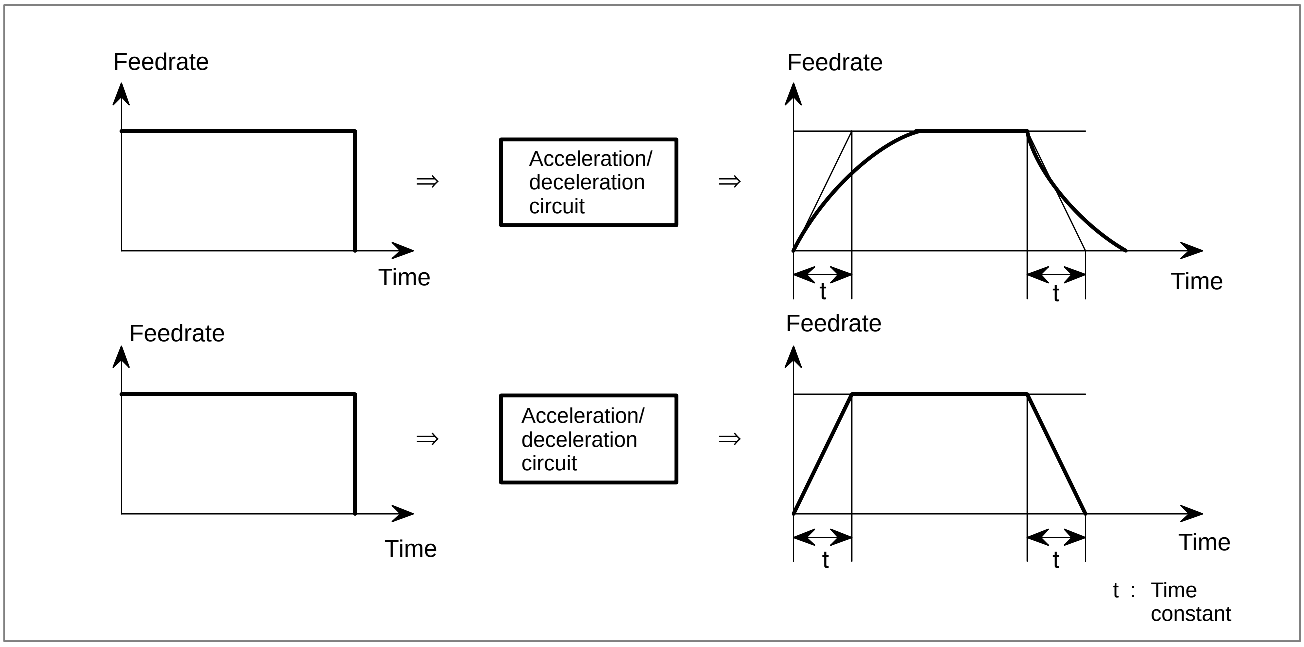
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If an identical time constant is specified, linear acceleration/deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/deceleration, thus reducing the time needed for acceleration and deceleration. If circular interpolation is performed, especially when high–speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.



Linear acceleration/deceleration after cutting feed interpolation function is enabled when the CTL bit (bit 0 of parameter No. 1610) is specified. The time constants for cutting feed and jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

1622	Time constant of exponential acceleration/deceleration, linear acceleration/ deceleration after interpolation in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of exponential acceleration/deceleration or linear acceleration/ deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation)

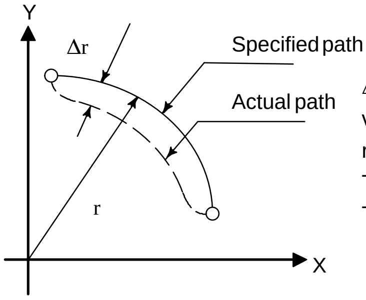
Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

Note

NOTE

1 If linear acceleration/deceleration after interpolation for cutting feed is enabled, linear acceleration/deceleration is executed during cutting feed and during a dry run. Linear acceleration/deceleration can also be executed during jog feed if the JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

2 In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr :Maximum radius error (mm)
v :Feedrate (mm/s)
r :Radius of arc (mm)
T_1 :Acceleration/deceleration time constant (s)
T_2 :Time constant of servo loop (s)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2}T_1^2 + \frac{1}{2}T_2^2\right)\frac{v^2}{r}$$ …Exponential acceleration/deceleration

$$\Delta r = \left(\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2\right)\frac{v^2}{r}$$ … Linear acceleration/deceleration after interpolation

If the error caused by the time constant of the servo loop is excluded, the error cause by linear acceleration/ deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration.

3 Linear acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. Acceleration/deceleration for cutting feed is executed even if acceleration/deceleration for jog feed is selected. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.4 Bell–Shaped Acceleration/ Deceleration after Cutting Feed Interpolation

General

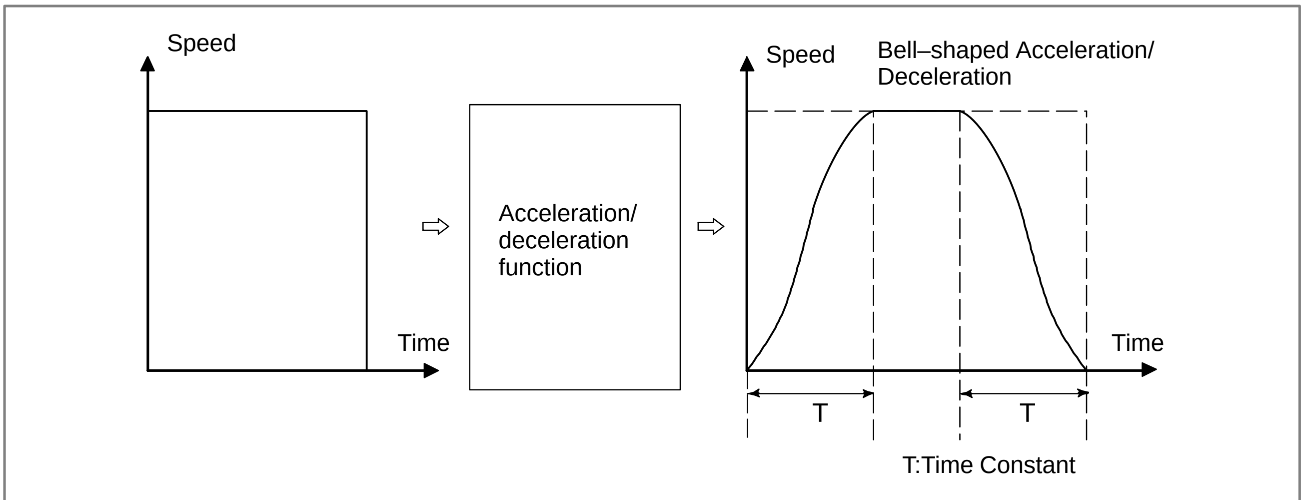
The bell–shaped acceleration/deceleration after cutting feed interpolation provides smooth acceleration and deceleration to reduce stress and strain on the machine. If this function is enabled (bit 1 of parameter No. 1610, CTB), acceleration/deceleration is performed as follows:

Cutting feed: Bell–shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or bell–shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.



Bell–shaped acceleration/deceleration after cutting feed interpolation is an optional function. This function is enabled when the CTB bit (bit 1 of parameter No. 1610) is specified. The time constants for cutting feed and for jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

NOTE
To use bell–shaped acceleration/deceleration after interpolation, set this parameter to 0 and select the acceleration/deceleration using CTBx, bit 1 of parameter No. 1610.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

CTBx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration or linear acceleration/deceleration after interpolation is applied (depending on the setting in CTLx, bit 0 of parameter No. 1610).
1 : Bell–shaped acceleration/deceleration after interpolation is applied.

NOTE
This parameter is effective only when the function of bell–shaped acceleration/deceleration after interpolation in cutting feed is provided. If the function is not provided, the setting in CTLx, bit 0 of parameter No. 1610, determines the type of acceleration/deceleration irrespective of the setting in this parameter.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1622	Time constant of exponential acceleration/deceleration, linear acceleration/ deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

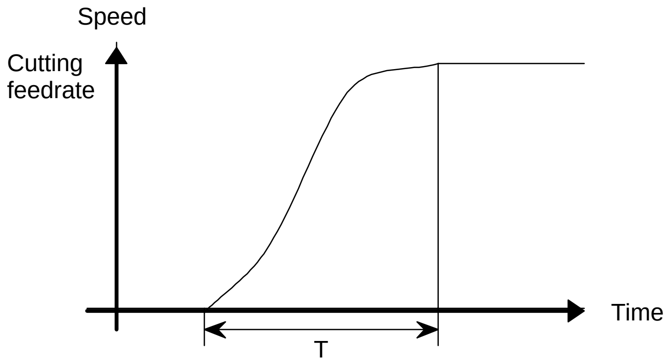
[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

<Bell–shaped acceleration/deceleration after cutting feed interpolation>



T : Total time. It is constant irrespective of feedrate. (Time constant is constant).

The curve corresponds to that T1 = T/2 and T2 = T/2 set in parameter No. 1620 and 1621.
No linear part exists.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/ deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determinded depending on the setting in parameter No. 1610.

Note

NOTE

1 If bell–shaped acceleration/deceleration after interpolation during cutting feed is enabled, bell–shaped acceleration/deceleration is executed during cutting feed and during a dry run. Bell–shaped acceleration/deceleration can also be executed during jog feed if the JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

2 In circular interpolation, the actual tool path after acceleration/deceleration deviates from the programmed arc in the radial direction. To overcome this radial deviation, see the note on linear acceleration/deceleration after interpolation for cutting feed in Subsection 7.2.3.

3 Bell–shaped acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. The time constant for acceleration/deceleration for jog feed is the same as that for cutting feed. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.5 Corner Control

7.2.5.1 In–position check

General

Whether the position of the servo motor is within a specified range is checked.

If the in–position check function is enabled, the CNC checks the position during deceleration. If the position is found to exceed the specified range, the CNC does not execute the next block.
In–position check is performed even for the axis that reached its torque limit.

> **NOTE**
> The purpose of in–position check is to check that the servo motor has reached within a specified range (specified with parameter No. 1826).

Signal

In–position signals INP1 to INP4 <F104#0 to #3>

[Classification] Output signal

[Function] These signals indicate that the control axes are set to the in-position condition.
They are provided for each control axis, and the number in the signal name corresponds to the control axis number.

I N P 1

1 . . . . The 1st axis is set to the in-position condition.
2 . . . . The 2nd axis is set to the in-position condition.
3 . . . . The 3rd axis is set to the in-position condition.
: :
: :

[Output condition] These signals turn to “1” in the following case :

- The servo error of the corresponding control axis is within the specified allowance.

These signals turn to “0” in the following cases :

- The servo error of the corresponding control axis exceeds the specified allowance

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F104					INP4	INP3	INP2	INP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI Inposition check at deceleration

0 : Performed
1 : Not performed

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

The in–position width is set for each axis.

When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in–position width, the machine is assumed to have reached the specified position.

Note

NOTE

1 The in–position signals may turn to “1” even during the movement if the axis is fed at very low speed.
2 The in–position check function is enabled, at the interface between two cutting blocks, in the following cases:

M series	When the exact stop command (G09) or exact stop mode command (G61) is specified
T series	When the error detect signal is on

7.2.5.2 In–position check independently of feed/rapid traverse

General

If separate in–position check for cutting feed and rapid traverse is executed, a small in–position check range can be specified between those cutting feed blocks that require a high degree of precision. A large in–position check range can be specified between those rapid traverse blocks that require quick positioning.

Signal

See Subsection 7.2.5.1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI Inposition check at deceleration

0 : Performed
1 : Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
1801			CIN	CCI				

[Data type] Bit

CCI The in–position area for cutting feed is:

0 : Set in parameter No. 1826 (same as for rapid traverse).
1 : Set in bit 5 (CIN) of parameter No. 1801.

CIN When bit 4 (CCI) of parameter No. 1801 = 1, the in–position area for cutting feed is:

0 : Use value in parameter No. 1827 if the next block is also for cutting feed, or use value in parameter No. 1826 if the next block is for rapid traverse.
1 : Use value in parameter No. 1827, regardless of the next block. (The setting of parameter No. 1826 is used for rapid traverse, and the setting of parameter No. 1827 is used for cutting feed.)

<table>
<tr><th colspan="2" rowspan="2"></th><th colspan="4">Parameter CIN (No. 1801#5)</th></tr>
<tr><th colspan="2">0</th><th colspan="2">1</th></tr>
<tr><td rowspan="2">Parameter CCI (No. 1801#4)</td><td>0</td><td>Rapid→Rapid
Rapid→Feed
Feed→Feed
Feed→Rapid</td><td>No. 1826
No. 1826
No. 1826
No. 1826</td><td>Rapid→Rapid
Rapid→Feed
Feed→Feed
Feed→Rapid</td><td>No. 1826
No. 1826
No. 1826
No. 1826</td></tr>
<tr><td>1</td><td>Rapid→Rapid
Rapid→Feed
Feed→Feed
Feed→Rapid</td><td>No. 1826
No.1826
No.1827
No. 1826</td><td>Rapid→Rapid
Rapid→Feed
Feed→Feed
Feed→Rapid</td><td>No. 1826
No. 1826
No. 1827
No. 1827</td></tr>
</table>

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

The in–position width is set for each axis.

When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in–position width, the machine is assumed to have reached the specified position.

1827	In–position width in cutting feed for each axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set an in–position width for each axis in cutting feed. This parameter is valid when bit 4 (CCI) of parameter No. 1801=1.

Note

NOTE

If the NCI bit (bit 5 of parameter No. 1601) is set to 1, so that position check is not performed during deceleration, this function is invalid. The system starts execution of the next block as soon as deceleration has been completed, without checking whether the servo position error is within the specified range.

7.2.5.3 In–position check disable signal

General

The in–position check disable signal NOINPS <G023#5> specifies whether to make in–position checks.
In addition, each of the individual–axis in–position check disable signals NOINP1 to NOINP4 <G359#0 to G359#3> specifies whether to make in–position checks for the corresponding axis.
If it is specified to make in–position checks, the CNC makes in–position checks during deceleration and does not start executing the next machining block until the corresponding axis gets in position.

Signal

In–position check disable signal NOINPS <G023#5>

[Classification] Input signal

[Function] Specifies whether to make in–position checks.

[Operation] If this signal is ”1”, the control unit behaves as follows:
No in–position check is made during deceleration regardless of the setting of the NCI parameter (bit 5 of parameter No. 1601). When no in–position check is made, the behavior of the control unit is the same as when the NCI parameter (bit 5 of parameter No. 1601) = 1.
If this signal is ”0”, the control unit behaves as follows:
Whether to make in–position checks is determined according to the setting of the NCI parameter (bit 5 of parameter No. 1601).

Individual–axis in–position check disable signals NOINP1 to NOINP4 <G359#0 to #3>

[Classification] Input signal

[Function] Each of these signals specifies whether to make in–position checks for the corresponding axis.

[Operation] If one of these signals is ”1”, the control unit behaves as follows:
No in–position check is made on the corresponding axis during deceleration regardless of the setting of the NCI parameter (bit 5 of parameter No. 1601) and the state of the in–position check disable signal NOINPS <G023#5>. When no in–position check is made, the behavior of the control unit is the same as when the NCI parameter (bit 5 of parameter No. 1601) = 1.
If one of these signals is ”0”, the control unit behaves as follows:
Whether to make in–position checks for the corresponding axis is determined according to the setting of the NCI parameter (bit 5 of parameter No. 1601) and the state of the in–position check disable signal NOINPS.

Siganl address

CNC→PMC

	#7	#6	#5	#4	#3	#2	#1	#0
F104			NOINPS					

	#7	#6	#5	#4	#3	#2	#1	#0
G359					NOINP4	NOINP3	NOINP2	NOINP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI In–position check at deceleration

0 : Performed
1 : Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
8007								NIS

[Data type] Bit

NIS In the in–position check of PMC axes, in–position check disable signal NOINPS <G023#5> and in–position check disable signals for individual axes NOINP1 to NOINP4 <G359#0 to #3> are:

0 : Invalid.
1 : Valid.

Note

NOTE

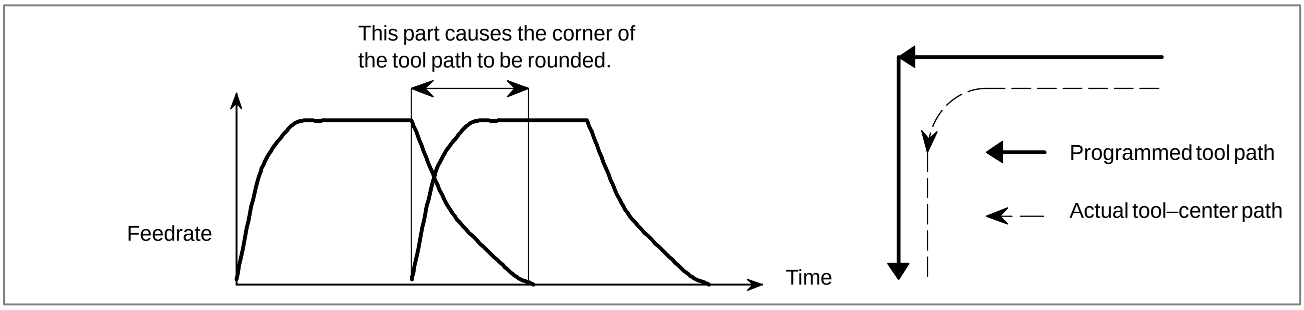
1 Although in–position checks of ordinary blocks can be disabled by using in–position check disable signal NOINPS <G023#5> and in–position check disable signals for individual axes NOINP1 to NOINP4 <G359#0 to #3> , the in–position check at the reference position obtained by a reference position return operation (G28 or G30) is always performed. The in–position check at an intermediate point, however, can be disabled.

2 When the in–position check disable signals for individual axes NOINP1 to NOINP4 <G359#0 to #3> are used, PMC–SB7 is necessary.

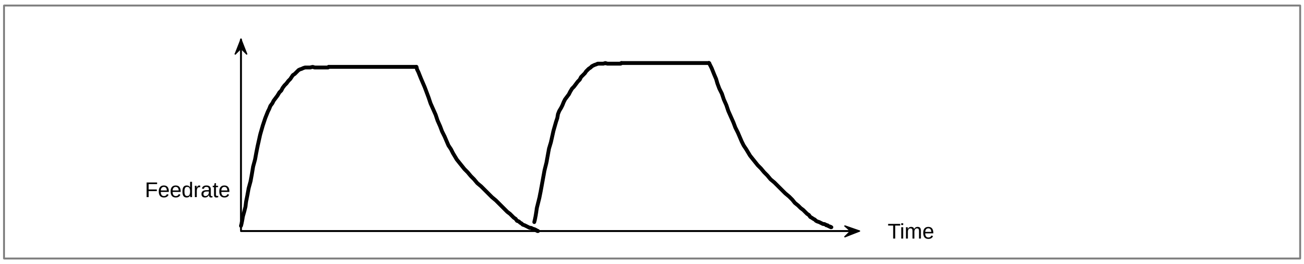
7.2.5.4 Error detect (T series)

General

Generally, the CNC does not zero the feedrate at the interface of two blocks during cutting feed.
Because of this, a corner of a tool path may be rounded.



If the error detect signal is used, it is possible to specify that a block not be started until the acceleration/deceleration of the previous block has been completed.



Signal

Error detect signal SMZ <G053, #6>

[Classification] Input signal

[Function] Enables error detection.

[Operation] If the signal is set to 1, the control unit operates as follows:

- At the interface of two blocks during cutting feed, the control unit waits until the acceleration/deceleration of the first block has been completed. Only then does the control unit execute the next block.
  The setting of the SMZ signal determines whether, at the interface of two cutting blocks, the control unit waits until the acceleration/deceleration of the previous block has been completed.

Siganl address

	#7	#6	#5	#4	#3	#2	#1	#0
G053		SMZ						

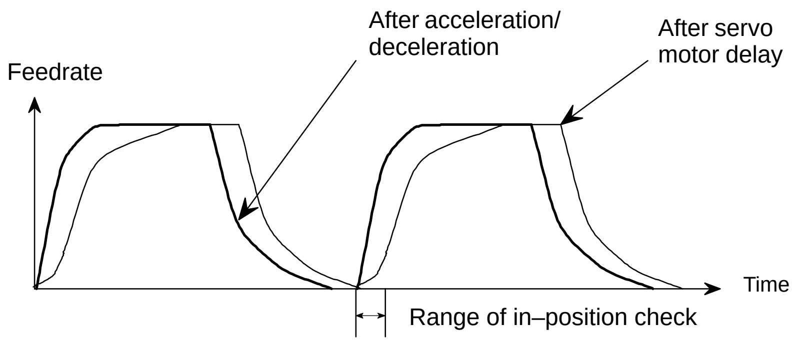
Note

NOTE

If the error detect signal is on, a cutting block is not executed until the acceleration/deceleration of the previous cutting block has been completed.

This function alone cannot prevent corner rounding due to delay caused by the servo motor, however.

To prevent corner rounding due to delay caused by the servo motor, use the in–position check function together with this function.



7.2.6 Feed Forward in Rapid Traverse

General

Feed–forward control can be performed even during rapid traverse. In this case, the servo position error is reduced, thus reducing the time required for positioning to within the specified range.

Parameter

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

[Data type] Bit

FFR Feed–forward control is enabled for

0 : Cutting feed only
1 : Cutting feed and rapid traverse

Reference item

For details of this function, refer to the “FANUC SERVO MOTOR α*i*s series Maintenance Manual (B–65285EN).”

8 AUXILIARY FUNCTION

8.1 MISCELLANEOUS FUNCTION/2ND AUXILIARY FUNCTION

General

- **Miscellaneous Function (M code)**

When an M code address is programmed, a code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. In some cases, however, up to three M codes can be specified for some types of machine tools (see 8.3 “Multiple M code per Block”)
Also, parameter No. 3030 can specify the maximum digits and if a specified value exceeds the maximum digits, an alarm may be issued.

- **2nd Auxiliary Function (B code)**

When up to eight digits are specified after address B, a code signal and strobe signal are sent. These signals are used to index the rotation axis of the machine. The code signal is retained until another B code is specified.

In each block, a single B code can be specified. The maximum number of digits that can be specified after address B is specified in parameter No. 3033. If more digits than the specified value are specified, an alarm occurs.
For the M series, the address for specifying the 2nd auxiliary function can be changed from B to another address (A, C, U, V, or W) by parameter setting (parameter No. 3460).

- **Command Format of 2nd Auxiliary Function**

· **Command range**

–99999999 to +99999999

· **Command method**

1. For the M series, a decimal point and a negative valve can be used for input by setting AUP (bit 0 of parameter No. 3450).
   With the T series, a decimal point and a negative value are always enabled regardless of the parameter setting.

Command	Output value
B10.	10000
B10	10

2. It is possible to change over the scale factor of B output, 1000 or 1 when the decimal point input is omitted, using the parameter DPI (No.3401#0).

Command		Output value
When DPI is 1:	B1	1000
When DPI is 0:	B1	1

3. It is possible to change over the scale factor of B output 1000 or 10000 when the decimal point input is omitted in the inch input system, using the parameter AUX (No.3405#0). When DPI=1.
   Command Output value
   When AUX is 1: B1 10000
   When AUX is 0: B1 1000

Basic procedure

The following signals are used with these functions. (For details of the spindle–speed function and tool function, see Chapters 9 and 10.)

Function	Program address	Output signal			Input signal
		Code signal	Strobe signal	Distribution end signal	Completion signal
Miscellaneous function	M	M00 to M31	MF	DEN	FIN
Spindle–speed function	S	S00 to S31	SF		
Tool function	T	T00 to T31	TF		
Secondary auxiliary function	B	B00 to B31	BF		

Each function uses different program addresses and different signals, but they all input and output signals in the same way, as described below. (A sample procedure for the miscellaneous function is described below. The procedures for the spindle–speed function, tool function, and secondary auxiliary function, are obtained simply by substituting S, T, or B in place of M.)

(1) Suppose that Mxxx is specified in a program.
   For xxx, the number of specifiable digits is specified in parameter Nos. 3030 to 3033 for each function. If more digits than the specified value are specified, an alarm occurs.

(2) Code signal M00 to M31 is sent to machine interface. After period TMF, specified in parameter No. 3010 (standard value: 16 msec), the strobe signal MF is set to 1. The code signal is the binary representation of the programmed value xxx.(*1) If a move, dwell, spindle–speed, or other function is specified in the same block as the miscellaneous function, the execution of the other function is started when the code signal of the miscellaneous function is sent.

(3) When the strobe signal is set to 1, the PMC reads the code signal and performs the corresponding operation.

(4) To execute an operation after the completion of the move, dwell or other function specified in the block, wait until distribution end signal DEN is set to 1.

(5) Upon completion of the operation, the PMC sets completion signal FIN to 1. The completion signal is used by the miscellaneous function, spindle–speed function, tool function, secondary auxiliary function, external operation function described later, and other functions. If any of these functions are executed simultaneously, the completion signal must be set to 1 upon completion of all the functions.

(6) If the completion signal remains set to 1 for longer than period TFIN, specified in parameter No. 3011 (standard value: 16 msec), the CNC sets the strobe signal to 0 and reports that the completion signal has been received.

(7) When the strobe signal is set to 0, set the completion signal to 0 in the PMC.

(8) When the completion signal is set to 0, the CNC sets all code signals to 0 and completes all sequences of the miscellaneous function.(*2)

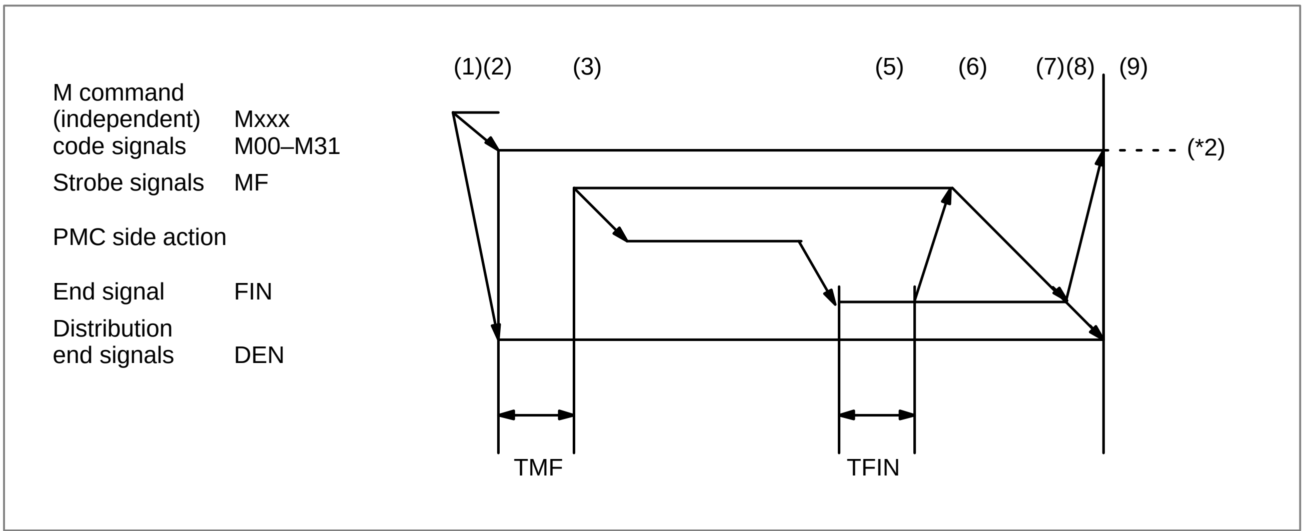
(9) Once all other commands in the same block have been completed, the CNC executes the next block.

*1 When the tool function is executed, the programmed tool number is sent as the code signal (T series).

*2 When the spindle–speed function, tool function, or secondary auxiliary function is executed, the code signal is maintained until a new code for the corresponding function is specified.

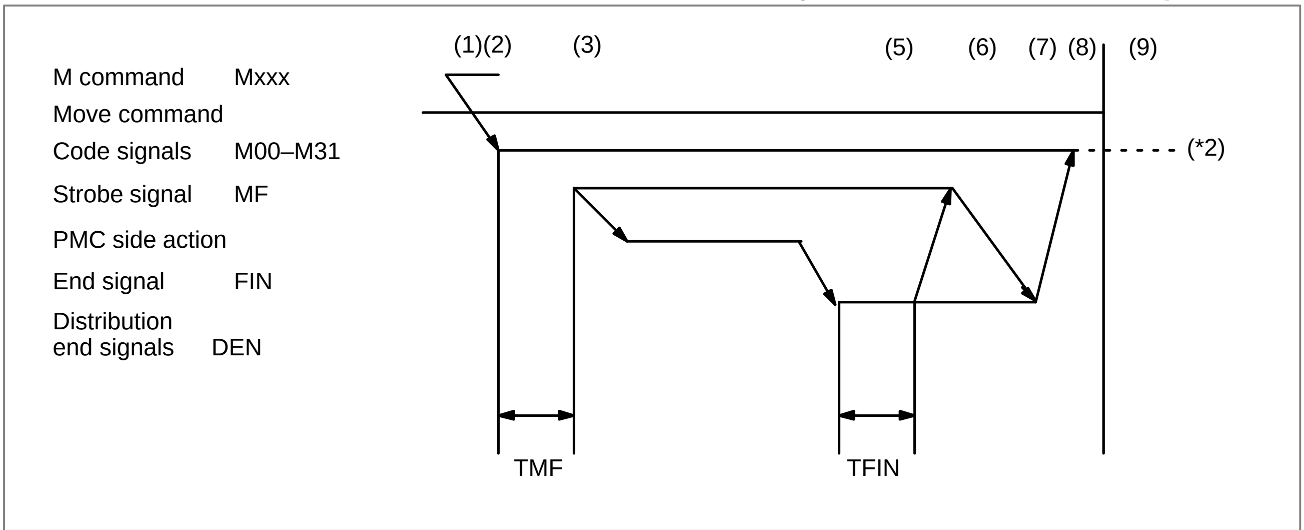
The timing diagram is shown below:

Example 1 Single miscellaneous function specified in a block

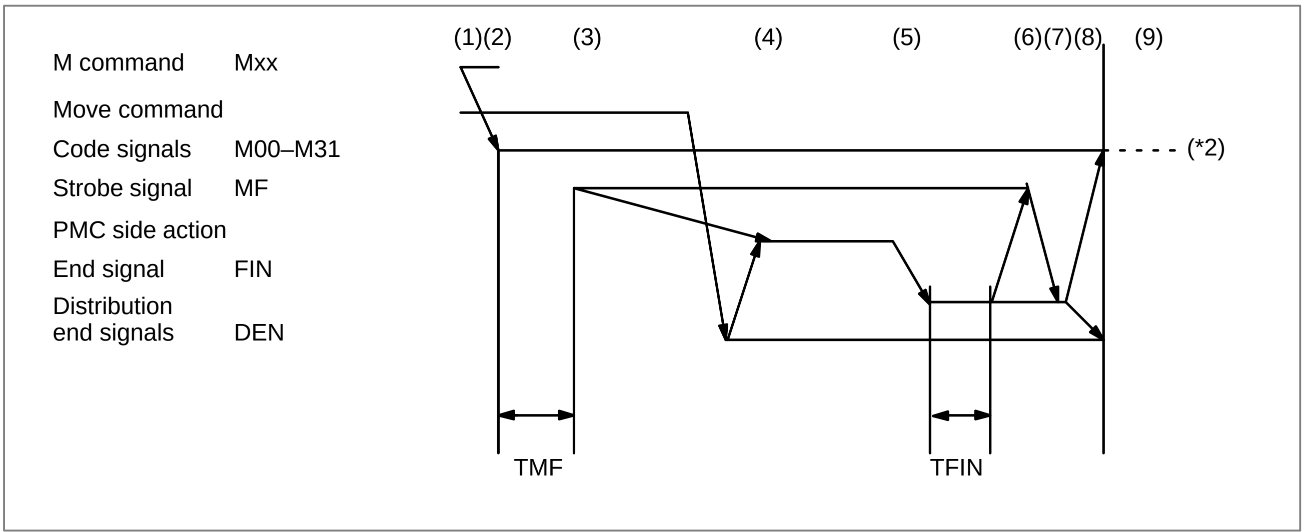


Example 2 Move command and miscellaneous function in the same block

2a. Execution of a miscellaneous function without waiting for move command completion



2b. Execution of a miscellaneous function after move command completion



Signal

Miscellaneous function code signals M00 to M31 <F010 to F013> Miscellaneous function strobe signal MF <F007#0>

[Classification] Output signal

[Function] These signals report the specification of miscellaneous functions.

[Output condition] For the output conditions and procedure, see the description of "Basic procedure" above.

NOTE

1 The following miscellaneous functions are only processed internally by the control unit; they are not output to the PMC when programmed:
   · M98, M99, M198
   · M code that calls a sub program (parameter No. 6071 to 6079)
   · M code that calls a custom macro (parameter No. 6080 to 6089)

2 Decode signals as well as the code signals and strobe signal are output for the miscellaneous functions listed below.
   M00, M01, M02, M30

Decode M signals DM00 <F009#7>, DM01 <F009#6>, DM02 <F009#5>, DM30 <F009#4>

[Classification] Output signal

[Function] These signals report particular miscellaneous functions are specified. The miscellaneous functions in a command program correspond to output signals as indicated below.

Command program	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

[Output condition] A decode M signal goes "1" when:

· The corresponding miscellaneous function is specified, and any move commands and dwell commands specified in the same block are completed. These signals are not output when the end signal of the miscellaneous function is returned before completion of such move commands and dwell commands.

A decode M signal goes "0" when:

· The FIN signal goes "1"

· Reset occurs

Spindle–speed code signals S00 to S31 <F022–F025> Spindle–speed strobe signal SF <F007#2>

[Classification] Output signal

[Function] These signals report that spindle speed functions have been specified.

[Output condition] For the output conditions and procedure, see the description of "Basic procedure" above.

Tool function code signals T00 to T31 <F026–F029> Tool function strobe signal TF <F007#3>

[Classification] Output signal

[Function] These signals report that tool functions have been specified.

[Output condition] For the output conditions and procedure, see the description of "Basic procedure" above.

Second auxiliary function code signals B00 to B31 <F030–F033> Second auxiliary function strobe signal BF <F007#4> (T series) <F007#7> (M series)

[Classification] Output signal

[Function] These signals report that second auxiliary functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

End signal FIN <G004#3>

[Classification] Input signal

[Function] This signal reports the completion of a miscellaneous function, spindle–speed function, tool function, second auxiliary function, or external operation function.

[Operation] For the control unit operation and procedure when this signal goes “1”, see the description of “Basic procedure” above.
The FIN signal must remain “1” for a certain time (TFIN, which is set by a parameter No. 3011) or longer. The FIN signal driven “1” is ignored if the FIN signal goes “0” before TFIN elapses.

WARNING
Only one end signal is used for all functions above. The end signal must go “1” after all functions are completed.

Distribution end signal DEN <F001#3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those miscellaneous functions, spindle–speed functions, 2nd auxilialy functions tool functions, and so forth that are contained in the same block and have been sent to the PMC. They also report that the end signal from the PMC is being awaited.

[Output condition] The DEN signal turns to “1” when:

· Waiting for the completion of miscellaneous functions, spindle–speed functions, tool functions, 2nd auxilialy functions and all other commands in the same block are completed, and the current position is in the in–position.

The DEN signal turns to “0” when:

· The execution of one block is completed

NOTE
A parameter NCI (No.1601#5) can specify, whether to only check if an acceleration/deceleration delay is eliminated, or to also check if a servo delay (error) has been reduced to within a certain range.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004					FIN			

	#7	#6	#5	#4	#3	#2	#1	#0
F001					DEN			
F007	BF			BF	TF	SF		MF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24

Parameter

3010	Time lag in strobe signals MF, SF, TF, and BF

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 to 32767

The time required to send strobe signals MF, SF, TF, and BF after the M, S, T, and B codes are sent, respectively.

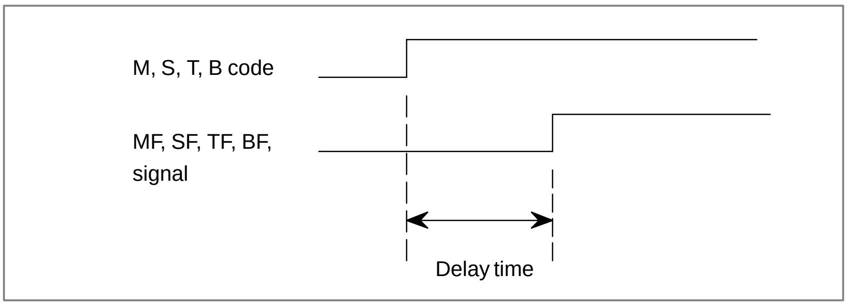


Fig. 8.1 (a) Delay time of the strobe signal

> **NOTE**
> The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.
>
> Example : When 30 is set, 32 ms is assumed.
> When 32 is set, 32 ms is assumed.
> When 100 is set, 104 ms is assumed.

3011	Acceptable width of M, S, T, and B function completion signal (FIN)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 to 32767

Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).

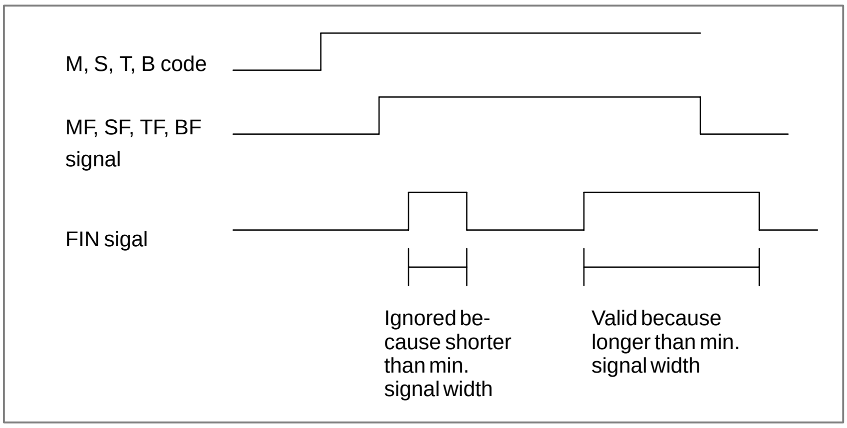


Fig. 8.1 (b) Valid Width of the FIN (M,S, T, and B Function Completion) Signal

> **NOTE**
> The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.
> Example: When 30 is set, 32 ms is assumed.

3030	Allowable number of digits for the M code
3031	Allowable number of digits for the S code
3032	Allowable number of digits for the T code
3033	Allowable number of digits for the B code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, T, and B codes.

> **NOTE**
> Up to 5 digits can be specified in the S code

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point

0 : The least input increment is assumed.
1 : The unit of mm, inches, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02	M30				

[Data type] Bit

M30 When M30 is specified in a memory operation:

0 : M30 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program will still be executed, starting from the beginning of the program.
1 : M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

M02 When M02 is specified in memory operation

0 : M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program will still be executed, starting from the beginning of the program.
1 : M02 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point

0 : Assumed to be 0.001
1 : Depending on the input increment. (For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

3411	M code preventing buffering 1
3412	M code preventing buffering 2
3413	M code preventing buffering 3
⋮	⋮
3420	M code preventing buffering 10

[Data type] Byte

[Valid data range] 0 to 255

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.

M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Minimum value 1 of M code preventing buffering
3422	Maximum value 1 of M code preventing buffering
3423	Minimum value 2 of M code preventing buffering
3424	Maximum value 2 of M code preventing buffering
3425	Minimum value 3 of M code preventing buffering
3426	Maximum value 3 of M code preventing buffering
3427	Minimum value 4 of M code preventing buffering
3428	Maximum value 4 of M code preventing buffering
3429	Minimum value 5 of M code preventing buffering
3430	Maximum value 5 of M code preventing buffering
3431	Minimum value 6 of M code preventing buffering
3432	Maximum value 6 of M code preventing buffering

[Data type] Word

[Valid data range] 0 to 65535

When a specified M code is within the range specified with parameter Nos. 3421 and 3422, 3433 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

CAUTION
1 The specification of a minimum value that exceeds the specified maximum value is invalid.
2 When there is only one data item, set the following: minimum value =maximum value.

	#7	#6	#5	#4	#3	#2	#1	#0
3450								
								AUP

AUP A second auxiliary function command, specified with a decimal point and a negative value is:
0 : Disabled.
1 : Enabled.

NOTE
With the T series, second auxiliary function commands specified with a decimal point and a negative value are always enabled regardless of the parameter setting.

3460	
	Name of a second auxiliary function

[Data type] Byte

Set the name of a second auxiliary function according to the table given below. Note that the same address as an axis name cannot be set.

Name	A	B	C	U	V	W
Setting	65	66	67	85	86	87

If a value other than those given above is set, address B is assumed.

Note

NOTE

1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:
   i) Simultaneous execution of the move command and miscellaneous function commands.
   ii) Executing miscellaneous function commands upon completion of move command execution.
   The selection of either sequence depends on the sequence of PMC.
2 The address used for specifying the 2nd auxiliary function (B or the address specified with parameter No. 3460) cannot be used as an axis address.
3 The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (No. 3411 to 3420).
4 For M00 and M01 only, miscellaneous function code signal, auxiliary function strobe signal, and M decode signals are sent; the control of program stop and optional stop shall be designed on the PMC side.
5 When the automatic operation is stopped by M02 or M30, it is necessary to send the external reset signal from the machine side to the CNC, instead of the FIN signal, When the external reset signal is returned against the M02 or M30, the control returns to the top of the program recently executed and enters the reset state. When the FIN signal is returned, the control returns to the beginning of the program recently executed and executes it from the top.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)

8.2 AUXILIARY FUNCTION LOCK

General

Inhibits execution of a specified M, S, T and B function.
That is, code signals and strobe signals are not issued.
This function is used to check a program.

Signal

Auxiliary function lock signal AFL <G005#6>

[Classification] Input signal

[Function] This signal selects auxiliary function lock. That is, this signal disables the execution of specified M, S, T, and B functions.

[Operation] When this signal turns to “1”, the control unit functions as described below.

(1) The control unit does not execute M, S, T, and B functions specified for memory operation, DNC operation, or MDI operation. That is, the control unit stops the output of code signals and strobe signals (MF, SF, TF, BF).

(2) If this signal turns to “1” after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FIN signal is received, and the strobe signal turns to “0”).

(3) Among the miscellaneous functions, M00, M01, M02, and M30 are executed even when this signal is “1”. All code signals, strobe signals, decode signals are output in the ordinary manner.

(4) Among the miscellaneous functions, even when this signal is “1”, those functions (M98 and M99) that are executed in the control unit without outputting their execution results are executed in the ordinary manner.

> **WARNING**
> Even when this signal is “1”, spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal MAFL <F004#4>

[Classification] Output signal

[Function] This signal reports the state of the auxiliary function lock signal AFL.

[Output condition] This signal turns to “1” when:

· The auxiliary function lock signal AFL is “1”

This signal turns to “0” when:

· The auxiliary function lock signal AFL is “0”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G005		AFL						

	#7	#6	#5	#4	#3	#2	#1	#0
F004				MAFL				

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	III.5.1	Machine lock and auxiliary function lock
	OPERATOR’S MANUAL (T series) (B–64114EN)	III.5.1	Machine lock and auxiliary function lock
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	III.5.1	Machine lock and auxiliary function lock
	OPERATOR’S MANUAL (T series) (B–64134EN)	III.5.1	Machine lock and auxiliary function lock

8.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK

General

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block.
Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

(Example)

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Y0Z0 ; : : :	M40M50M60 ; G28G91X0Y0Z0 ; : : : : :

Basic procedure

(1) Assume that "MaaMbbMcc;" was commanded by the program.

(2) The 1st M command (Maa) sends the code signals M00 to M31 in a manner similar to the conventional one-block single command. The strobe signal MF is set to "1" after a time TMF set by parameter No. 3010 (Standard setting: 16 msec).
The second M command (Mbb) sends the code signal M200-M215, the third M command (Mcc) sends the code signal M300-M315, and their respective strobe signals MF2 and MF3 are set to "1".
Furthermore, the three code signals are sent simultaneously.
The strobe signals MF, MF2, and MF3 become "1" at the same time.
The code signal is a binary notation of the program command aa, bb and cc.

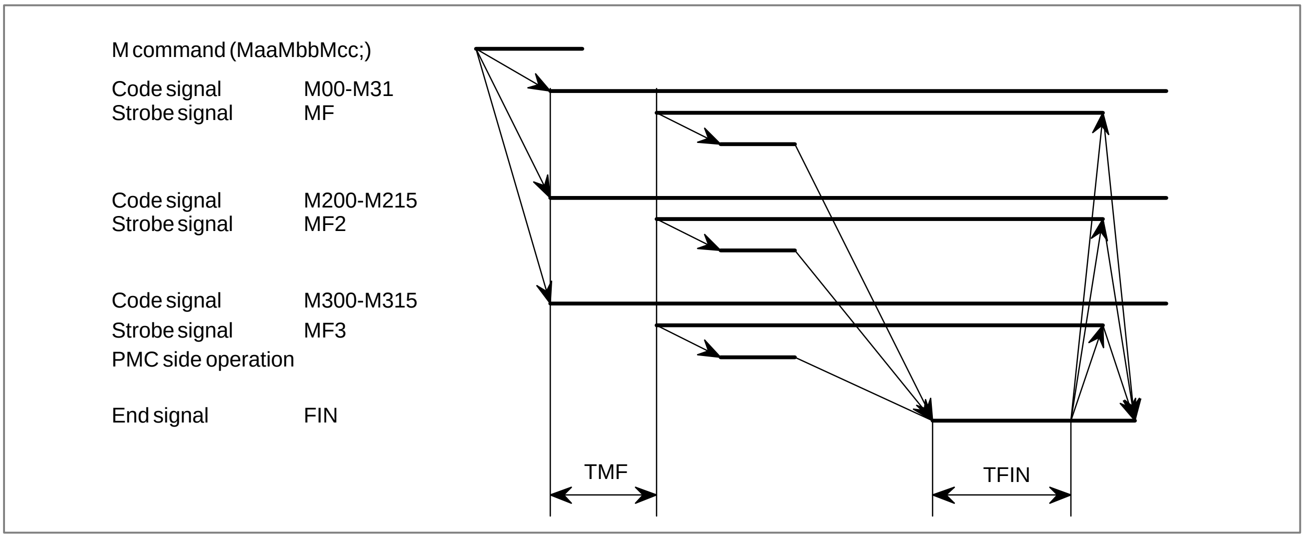
(3) On the PMC side, the code signals corresponding to the respective strobe signals are read when the strobe signals become "1", and the appropriate operations are performed.

(4) When the operation of all M commands ends on the PMC side, the end signal (FIN) is set to "1".

(5) When the completion signal stays "1" for a time (TFIN) set by parameter No. 3011 (Standard: 16 msec), all strobe signals (MF, MF2 and MF3) are set to "0" at the same time and the reception of completion signal is reported.

(6) On the PMC side, when MF, MF2 and MF3 are set to "0", the completion signal is set to "0".

A time chart for this procedure is shown below:



Signal

2nd, 3rd M function code signal M200 to M215 <F014, F015> M300 to M315 <F016, F017> 2nd, 3rd M Function strobe signal MF2 <F008#4> MF3 <F008#5>

[Classification] Output signal

[Function] Indicates that second and third auxiliary functions have been issued.

[Output condition] The output conditions and procedures are the same as that described in “Basic procedure”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008			MF3	MF2				
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308

Parameter

		#7	#6	#5	#4	#3	#2	#1	#0
3404		M3B							

[Data type] Bit

M3B The number of M codes that can be specified in one block

0 : One
1 : Up to three

Caution

CAUTION

1 M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.

2 Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block. Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the PMC. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks.
The M codes which can be specified in a single block must be those which the CNC sends the M code signals to the PMC side.

Note

NOTE

1 CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For example, M42 can be specified only after the mechanical operation of M41 is completed.

2 The 1st M code can be up to 8 digits and 2nd, 3rd M codes can be the values up to 65535.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.11.2	Multiple M commands in a single block
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.11.2	Multiple M commands in a single block
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.11.2	Multiple M commands in a single block
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.11.2	Multiple M commands in a single block

8.4 HIGH–SPEED M/S/T/B INTERFACE

General

To accelerate M/S/T/B function execution, the high–speed M/S/T/B interface has simplified the transfer of the strobe and completion signals of the M/S/T/B functions.

Whether to use the usual system or high–speed system for strobe signal and completion signal handling can be specified by parameter MHI (No. 3001#7).

The description below uses the miscellaneous functions (M code commands) as an example. The same description applies to the spindle–speed function (S code), tool function (T code) and 2nd auxiliary function (B code).

Basic procedure

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the NC system sends out the code signals M00 to M31.
The CNC system inverts the logical level of the strobe signal MF, that is, from “0”to “1”, or from “1” to “0”.

(3) The CNC system inverts the strobe signal, then when the logical level of the auxiliary function completion signal MFIN becomes the same as the strobe signal, the CNC assumes the completion of PMC sequence.

With the usual method, the operation is assumed to be completed when a falling edge (“1” to “0”) of the M/S/T/B completion signal FIN is received after a rising edge (“0” to “1”) of the FIN signal is detected. This new system, on the other hand, assumes the operation has been completed upon detection of only one transition of the completion signal.
In addition , the usual system uses only one completion signal (FIN) common to the M/S/T/B functions. This new system uses a different completion signal for each of the M, S, T, and B functions; the completion signals for the M, S, T, and B functions are MFIN, SFIN, TFIN, and BFIN, respectively.
The Fig. 8.4 (a) below shows the timing chart of these signals with the new system. For comparison, Fig. 8.4 (b) shows the timing chart of the conventional system.

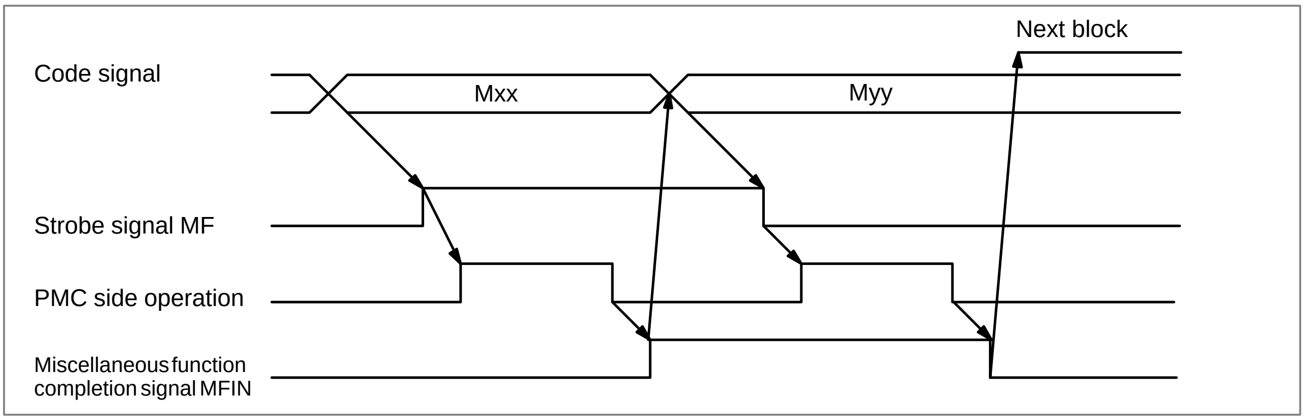


Fig. 8.4 (a) Timing chart of the high–speed system

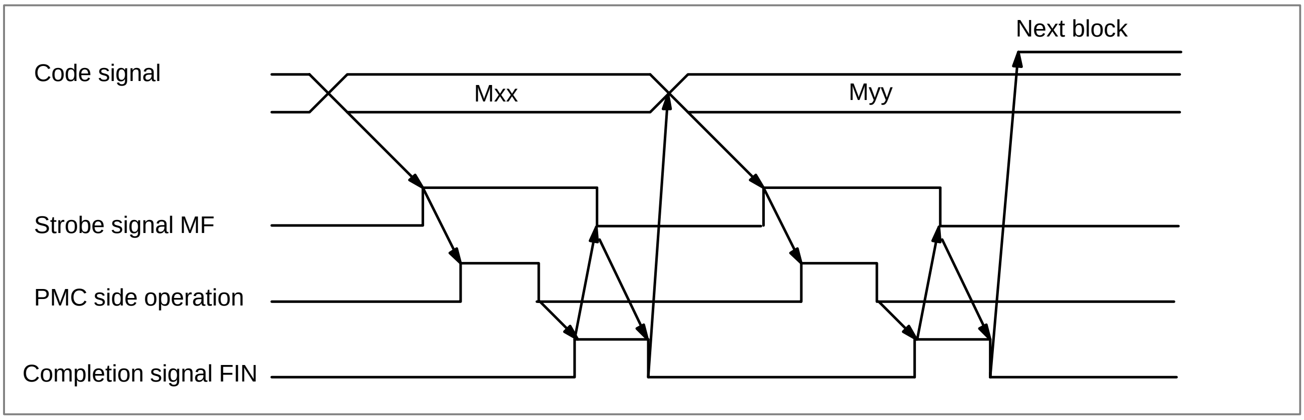


Fig. 8.4 (b) Timing chart of the usual system

A high–speed interface can also be used for multiple M commands issued for one block. This interface provides separate completion signals for each M code. They are called MFIN (the same name as for the single M command per block function), MFIN2, and MFIN3, respectively. The signal transfer sequence for multiple M codes per block is the same as that for a single M code per block.

The high–speed interface can also be used for the external operation function. In this case, special external operation signal EFD and completion signal EFIN are used. The procedure for sending and receiving these signals is identical to that for sending and receiving the strobe and completion signals of the miscellaneous function (M series).

Signal

Miscellaneous function completion signal MFIN <G005#0>

[Classification] Input signal

[Function] Reports that the execution of a miscellaneous function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and aprocedure of the contol unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

Spindle function completion signal SFIN <G005#2>

[Classification] Input signal

[Function] Reports that the execution of a spindle speed function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

Tool function completion signal TFIN <G005#3>

[Classification] Input signal

[Function] Reports that the execution of a tool function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

2nd auxiliary function completion signal BFIN <G005#4> (T series) <G005#7> (M series)

[Classification] Input signal

[Function] Reports that the execution of a second auxiliary function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

2nd, 3rd M function completion signals MFIN2, MFIN3 <G004#4, #5>

[Classification] Input signal

[Function] Indicate that when the high–speed interface is used for multiple M commands per block, the second to 3rd M functions have been completed.

[Operation] See “Basic procedure” for how the control unit operates and what it performs when the signal turns to “1” and “0”.

External operation signal for high–speed interface (M series) EFD <F007#1>

[Classification] Output signal

[Function] Indicates that positioning for the external operation function has been completed for the high–speed M, S, T, or B interface, and that another external operation is required.

[Operation] Refer to the description of the output conditions and procedure described in “basic procedure.”

External operation function completion signal (M series) EFIN <G005#1>

[Classification] Input signal

[Function] Indicates that the external operation function has been completed for the high–speed M, S, T, or B interface.

[Operation] The “basic procedure” describes the procedure and operation of the control unit when the signal is set to 1 or 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004			MFIN3	MFIN2				
G005	BFIN			BFIN	TFIN	SFIN	EFIN	MFIN

	#7	#6	#5	#4	#3	#2	#1	#0
F007							EFD	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

MHI Exchange of strobe and completion signals for the M, S, T, and B codes

0 : Normal
1 : High–speed

Note

NOTE
1 The strobe signals MF, SF, TF, and BF are “0” when the power is turned on.
2 When the control unit is reset, MF, SF, TF, and BF are set to “0”.

Reference item

CONNECTION MANUAL (This manual)	8.1	Miscellaneous function/2nd auxiliary function
	8.3	Multiple M commands in a block
	11.7	External operation function

9 SPINDLE SPEED FUNCTION

9.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

General

When up to five digits are specified after address S, code and strobe signals are sent out and used to control the spindle speed. The code signals are retained until another S code is issued.

One S code is used for each block. Parameter No. 3031 can be used to specify the maximum number of digits. If a number greater than the maximum number is specified, an alarm can be raised.

Signal

Refer to section 8.1.

Parameter

3031	Allowable number of digits for the S code

[Data type] Byte

[Valid data range] 1 to 5

Set the allowable numbers of digits for the S codes.

Note

NOTE

1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:
   i) Simultaneous execution of the move command and miscellaneous function commands.
   ii) Executing miscellaneous function commands upon completion of move command execution.

   The selection of either sequence depends on the PMC processing sequence.

2 For S code output when the spindle serial output/spindle analog output is used, refer to section 9.3.

9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT

General

There are two types of spindle motor control interfaces, spindle serial output and spindle analog output.

The spindle serial output interface can control up to two serial spindles. The spindle analog output interface can control one analog spindle.

The table below lists the relationships between the spindle control interfaces and the configuration of the spindle.

Spindle serial output	Spindle analog output	First spindle	Second spindle	Third spindle
○	○	First serial spindle The PC can be used.	—	Analog spindle The PC cannot be used.
○	×	First serial spindle The PC can be used.	Second serial spindle The PC can be used. (*)	—
×	○	Analog spindle The PC can be used.	—	—
×	×	See Section 9.1. ⇒ Controlled by the PMC using an external interface.		

· PC = position coder

· (*) The multispindle function is necessary to use the position coder on the second spindle.

· See section 15.4 or 9.9 for how to control the speed of the second and third spindles.

The table below lists the relationship between the spindles and functions.

○=Available ×=Unavailable

Function \ Spindle	Serial spindle		Analog spindle	
	First serial spindle	Second serial spindle	When used as the first spindle (with no serial SP)	When used as the third spindle (with a serial SP)
Thread cutting/feed per revolution (synchronous feed)	○	○ (*1)	○	×
Constant surface speed control	○	○ (*1)	○	×
Spindle speed fluctuation detection	○	○ (*1)	○	×
Actual spindle speed output (T series)	○	○ (*1)	○	×
Spindle positioning (T series)	○	×	○	×
Cs contour control	○	×	×	×
Multi–spindle (*2)	○ (First spindle)	○ (Second spindle)	×	○ (Third spindle)
Rigid tapping	○	○ (*1)	○	×
Spindle synchronous control	○ Master	○ Slave	×	×
Spindle control unit functions, such as spindle orientation, spindle output switching, spindle switching, and etc. (*3)	○	○	○	○
Polygon turning (T series) (using the servo motor axis and spindle)	○	○ (*1)	○	×
Spindle output control by the PMC	○	○	○	○

NOTE

1 The multispindle function is necessary. The function cannot be used for the first and second spindles simultaneously.
2 The multispindle function can control the speed of three spindles and switch the feedback signal between two position coders. It also can work without the second or third spindle.
3 These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports those functions.

The signals and parameters for spindle speed control are common to both spindle serial output and spindle analog output. (See Section 9.3.)

The table below lists the differences related to direct control of the spindle control unit.

	Spindle control unit for spindle serial output interface	Spindle control unit for spindle analog output interface
Parameters for the spindle control unit	Specified as CNC parameters (4000 to 4351/S1, S2) Used after being transferred to the spindle control unit	Directly specified for the spindle control unit
Control signal for the spindle control unit	Connected to the PMC via the CNC G0070 to G0073 and F0045 to F0048: Addresses for the first spindle G0074 to G0077 and F0049 to F0052: Addresses for the second spindle	Connected to the PMC via an external contact
Spindle speed command interface	Digital data in a range from 0 to ± maximum spindle motor speed	Analog voltage from 0 to ±10 V (excluding portion for offset voltage adjustment)
Position coder interface	Connected to the CNC via the spindle control unit	Connected directly to the CNC

Signal

· Spindle control unit signals for the serial spindle

<G0070 to G0073> (input), <F0045 to F0048> (output)
→ for the first serial spindle
<G0074 to G0077> (input), <F0049 to F0052> (output)
→ for the second serial spindle

These addresses are on the CNC. Actually, however, they are input/output signals for the spindle control unit for the serial spindle.

For details of the signals belonging to these addresses, refer to the manuals for the serial spindle:

FANUC SERVO AMPLIFIER α*i* series Descriptions (B–65282EN)
FANUC AC SPINDLE MOTOR α*i* series Descriptions (B–65272EN)
FANUC AC SPINDLE MOTOR α*i* series Parameter Manual (B–65280EN)

Signal address

• For 1st SERIAL SPINDLE

	#7	#6	#5	#4	#3	#2	#1	#0
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072	RCHHGA	MFNHGA	INCMDA	OVRIDA	DEFMDA	NRROA	ROTAA	INDXA
G073						MPOFA	SLVA	MORCMA

	#7	#6	#5	#4	#3	#2	#1	#0
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
F047							INCSTA	PC1DEA
F048								

• For 2ND SERIAL SPINDLE

	#7	#6	#5	#4	#3	#2	#1	#0
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1A	CTH2B	TLMHB	TLMLB
G075	RCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076	RCHHGB	MFNHGB	INCMDB	OVRIDB	DEFMDB	NRROB	ROTAB	INDXB
G077						MPOFB	SLVB	MORCMB

	#7	#6	#5	#4	#3	#2	#1	#0
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
F051							INCSTB	PC1DEB
F052								

Parameter

- **Connection of serial spindle control unit**

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2			ISI	

[Data type] Bit

ISI Specifieds whether the serial spindle interface is used.

0 : Used
1 : Not used

> **NOTE**
> 1 This parameter is enabled only when the serial spindle interface is provided. The parameter is used when the CNC is started after serial spindle interface control is temporarily disabled during startup adjustment of the CNC. This bit should normally set to be 0.
> 2 When Serial spindle is used with Analog spindle and this parameter is set to “1”, Analog spindle becomes 1st spindle.

SS2 The number of connections in serial spindle control

0 : 1
1 : 2

> **NOTE**
> To connect two serial spindles, set jumper S1 on the 1st serial spindle control unit to B.
> (For S series SERIAL SPINDLE AMPLIFIER)

- **Parameters of serial spindle control unit**

No. 4000 to 4351: S1 → For 1st serieal spindle
S2 → For 2nd serial spindle

The above parameters are on the CNC, but actually they are used for the spindle control unit of serial spindle.
For details of these parameters, refer to the following manual:

FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS MANUAL (B–65272EN)

FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL (B–65280EN)

Alarm and message

Number	Message	Contents
749	S–SPINDLE LSI ERROR	It is serial communication error while system is executing after power supply on. Following reasons can be considered. 1) Optical cable connection is fault or cable is not connected or cable is cut. 2) Motherboard (LCD–mounted type CNC) or MAIN CPU board (stand–alone type CNC) board is fault. 3) Spindle amp. printed board is fault. 4) The spindle amplifier is under an abnormal condition. (The SPM indication is A, A1, A2, or the like, depending on the type of the abnormality.) If this alarm occurs when CNC power supply is turned on or when this alarm can not be cleared even if CNC is reset, turn off the power supply also turn off the power supply in spindle side. If the spindle amplifier is under an abnormal condition, check the SPM indication (A, A1, A2, or the like). Then, refer to the FANUC SERVO MOTOR αi series MAINTENANCE MANUAL (B–65285EN) to solve the problem.
750	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated. 4) The second spindle (when SP2, bit 4 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). See diagnostic display No. 409 for details.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
754	SPINDLE–1 ABNORMAL TORQUE ALM	Abnormal first spindle motor load has been detected.
762	SECOND SPINDLE MODE CHANGE FAULT	Refer to alarm No. 752.(For 2nd axis)
764	SPINDLE–2 ABNORMAL TORQUE ALM	Same as alarm No. 754 (for the second spindle)

DIAGNOSIS SCREEN

- **Information on spindle control**

	#7	#6	#5	#4	#3	#2	#1	#0
400				SAI	SS2	SSR	POS	SIC

SIC 0: No module is present for spindle serial output.
1: A module for spindle serial output is present.

POS 0: No module is present for spindle analog output.
1: A module for spindle analog output is present.

SSR 0: Spindle serial output is not used.
1: Spindle serial output is used.

SS2 0: The second spindle is not used with spindle serial output.
1: The second spindle is used with spindle serial output.

SAI 0: Spindle analog output is not used.
1: Spindle analog output is used.

401	Alarm condition for the first serial spindle (AL–??)
402	Alarm condition for the first serial spindle (AL–??)

- **Communication error on spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
408	SSA		SCA	CME	CER	SNE	FRE	CRE

CRE 1 : CRC error (warning)

FRE 1 : Framing error (warning)

SNE 1 : Mismatch between sending and receiving sections

CER 1 : Abnormal reception

CME 1 : No answer during auto scanning

SCA 1 : Communication error in the spindle amplifier

SSA 1 : System error in the spindle amplifier

(These errors are reflected in spindle alarm 749. They are caused by noise, disconnection, or instantaneous power interruption.)

- **Information related to the activation of the spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SHE 1 : Abnormal operation in the serial spindle communication module of the CNC

S1E 1 : Abnormal operation on the first spindle during activation

S2E 1 : Abnormal operation on the second spindle during activation

SPE 1 : Serial spindle parameter not meeting activation conditions

(These errors are reflected in spindle alarm 750.)

- **Load and speed meter readings for the serial spindle**

410	First serial spindle: Load meter reading (%)
411	First serial spindle: Speed meter reading (min^{-1})
412	Second serial spindle: Load meter reading (%)
413	Second serial spindle: Speed meter reading (min^{-1})

To display the load and speed meter readings, the following parameters must be specified correctly.

Maximum motor speed: Parameter No. 4020 (main) and 4196 (sub)

Load meter reading at maximum output:
Parameter No. 4127 (main) and 4276 (sub)

> **NOTE**
> The spindle switch function is used for main/sub switching. Select main if the spindle switch function is not used.

- **Position error display during spindle synchronization**

414	Master spindle motion error during spindle synchronization
415	Slave spindle motion error during spindle synchronization
416	Absolute value of synchronization error during spindle synchronization

The display for diagnosis No. 414 to 416 are in pulse units (one pulse = 360/4096 degrees)

- **Position error display during spindle synchronization**

417	First serial spindle: Position coder feedback information
418	First serial spindle: Position error
419	Second serial spindle: Position coder feedback information
420	Second serial spindle: Position error

The above display data is the information obtained directly from the serial spindle control unit.

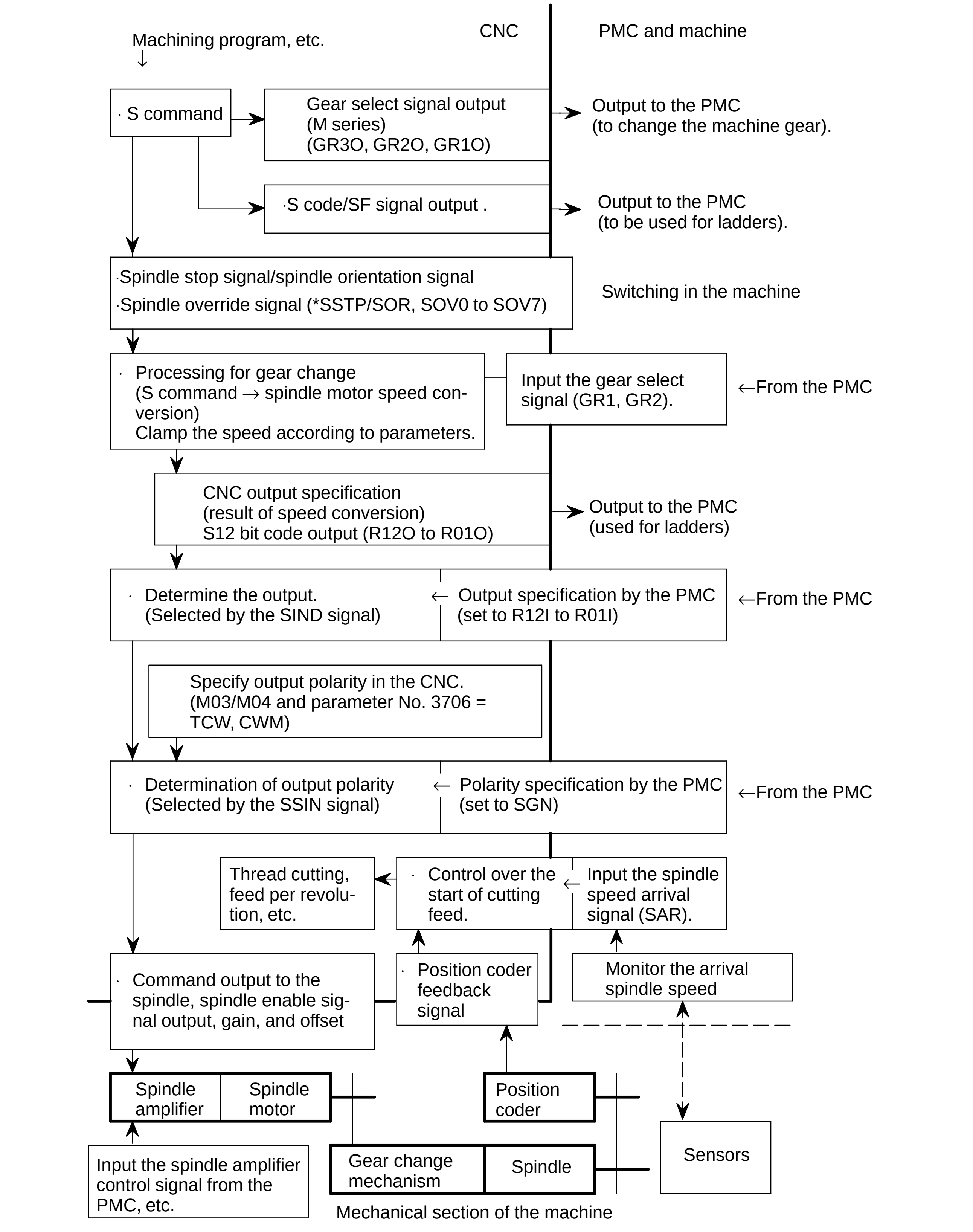
9.3 SPINDLE SPEED CONTROL

General

This section describes spindle speed control. It also explains the position coder and the spindle speed arrival signal (SAR).

Command flow of spindle speed control

The following chart summarizes spindle speed control.



- **S command**

The S command specifies the spindle speed entered from machining programs, etc. for the CNC.

For constant surface speed control (during G96 mode), the CNC converts the specified surface speed to the spindle speed.

In the M series with bit 4 (GTT) of parameter No. 3706 = 0 without the constant surface speed control, the CNC specifies the gear stage for the desired spindle speed to the PMC according to parameter No. 3741, 3742, and 3743, and the S command.

(GR3O, GR2O, GR1O <F034#2, #1, #0>)

- **S code/SF signal output**

With the spindle serial output or spindle analog output, the spindle control function in the CNC converts the S command value to the output value for the spindle motor. In addition, the S code/SF signal's response to S command input changes as follows, compared with the case where gear change and constant surface speed control are not used.

M series → Outputs the S code.
The SF signal is output only when the CNC directs the PMC to change the gear.

T series → Outputs neither S code nor SF signal.
(This is because the S code is not always the spindle speed when the constant surface speed control option is used.)

If you use the S code for processing in the PMC ladder, you must specify parameters related to parameter No. 3705.

- **Spindle stop signal (*SSTP)**

This signal sets the S command value in the CNC to 0. If the CNC is specifying the spindle output (see descriptions on the SIND signal), this signal sets the speed command for the spindle to 0.

Even if the spindle stop signal is not used, the signal must be set to logical 1 for the CNC to perform spindle speed control.

- **Spindle orientation signal (SOR)**

If the spindle orientation signal is logical 1 and the spindle stop signal is logical 0, the spindle rotates in the direction specified by bit 5 (ORM) of parameter No. 3706 at a constant speed specified by parameter No. 3732.

Because the spindle rotates at a constant speed regardless of the gear stage, this signal can be used to rotate the spindle to drive the stopper or pin during mechanical spindle positioning.

For the M series, setting parameter GST (bit 1 of parameter No. 3705) enables the spindle motor to rotate at a constant speed at an M type gear shift. This function can be used for gear shifting because it maintains a constant speed of the gear change mechanism.

- **Spindle speed override signal (SOV00 to SOV07)**

This signal specifies an override of 0% to 254% for the specified S value for spindle control.

However, the spindle speed override function is disabled when the CNC is in the following state:

Tapping cycle (M series : G84, G74 T series : G84, G88)
Thread cutting (M series : G33 T series : G32, G92, and G76)

When the spindle speed control is performed but the spindle speed override is not used, set the override value to 100%.

- **Processing for gear changing**

Although the S command contains the spindle speed, the object that is actually controlled is the spindle motor. Therefore, the CNC must have some provision to detect the gear stage between the speed and spindle motor.

There are two types of gear selection methods:

<u>M type</u>

The CNC selects a gear stage according to the range of speed for each gear stage previously specified in a parameter, as directed by the S command, and informs the PMC of the selected gear stage (one of the three gear stages) using the gear select signal output (GR3O, GR2O, GR1O).

Also, the CNC outputs the spindle speed based on the selected gear stage (output as the gear select signal).

<u>T type</u>

The gear stage (one of the four gear stages) being currently used by the machine is specified by the gear select signal inputs (GR1, GR2).

The machine determines which gear to use.

The CNC outputs the appropriate speed command for the selected gear range.

- **Selection of gear change system**

The M series system can use either M or T type.

M type ← Without constant surface speed control option, and bit 4 (GTT) of parameter No. 3706 = 0

T type ← With constant surface speed control, or bit 4 (GTT) of parameter No. 3706 = 1

The T series system can use only T type.

- **Details of M type (Output of GR1O, GR2O, GR3O)**

By specifying from S0 to S99999 in memory or MDI operation, the CNC outputs a command corresponding to the spindle speed. There is a two-speed (GR1O and GR2O) or three-speed range (GR1O, GR2O, GR3O), set by parameter nos. 3741-3743, and the gear selection signal is output simultaneously. When the gear selection signal is changed, the SF signal is output at the same time (parameter SFA no. 3705#6).
The meaning of the gear signals is shown below:

	Gear 2–stage	Gear 3–state	Remarks
GR1O	Low	Low	Low: Low Gear Middle: Middle Gear High: High Gear
GR2O	High	Middle	
GR3O		High	

The speed commands output to the spindle motor are as follows:

· For the serial spindle, the speed commands are processed as values 0 to 16383 between the CNC and spindle control unit.

· For the analog spindle, the speed commands are output to the analog voltage signal SVC as analog voltages 0 to 10 V.

The following descriptions exemplify the analog spindle. However, they can be applied to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.

- M type gear change method A (Fig. 9.3 (a))

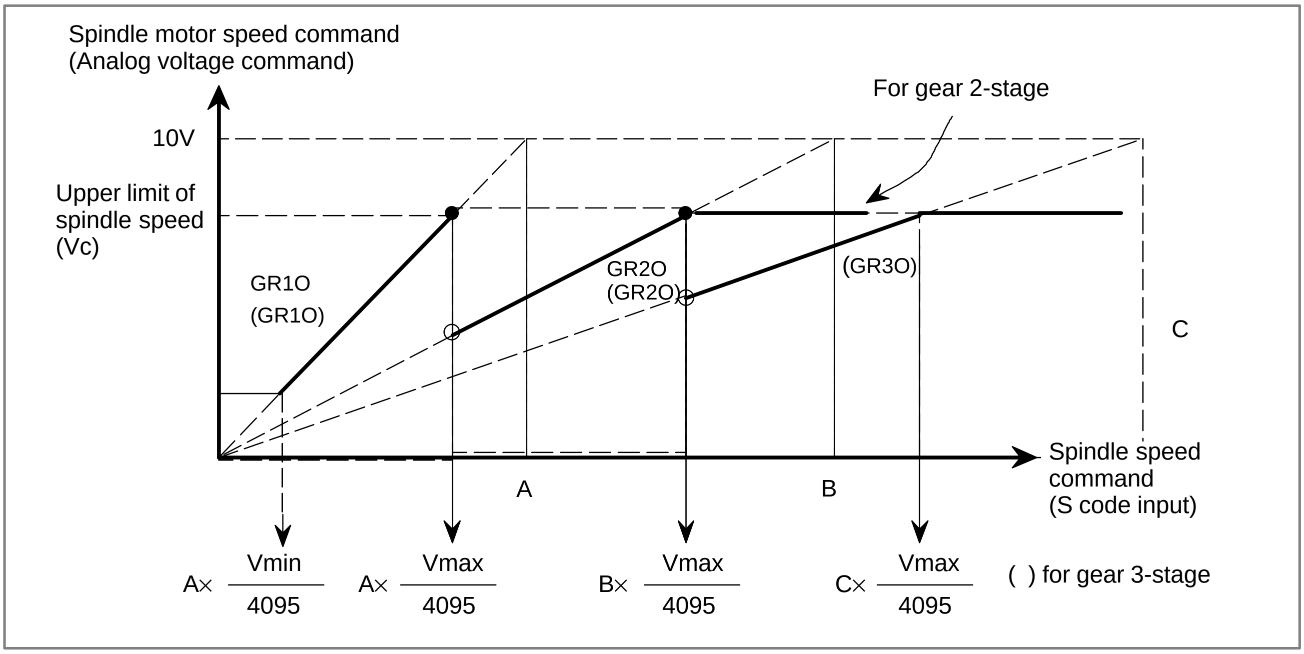


Fig. 9.3 (a) S code input and output voltage

Set the following values as parameters:

· Constant Vmax: for upper limit of spindle speed (parameter No. 3736)

$$\text{Vmax}=4095\times\frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Constant Vmin; for lower limit of spindle speed (parameter No. 3735)

$$\text{Vmin}=4095\times\frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Spindle speed A (min^{-1}) ; at command voltage 10V and low gear (parameter no. 3741)

· Spindle speed B (min^{-1}) ; at command voltage 10V and high gear (or middle-high gear) (parameter no. 3742)

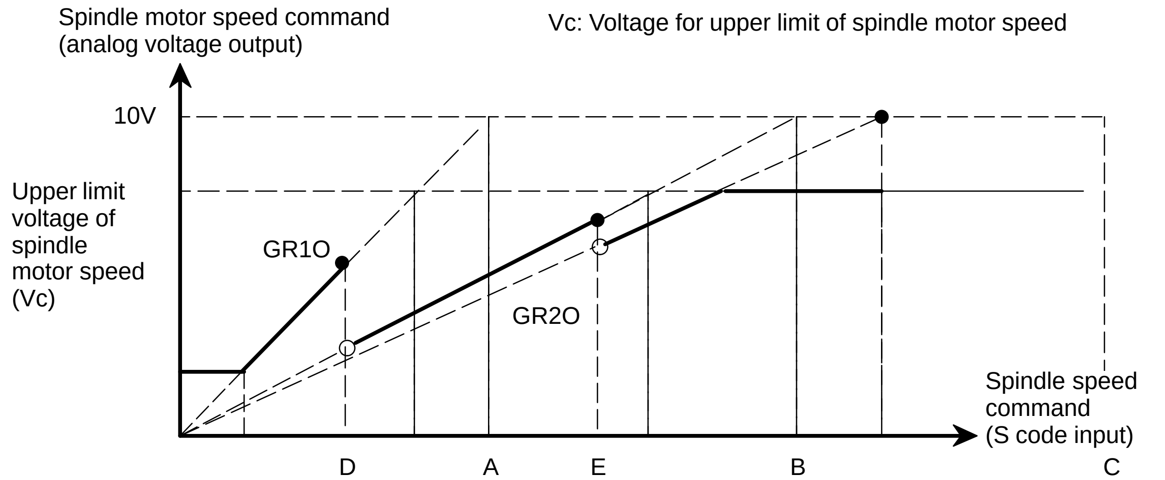
· Spindle speed C (min^{-1}) ; at command voltage 10V and high gear (parameter no. 3743)

NOTE

If a specified voltage of 10 V is already higher than the acceptable input voltage for the spindle drive system, calculate the spindle speed that corresponds to 10 V using a proportional calculation method and use it instead. Now, in response to the specified S code, the speed command and gear select commands (GR3O, GR2O, GR1O) are output to the spindle motor as shown in Fig. 9.3. (a).

- Gear change point during tapping cycle mode (G84, G74)

In case of G84 (tapping cycle) or G74 (counter tapping cycle) the gear shift speed is changed by parameter SGT(No. 3705#3). In this case, gear shift is performed at the speed set by parameter nos. 3761 and 3762 (Fig. 9.3 (b)).



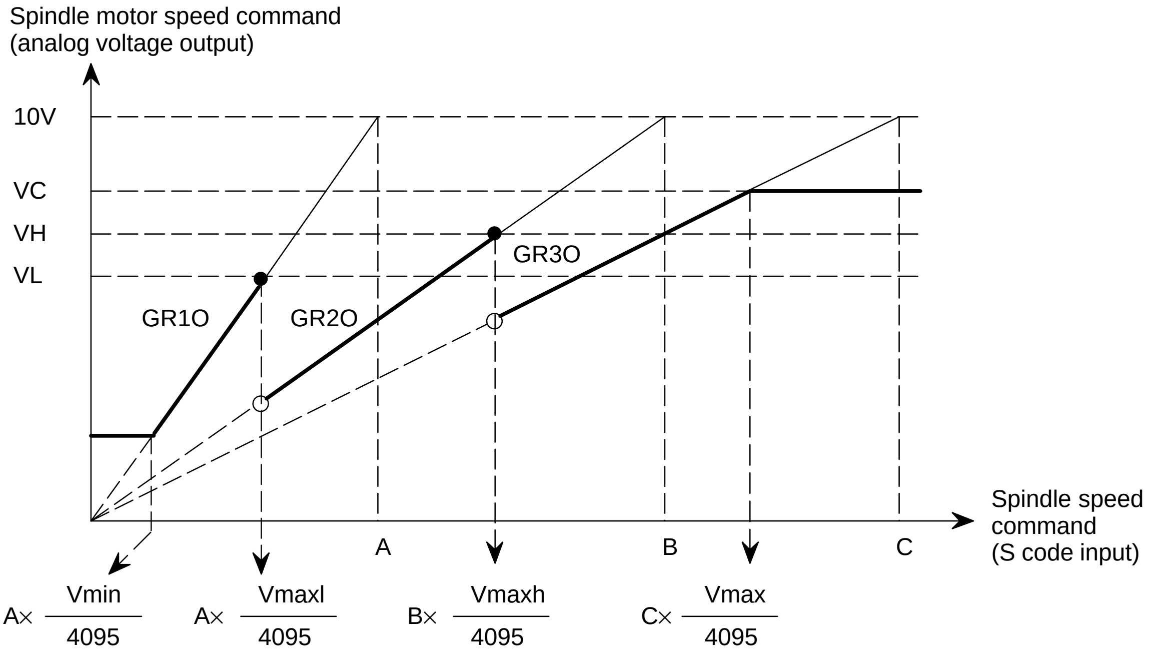
D : Gear shift point (parameter No. 3761) for low/high speed change (or low/medium speed change) (set by parameter as min^{-1} value) during tapping.

E : Gear shift point (parameter No. 3762) (3-stage gear) for medium/high speed change (set by parameter as min^{-1} value) during tapping.

Fig. 9.3 (b) S code input and output voltage (in tapping)

- M type gear change method B (M series) (Fig. 9.3 (c))

The speed (min^{-1}) at which the low-speed and the high-speed gears are changed over can be set as a parameter (No.3751, 3752) by setting parameter SGB (No. 3705# 2). When a 3-stage gear is used, it is possible to set the speeds (min^{-1}) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters No. 3751, 3752.



VC: Voltage corresponding to the upper limit of output value to spindle motor.

VH: Voltage corresponding to the upper limit of output value to spindle motor with high-speed gears (medium speed gear for 3-stage gear)

VL: Voltage corresponding to the upper limit of output value to spindle motor at low-speed gear

Fig. 9.3 (c) M type gear change B

When using this function, set the following parameters:

· Constant Vmax (Parameter No.3736) related to the upper limit of spindle motor speed (min^{-1})

$$\text{Vmax} = 4095\times \frac{\text{Upper limit of spindle motor speed (min}^{-1}\text{)}}{\text{Spindle motor speed (min}^{-1}\text{) when the command voltage is 10V}}$$

· Constant Vmin (Parameter No. 3735) related to the lower limit of spindle motor speed (min^{-1})

$$\text{Vmin} = 4095\times \frac{\text{Lower limit of spindle motor speed (min}^{-1}\text{)}}{\text{Spindle motor speed (min}^{-1}\text{) when the command voltage is 10V}}$$

· Constant Vmaxl (Parameter No. 3751) related to the upper limit of spindle motor speed (min^{-1}) with low-speed gears

$$\text{Vmaxl} = 4095\times \frac{\text{Upper limit of spindle motor speed (min}^{-1}\text{) with low-speed gears}}{\text{Spindle motor speed (min}^{-1}\text{) when the command voltage is 10V}}$$

· Constant Vmaxh (Parameter No. 3752) related to the upper limit of spindle motor speed (min^{-1}) with high-speed gears (medium-speed gear for 3-stage gear)

$$\text{Vmaxh} = 4095\times \frac{\text{Upper limit of spindle motor speed (min}^{-1}\text{) with high-speed gears (medium-speed gear for 3-stage gear)}}{\text{Spindle motor speed (min}^{-1}\text{) when the command voltage is 10V}}$$

· Spindle speed A (Parameter No.3741) (min^{-1}) with low-speed gears when the command voltage is 10V

· Spindle speed B (Parameter No.3742) (min^{-1}) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-stage)

· Spindle speed C (Parameter No.3743) (min^{-1}) with high-speed gears when the command voltage is 10V (3-stage gear)

Spindle motor speed commands (0 to 10V) and gear selecting signals (GR1O, GR2O, GR3O) are issued on each S code command as shown in the figure:

CAUTION

1 In a tapping cycle when parameter SGT (No. 3705 #3) is set, the gears are changed over at the gear changing point for tapping.

2 For this function (parameter SGB=1 (No. 3705#2)), when only one-stage gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-stage gear is used, it is calculated according to Vmaxh. Therefore, when SGB is 1, set Vmaxl when only one-stage gear is used, Vmaxl and Vmaxh when 2-stage gear is used.

- **Time chart**

When S code is commanded, the I/O signal time chart is :

- When Gear select signal does not change

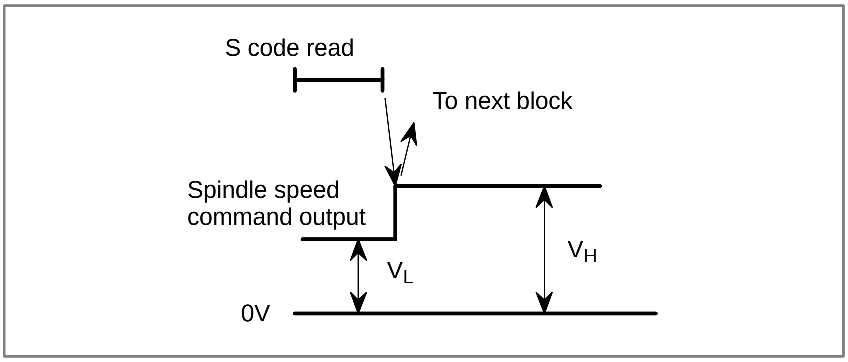


Fig. 9.3 (d) Time chart when gear select signal does not change

In this case, the SF signal is not output and the CNC advances to the next block automatically after the next spindle speed command is output.

- When Gear select signal change

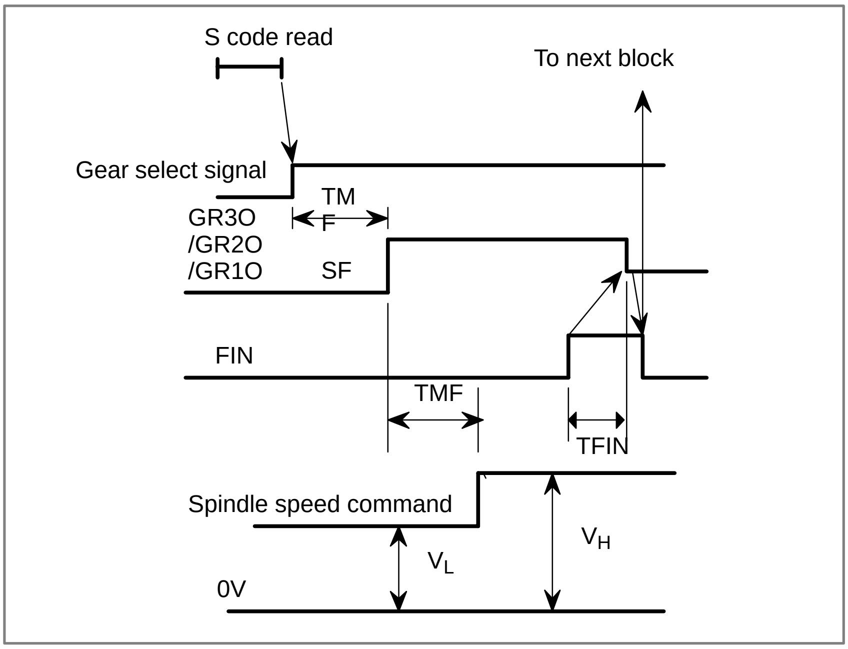


Fig. 9.3 (e) Time chart when gear select signal changes

In this case, the gear select signal is output; after elapse of the time constant set by parameter (TMF), the SF signal is output. After another TMF elapse, the spindle speed command is output. On the PMC side, change the gears by this signal, and return the FIN signal after the end of gear change. The time chart for SF and FIN signals is the same as in S code output.TMF, set by parameter No. 3010, is common to M, S and T functions.

Moreover, specifying bit 6 (SFA) of parameter No. 3705 can specify that the SF signal be output even if no gear change is used.

- **Details of T type (Input of GR1, GR2)**

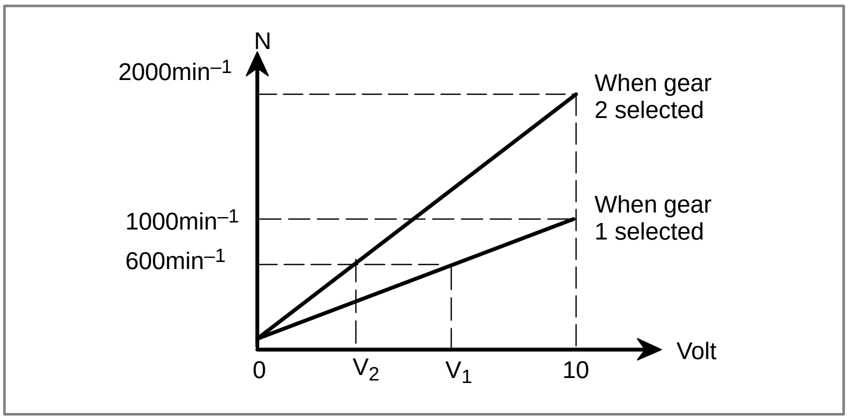
To perform the T type gear change, the maximum spindle speed for each gear side must be set in parameter No. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear No.	Parameter No. for max. spindle speed
0	0	1	No. 3741
1	0	2	No. 3742
0	1	3	No. 3743
1	1	4	No. 3744

The following descriptions apply to the analog spindle. Like the descriptions of the M type, they also apply to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.

In addition, for the speed command output to the spindle motor, analog voltages 0 to 10 V for analog spindle control correspond to digital data 0 to 16383 for serial spindle control. However, it might be easier if you consider them code signals from 0 to 4095 for convenience sake without distinguishing between serial and analog spindles.

Assume that gear switching is two stage switching. If the spindle speed with the output voltage 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds by the parameter No. 3741, 3742. In this case, the analog voltage has the linear relationship shown below.



When spindle speed S=600 is given, V_1 (for G1) or V_2 (for G2) is calculated inside the CNC and output to the machine side.

V_1: 6(V)
V_2: 3(V)

The value of output voltage V is calculated automatically from the following equations:

$$V = \frac{10N}{R}$$

R: Spindle speed at 10V output voltage

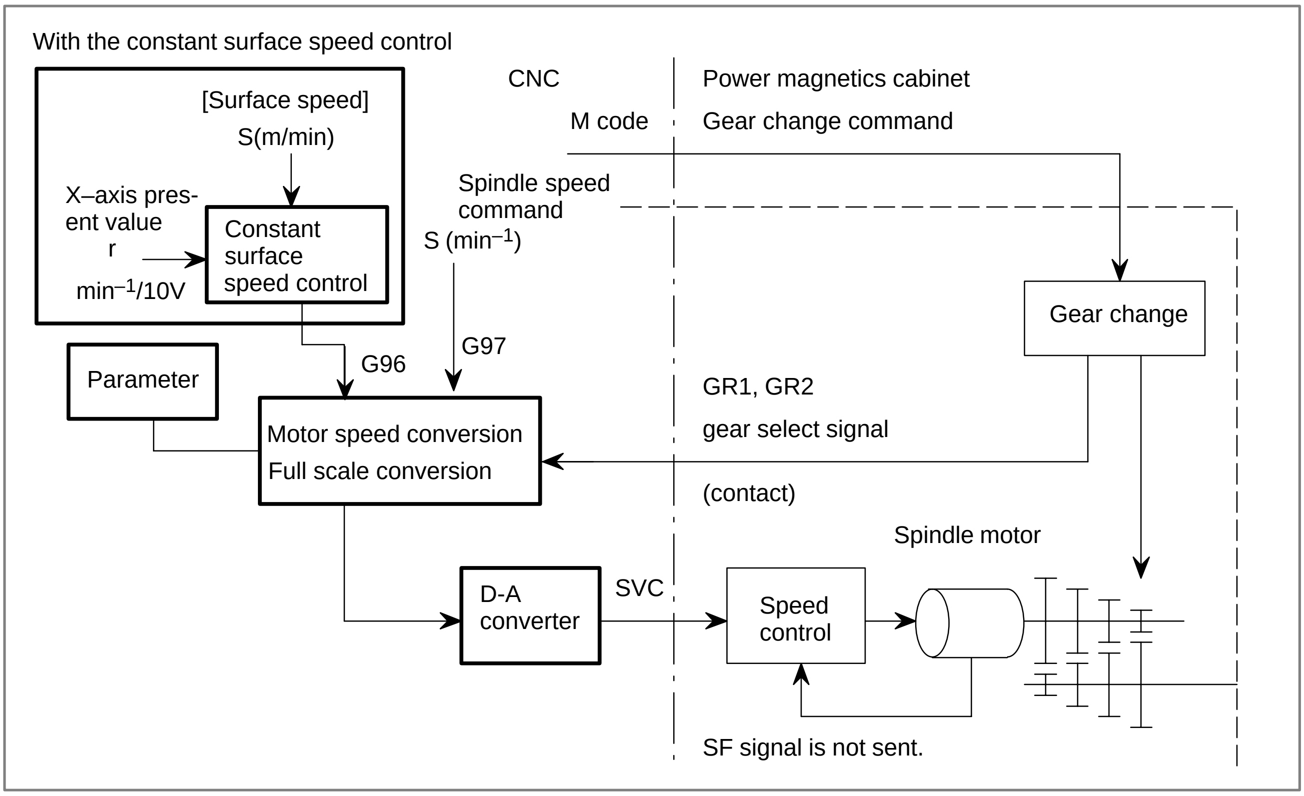
N: Spindle speed given by S command

This is equivalent to the G97 mode for constant surface speed control.

See Section 9.5 for operations during the constant surface speed control mode (G96).

In addition, parameter No. 3772 (upper limit to the spindle speed) can specify speed clamping for all gear positions.

Reference→ Block Diagram for Analog Voltage Output



- **Determination of output R12O–R01O (Output) R12I–R01I (Input) SIND (Input)**

Using the above processing for gear change, the CNC calculates the speed command output to the spindle motor that is necessary to obtain the specified spindle speed with the gear.

For either serial spindle or analog spindle control, the calculation results are output as the S12 bit code signal from 0 to 4095 to the PMC.

(R12O to R01O<F037#3 to F036#0>)

After the calculation results are received, the SIND signal <G033#7> determines which is to be used, the speed command output calculated by the CNC or the data specified in the PMC. Thus the speed command output to the spindle motor is determined. (See also Section 15.4.)

- **Determination of output polarity SSIN/SGN (Input)**

The speed command output to the spindle motor is determined as described above, but the actual output polarity is determined by the CNC as follows:

· If bit 7 (TCW) of parameter No. 3706 = 0
→ Determined according to bit 6 (CWM) of parameter No. 3706

· If bit 7 (TCW) of parameter No. 3706 = 1
→ Determined according to bit 6 (CWM) of parameter No. 3706 and M03/M04 given to the CNC

After that, the SSIN signal <G033#6> determines which is to be used, the output polarity calculated by the CNC or the polarity specified in the PMC. In this way, the polarity of the speed command output to the spindle motor is determined. (See also Section 15.4.)

Keep the following in mind: Even with bit 7 (TCW) of parameter No. 3706 = 1, the CNC cannot determine the output polarity if it has not issued M03/M04, and therefore, actual output does not work even if the speed command has been specified.

- **Command output to spindle**

According to the speed command output and the polarity determined so far, the command is sent to the spindle control unit as follows:

· For serial spindle → Digital data 0 to ± 16383
· For analog spindle → Analog voltage 0 to ± 10 V

- **Requirement of output**

After power is switched on, a nonzero command is output to the spindle only when the following conditions are met: A nonzero spindle speed command is specified, and the output polarity is determined.

With bit 7 (TCW) of parameter No. 3706 = 1, no command output is sent to the spindle, until an M03/M04 is issued because the output polarity is not determined.

- **Requirement to stop output**

The command output to the spindle is reset to 0 when a command to specify so (such as *SSTP = 0 or S0 command) is issued.

M05, emergency stop, or reset does not cause the CNC to reset the command output to the spindle to 0.

- **Spindle enable signal ENB <F001#4>**

Another output related to spindle control is the spindle enable signal ENB.

The ENB signal is logical 1 when a nonzero command output is sent to the spindle. If the command is logical 0, the ENB signal becomes logical 0.

When the analog spindle is being used, an offset voltage in the spindle motor speed amplifier may cause the spindle motor to rotate at low speed even if the command output (in this case, analog voltage) to the spindle is zero. The ENB signal can be used to stop the motor in such a case.

- **Gain and offset**

The analog spindle may require gain and offset voltage adjustment depending on the spindle motor speed amplifier being used.

The following parameters are available for such adjustment.

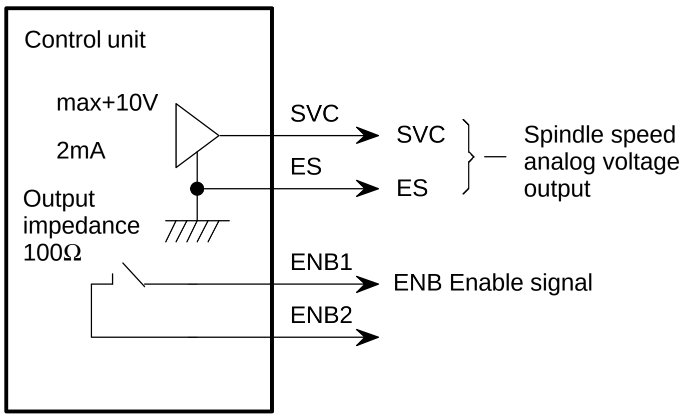
· Analog spindle as the first spindle
→ Gain adjustment data: Parameter No. 3730
Offset voltage compensation: Parameter No. 3731

· Analog spindle as the third spindle
→ Gain adjustment data: Parameter No. 3820
(valid for multispindle control)
Offset voltage compensation: Parameter No. 3821

- **Electrical specification of analog spindle interface**

The signals related to analog spindle interface are described below.

The ENB1 and ENB2 signals are turned on and off under the same condition as for the ENB signal <F001#4>. They can be used also for the serial spindle.



Name	Connector/Pin
SVC	JA8A/7
ES	JA8A/5
ENB1	JA8A/8
ENB2	JA8A/9

> **WARNING**
> Since the output voltage is a weak signal, do not relay it through contacts.

- **Position coder feedback signal**

The position coder is necessary for thread cutting or feed per revolution.

The position coder detects the actual spindle speed and the one–rotation signal (used to detect a fixed point on the spindle for thread cutting).

Ideally, the position coder should be connected directly to the spindle (with a gear ratio of 1:1). If it is necessary to use a gear, select a gear ratio from 1:1, 1:2, 1:4, and 1:8 that reduces the position coder speed.

When using a gear between the spindle and position coder, specify the gear ratio in bits 1 and 0 (PG2, PG1) of parameter No. 3706.

See Section 9.10 for position coder connection for rigid tapping.

- **Speed arrival signal (SAR)**

The spindle speed arrival signal SAR is an input signal used as a condition for the CNC to start cutting feed. This signal is used generally when cutting feed should be started after the spindle reaches the specified speed.

In this case, a sensor is used to check the spindle speed. The detected speed is sent to the CNC via the PMC.

When the above operation is performed continuously using the PC ladder, however, cutting feed may be started based on the SAR signal indicating the previous spindle state (spindle speed before change), if the spindle speed change command and the cutting feed command are issued at the same time.

To avoid the above problem, monitoring the SAR signal can be deferred for a time specified by parameter No. 3740 after the S command or cutting feed command was issued.

When using the SAR signal, set bit 0 (SAR) of parameter No. 3708 to 1.

Item No. 06 (SPINDLE SPEED ARRIVAL CHECK) on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

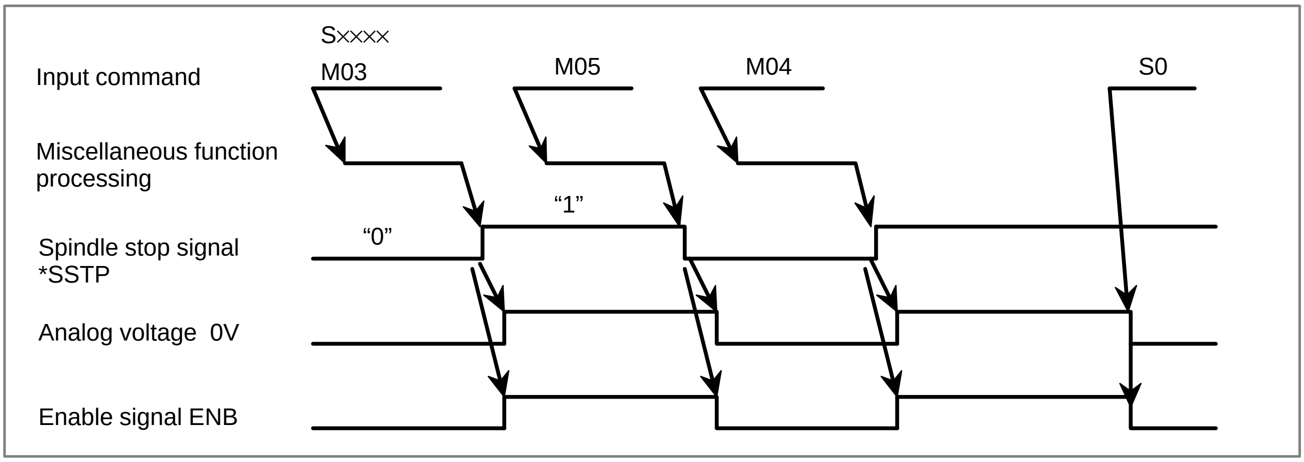
Signal

Spindle stop signal *SSTP<G029#6>

[Classification] Input signal

[Function] The command output to the spindle is disabled.

[Operation] When the spindle stop signal turns to “0” , the output voltage becomes 0V and the enable signal ENB turns to “0” (M05 is not output). When this signal turns to “1”, the analog voltage returns to its original value and the ENB signal turns to “1”.



The above time chart is an example. Actually, the time chart should meet the specification of the spindle control unit.

· When this signal is not used, always set the signal to “1”.

· M03, M04, M05 are not processed inside the CNC.

Spindle orientation signal SOR <G029#5>

[Classification] Input signal

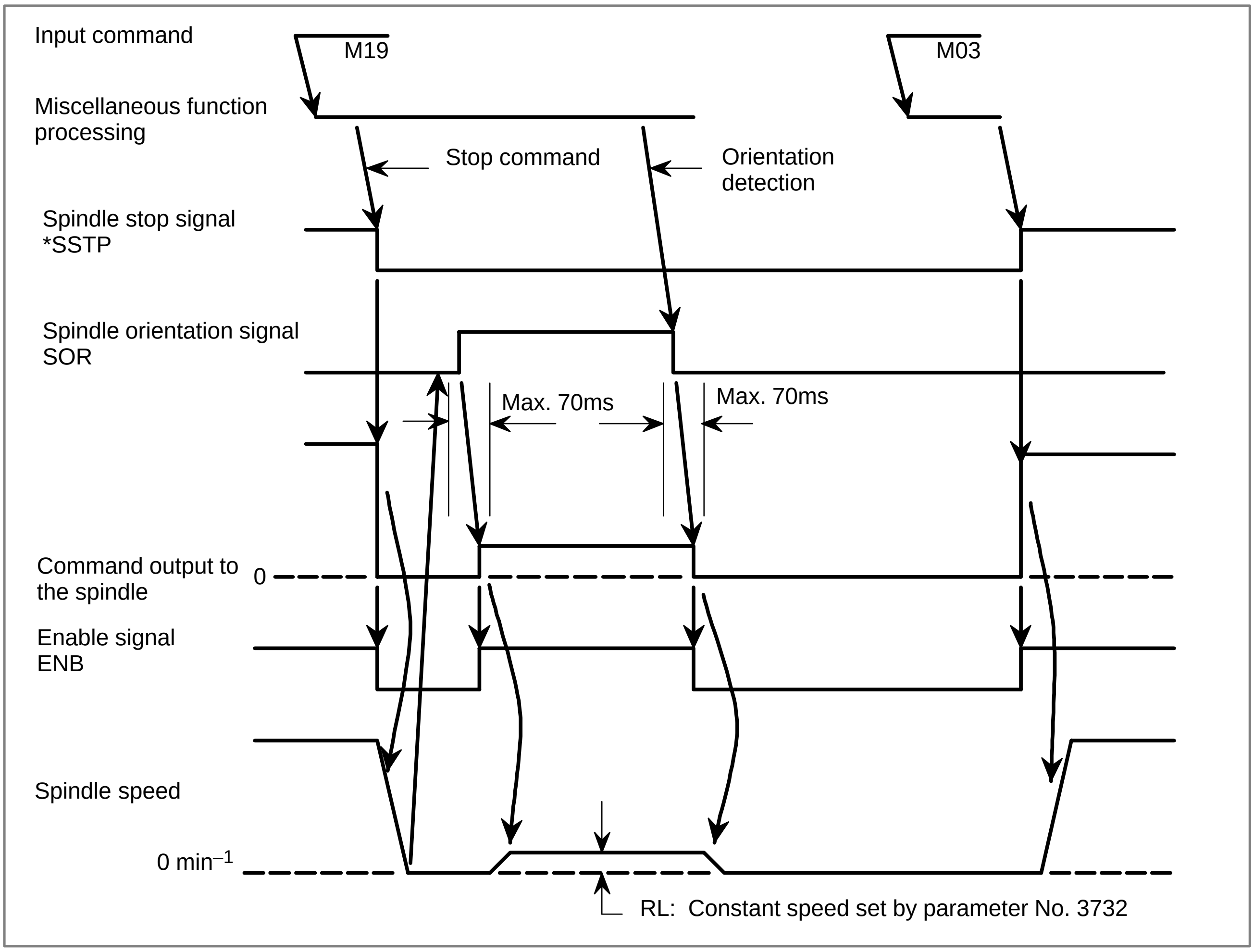
[Function] The spindle or the spindle motor is rotated at a constant speed.

[Operation] When the spindle orientation signal turns to “1” and the spindle stop signal *SSTP turns to “0”, a spindle speed command which lets the spindle rotate at the constant speed set by parameter No. 3732 is output. The enable signal ENB also turns to “1”. This signal is disabled when the spindle stop signal is “1”.

When the spindle speed for orientation is set by parameter GST No. 3705#1 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 3732 with an output polarity set by parameter ORM (No. 3706#5), but the gear select signal does not change. For example, if the SOR signal is turned to “1” with high gear selected, and the speed set to parameter No. 3732 is in the low gear range, the gear select signal does not change and the command output is calculated and output to obtain the set speed at high gear.

When the spindle motor speed is set by parameter GST (No. 3705#1)=1, the command output is output regardless of gear select signal. When the spindle motor speed is set, it is used for gear shift.

Example of usage is shown below:



Spindle speed override signal SOV0 to SOV7 <G030>

[Classification] Input signal

[Function] The spindle speed override signal specifies an override from 0% to 254% in 1% units for the S command sent to the CNC.

[Operation] An override value in binary must be set in 8 bits from SOV7 to SOV0.

The spindle speed override function is disabled (an override of 100% is applied) under the following conditions:

· Tapping cycle (M series : G84, G74 T series : G84, G88)
· Thread cutting (M series : G33 T series : G32, G92, G76)

The spindle override can be enabled in a tapping cycle or in the thread cutting mode if TSO (bit 6 of parameter No.3798) is set accordingly. Whether the function is enabled during rigid tapping depends on the setting of the rigid tapping.

→ When this function is not in use, specify an override of 100%; otherwise, an override of 0% becomes effective, thus disabling the spindle from rotating.

Spindle speed arrival signal SAR <G029#4>

[Classification] Input signal

[Function] The SAR signal initiates cutting feed. In other words, if the signal is logical 0, cutting feed will not start.

[Operation] Generally, this signal is used to inform the CNC that the spindle has reached the specified speed.

For this purpose, the signal must be set to 1 only after the actual speed of the spindle has reached the specified speed.

Setting parameter No. 3740 with a wait time before the start of checking the SAR signal inhibits cutting feed from starting under a condition of SAR = 1 specified before the change of the spindle command.

To use the SAR signal, it is necessary to set bit 0 (SAR) of parameter No. 3708 to 1.

The CNC checks the SAR signal under the following conditions:

a. Bit 0 (SAR) of parameter No. 3708 is set to 1.
b. Before starting distribution of the first feed (move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 3740 has elapsed after the feed block is read.
c. Before starting distribution of the first feed command block after an S code is commanded. The wait time for checking is the same as in item b.
d. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to “1”, feed begins.

CAUTION
According to the conditions of item d above, note that if the circuit is so designed that SAR is turned to “0” simultaneously with the output of an S code and the change of spindle speed is initiated by the DEN signal, the operation will stop. That is, the spindle speed does not reach the commanded speed because the CNC is waiting for the DEN signal and distribution is not started because the CNC is waiting for the SAR signal.

Spindle enable signal ENB <F001#4>

[Classification] Output signal

[Function] Informs absence or presence of spindle output command.

[Output condition] The ENB signal becomes logical 0 when the command output to the spindle becomes logical 0. Otherwise, the signal is logical 1.

During analog spindle control, S0 may not be able to stop the spindle from rotating at low speed because of an offset voltage in the spindle motor speed control amplifier. In such a case, the ENB signal can be used to provide a condition to determine whether to stop the motor.

The analog spindle interface (JA40) has electric signals (ENB1 and ENB2) similar to the ENB. These signals work under the same conditions as with the ENB signal.

The ENB signal can be used also for serial spindle control.

Gear selection signal GR1O, GR2O, GR3O <F034#0 to #2>

[Classification] Output signal

[Function] The gear select signal specifies a gear stage to the PMC.

[Output condition] For details of this signal, see descriptions on the M type gear selection method in General.

Gear selection signal GR1, GR2 <G028#1, #2>

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

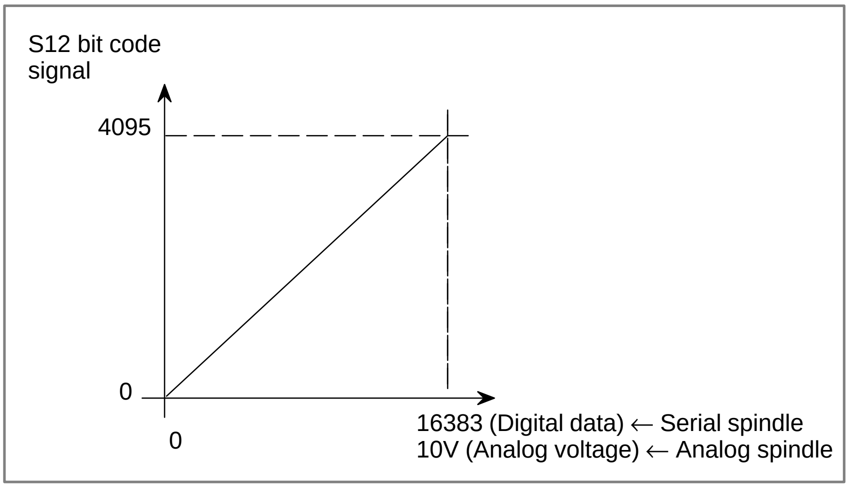
[Output condition] For details of this signal, see descriptions on the T type gear selection method in General.

S12–bit code signal R01O to R12O <F036#0 to F037#3>

[Classification] Output signal

[Function] This signal converts the spindle speed command value calculated by the CNC to code signals 0 to 4095.

[Output condition] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



This signal converts the spindle speed command value calculated by the spindle control function of the CNC to data from 0 to 4095 (for both serial and analog spindle control) and outputs the result. Note that the conversion result is not the actual output value. (See Section 15.4.)

Other signals

Spindle speed function code signal S00 to S31 <F025 to F022> (Output) Spindle speed function strobe signal SF<F007#2> (Output)

See Sections 9.1 and 15.4 for these signals.

Spindle speed output control signal by PMC SIND<G033#7> (Input) R01I to R12I <G032#0 to G033#3> (Input) SSIN <G033#6> (Input) SGN <G033#5> (Input)

See Section 15.4 for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028						GR2	GR1	
G029		*SSTP	SOR	SAR				
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I

	#7	#6	#5	#4	#3	#2	#1	#0
F001				ENB				
F007						SF		
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F034						GR3O	GR2O	GR1O
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT	SGB	GST	ESF

[Data type] Bit

ESF When the spindle control function (S analog outpu or S serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:

0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

> **NOTE**
> For the T series, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.
> For the M series, SF is not output:
> (1) For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode
> (2) When bit 5 (NSF) of parameter No. 3705 is set to 1

GST: The SOR signal is used for:

0 : Spindle orientation
1 : Gear shift

SGB: Gear switching method

0 : Method A (Parameters No. 3741 to 3743 for the maximum spindle speed at each gear are used for gear selection.)
1 : Method B (Parameters No. 3751 and 3752 for the spindle speed at the gear switching point are used for gear selection.)

SGT: Gear switching method during tapping cycle (G84 and G74)

0 : Method A (Same as the normal gear switching method)
1 : Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters No. 3761 and 3762).

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.
1 : Output for an S command.

> **NOTE**
> The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S――;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface speed control,

0 : SF is output.
1 : SF is not output.

SFA: The SF signal is output:

0 : When gears are switched
1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM	ORM				PG2	PG1
	TCW	CWM	ORM	GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method

0 : Type M
1 : Type T

NOTE
1 Type M:
The gear selection signal is not entered externally. In response to an S command, the CNC selects a gear according to the speed range for each gear specified in parameters. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered from the PMC. The spindle speed corresponding to the gear selected by this signal is output.
2 When the constant surface speed control is selected, type T is selected, regardless of whether this parameter is specified.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No. 3705#2 SGB, No. 3751, No. 3752, No. 3705#3 SGT, No. 3761, No. 3762, No. 3705#6 SFA, No. 3735, No. 3736
However, parameter No. 3744 is valid.

ORM Voltage polarity during spindle orientation
0 : Positive
1 : Negative

TCW, CWM Voltage polarity when the spindle speed voltage is output

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO						

[Data type] Bit

TSO During a threading or tapping cycle, the spindle override is:
0 : Disabled (tied to 100%).
1 : Enabled.

NOTE
The operation during rigid tapping depends not only on this parameter but on the setting of rigid tapping.

	#7	#6	#5	#4	#3	#2	#1	#0
3709								SAM

[Data type] Bit

SAM The sampling frequency to obtain the average spindle speed
0 : 4 (Normally, set to 0.)
1 : 1

3730	Data used for adjusting the gain of the analog output of spindle speed

[Data type] Word

[Unit of data] 0.1 %

[Valid data range] 700 to 1250

Set data used for adjusting the gain of the analog output of spindle speed.

[Adjustment method]
(1) Assign standard value 1000 to the parameter.
(2) Command the maximum spindle speed.
(3) Measure the output voltage.
(4) Assign the value obtained by the following equation to parameter No. 3730.

$$\text{Set value} = \frac{10\ (V)}{\text{Measured data (V)}} \times 1000$$

(5) After setting the parameters, command the maximum spindle speed, confirm that the output is 10V.

> **NOTE**
> This parameter needs not to be set for serial spindles.

3731	Compensation value for the offset voltage of the analog output of the spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to +1024

Set compesation value for the offset voltage of the analog output of the spindle speed.
Set value =–8191 × Offset voltage (V)/12.5

[Adjustment method]
(1) Assign standard value 0 to the parameter.
(2) Command “0”.
(3) Measure the output voltage.
(4) Assign the value obtained by the following equation to parameter No. 3731.

$$\text{Set value} = \frac{-8191 \times \text{Offset voltage (V)}}{12.5}$$

(5) After setting the parameters, command “0”, confirm that the output is 0V.

> **NOTE**
> This parameter need not to be set for serial spindles.

3732	The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift

[Data type] Two–word

[Valid data range] 0 to 20000

Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.

When GST, #1 of parameter 3705, is set to 0, this is the spindle speed during spindle orientation in min^{-1}.

When GST, #1 of parameter 3705, is set to 1, this is the spindle motor speed during spindle gear shift calculated from the following formula.

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 16383 \text{ (For a serial spindle)}$$

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 4095 \text{ (For an analog spindle)}$$

3735	
	Minimum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the minimum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Minimum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$

> **NOTE**
> If the function of constant surface speed control or bit 4 (GTT) of parameter No. 3706 is specified, this parameter is invalid.

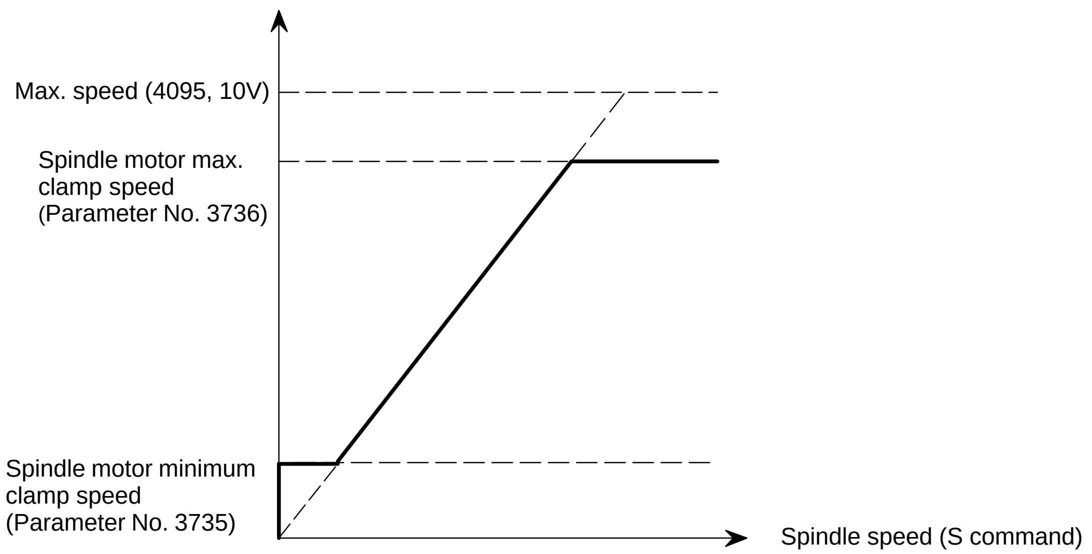
3736	
	Maximum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the maximum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Maximum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$



NOTE

If the function of constant surface speed control or bit 4 (GTT) of parameter No. 3706 is specified, this parameter is invalid.
In this case, the maximum clamp speed of spindle motor cannot be specified. However, the maximum spindle speed can be specified by the following parameters.
Parameter No.3772 (for the first axis)
Parameter No.3802 (for the second axis)
Parameter No.3822 (for the third axis)

3740	Time elapsed prior to checking the spindle speed arrival signal

[Data type] Byte

[Unit of data] msec

[Valid data range] 0 to 225

Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

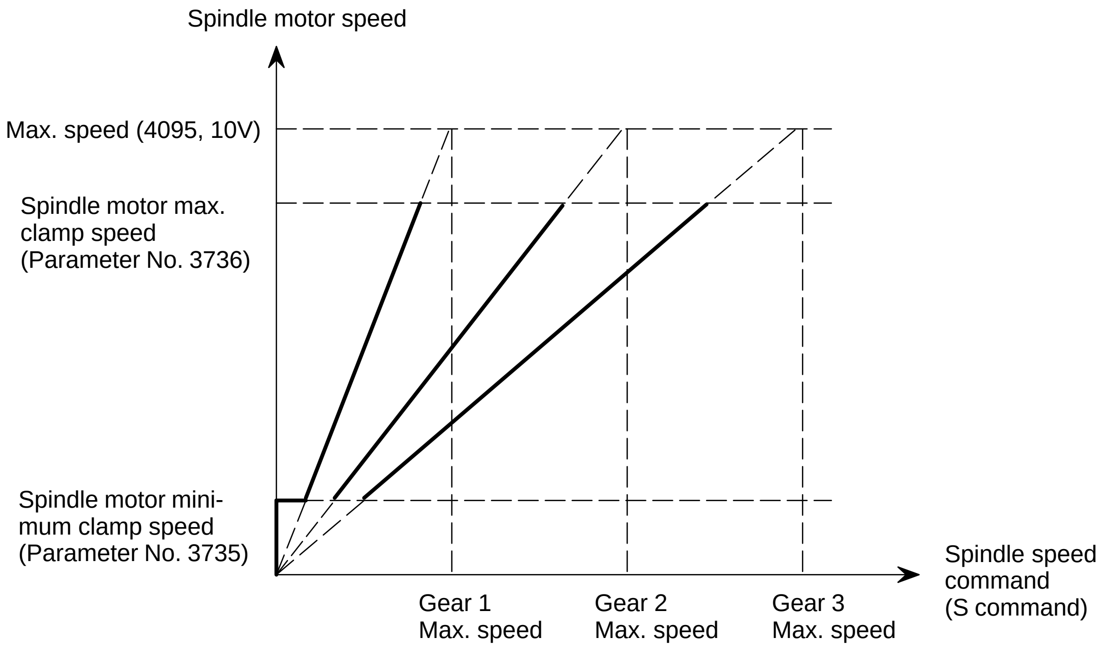
3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Set the maximum spindle speed corresponding to each gear.



3751	
	Spindle motor speed when switching from gear 1 to gear 2

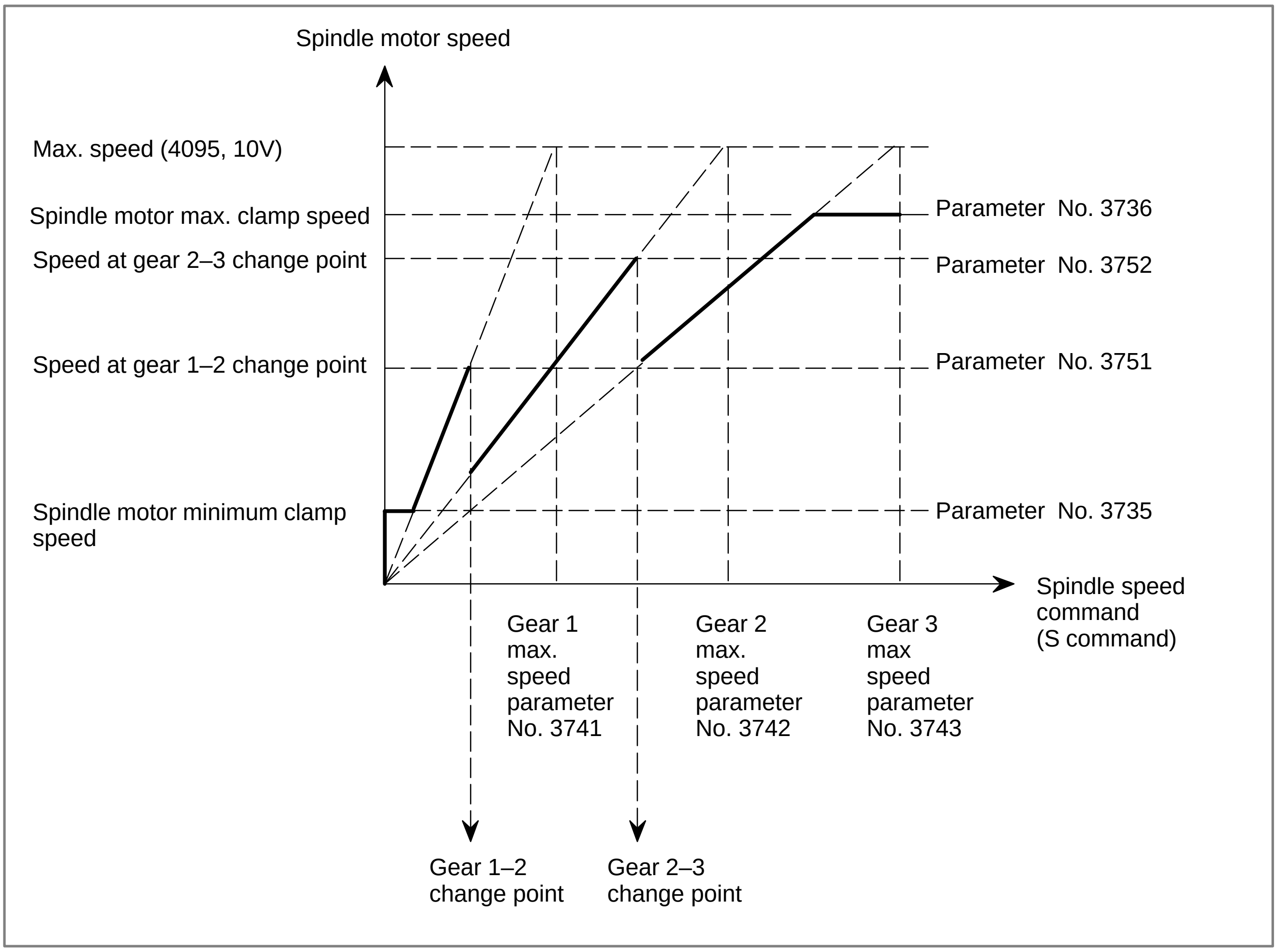
3752	
	Spindle motor speed when switching from gear 2 to gear 3

[Data type] Word

[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.

$$\text{Set value} = \frac{\text{Spindle motor speed when the gears are switched}}{\text{Maximum spindle motor speed}} \times 4095$$



3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping

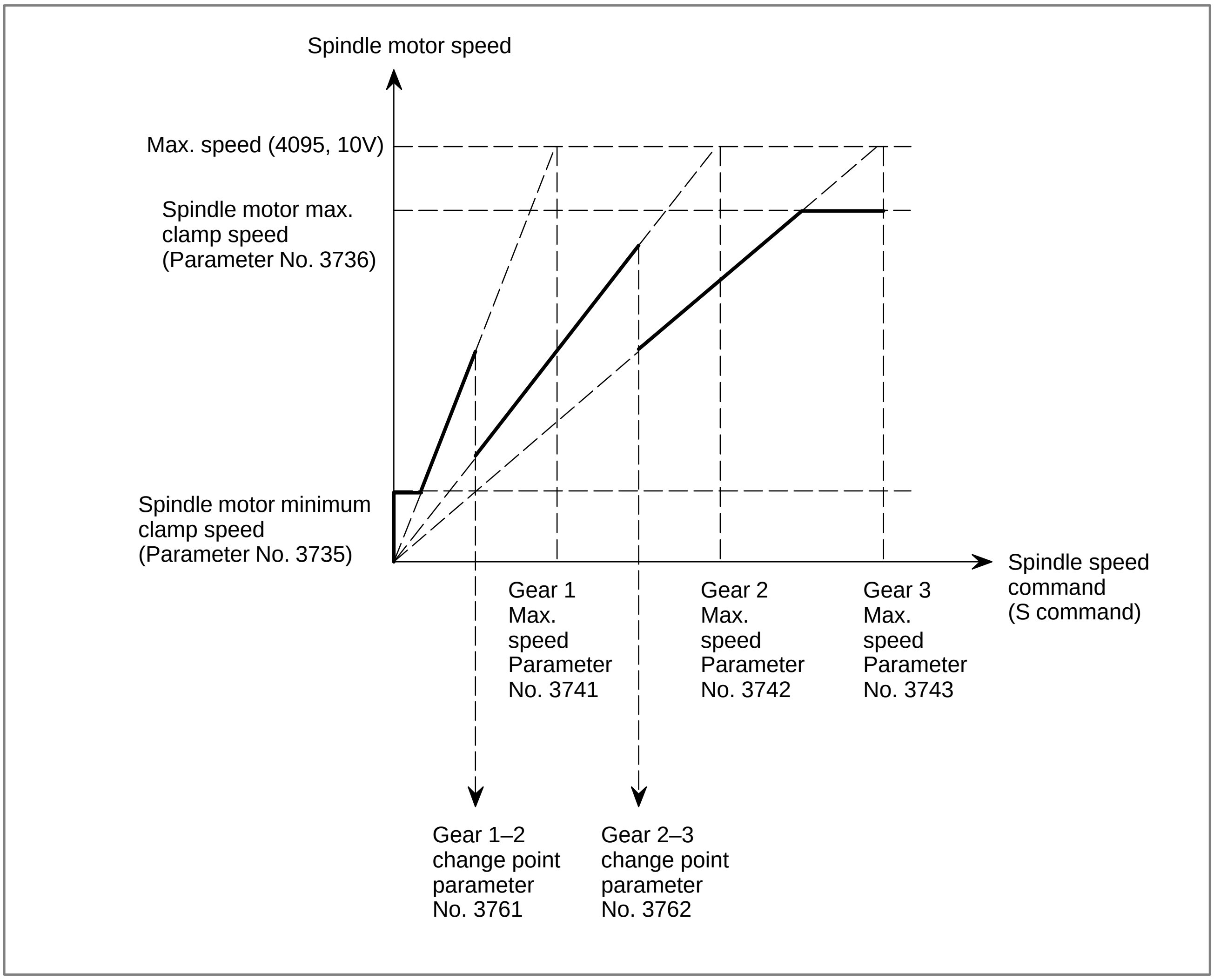
3762	
	Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



3772	Maximum spindle speed

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

WARNING
1 When 0 is set in this parameter, the speed of the spindle is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE
1 In the M series, this parameter is valid only when the constant surface speed control is present.
2 When the constant surface speed control is present, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.
3 When the multi–spindle control option is present, set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

Caution

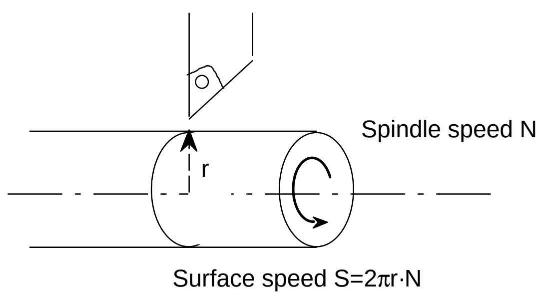
CAUTION

This section mentioned a spindle speed control that should be prepared on the CNC side. But it is also necessary to design the signals to the spindle control unit.
Consult the manual of the spindle control unit used and take necessary actions on the spindle control unit.

9.4 CONSTANT SURFACE SPEED CONTROL

General

With the spindle serial output or analog output function, specifying the surface speed (m/min or feet/min) directly in an S command makes it possible to change the spindle output continuously so as to maintain a constant surface speed at a programmed point. (For the rapid traverse command, however, the surface speed for the end point is output at the beginning of rapid traverse.)



Whether or not constant surface speed control is performed is selected by G code.

G96: Constant surface speed control performed. S in the G96 mode is m/min or feet/min.

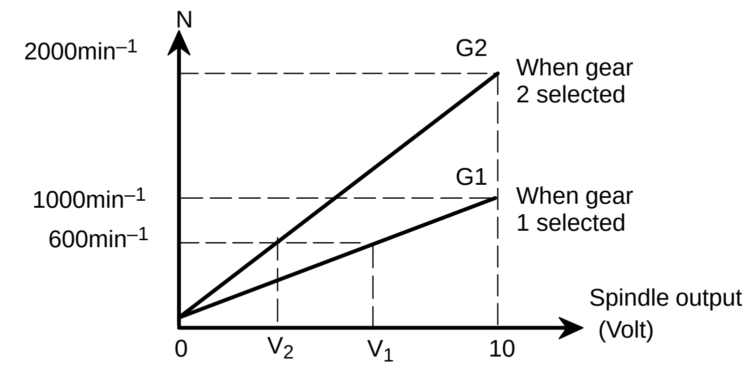
G97: Constant surface speed control not performed. S in the G97 mode is rev/min.

To perform constant surface speed control, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameter No. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear number
0	0	1
1	0	2
0	1	3
1	1	4

Example of Spindle Analog Output

Assume that gear switching is two stage switching. If the spindle speed with the output 10 V is 1000 min^{-1} for the low speed gear (G1) and 2000 min^{-1} for the high speed gear (G2), set these speeds to the parameter No. 3741, 3742, respectively. In this case, the spindle output has the linear relationship shown below:



Here, S = 60 m/min is given as the surface speed; if the position of the present X-axis cutter is 16 mm from the center, the spindle speed N becomes 600 min^{-1} ($S = 2\pi r N$). Therefore, V_1 (for G1) or V_2 (for G2) is calculated inside the CNC and output to the machine side.

V_1: 6(V)

V_2: 3(V)

The value of output voltage V is calculated automatically from the following equations:

(i) G96

$$V = \frac{10S}{2\pi rR}$$

R: Spindle speed (min^{-1}) at 10V output voltage (that is , spindle speed set by parameter No. 3741 to No. 3744)

S: Surface speed (m/min) specified by S command

r: Radius value in the X-axis direction (m)

(ii) G97

$$V = \frac{10N}{R}$$

R: Spindle speed at 10V output voltage (min^{-1})

N: Spindle speed given by S command (min^{-1})

Spindle Serial Output

The output to the spindle in spindle serial output is a digital data.

Therefore assume the following relation for calculation:

Spindle analog output (voltage) 10V = Spindle serial output (digital data) 4095.

The above calculation becomes as follows:

The value of Spindle output D:

(i) G96

$$D= \frac{4095S}{2\pi rR}$$

R: Spindle speed (min^{-1}) at maximum spindle motor speed (that is , spindle speed set by parameter No. 3741 to No. 3744)

S: Surface speed (m/min) specified by S

r: Radius value in the X-axis direction (m)

(ii) G97

$$D= \frac{4095N}{R}$$

R: Spindle speed at maximum spindle motor speed (min^{-1})

N: Spindle speed given by S command (min^{-1})

Signal

Gear selection signal (Input) GR2, GR1 <G028#2, #1>

Refer to section 9.3 “Spindle Speed Control”.

Constant surface speed signal CSS <F002#2>

[Classification] Output signal

[Function] This signal indicates that constant surface speed control is in progress.

[Output condition] “1” indicates that constant surface speed control mode (G96) is in progress, while “0” indicates it is not.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002						CSS		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1405						FPR		

[Data type] Bit

FPR Specifies the feed–per–revolution function with no position coder.

0 : Not used.
1 : Used.

> **NOTE**
> If you set this parameter to 1, reset parameter NPC (bit 0 of parameter No. 1402) to 0.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] min^{-1}

3770	Axis as the calculation reference in constant surface speed contorol

[Data type] Byte

[Valid data range] 0, 1, 2, 3, ..., number of control axes

set the axis as the calculation reference in constant surface speed contorol.

> **NOTE**
> When 0 is set, constant surface speed control is always applied to the X–axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3771	Minimum spindle speed in constant surface speed control mode (G96)

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Set the minimum spindle speed in the constant surface speed control mode (G96).
The spindle speed in constant surface speed control is clamped to the speed given by parameter 3771.

3772	Maximum spindle speed

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum spindle speed is specified, or the spindle speed exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

NOTE
1 In the M series, this parameter is valid when the constant surface speed control is present.
2 When the constant surface speed control is present, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

Alarm and message

Number	Message	Description
190	ILLEGAL AXIS SELECT	The axis specification is wrong in constant surface speed control. (See parameter No. 3770.) The specified axis command (P) contains an illegal value. Correct the program.

Caution

CAUTION
1 If the spindle speed corresponding to the calculated surface speed exceeds the speed specified in the spindle speed clamp command (G50S_ for T series and G92S_ for M series) during the G96 mode, the actual spindle speed is clamped at the value specified in the spindle speed clamp command.
If the specified spindle speed is lower than the value specified in parameter No. 3771, the actual spindle speed is clamped at the speed specified by No. 3771.
2 If the constant surface speed control function is provided for a machining center system, it affects gear change under normal spindle control. (See Section 9.3.)

Note

NOTE
Simultaneous use of multi–spindle control enables constant surface speed control for spindles other than the first spindle. (See Section 9.9.)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)

9.5 SPINDLE SPEED FLUCTUATION DETECTION

General

With this function, an overheat alarm (No. 704) is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

Detection of Spindle Speed Fluctuation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Sd or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

Sd: The allowable constant variation width which is independent of the specified spindle speed (Sd is set with parameter No. 4913.)

Sr: The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r is set with parameter No. 4912.)

Sm: Sd or Sr, whichever is greater

Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met:

· The actual spindle speed falls in a range of (Sc – Sq) to (Sc + Sq)

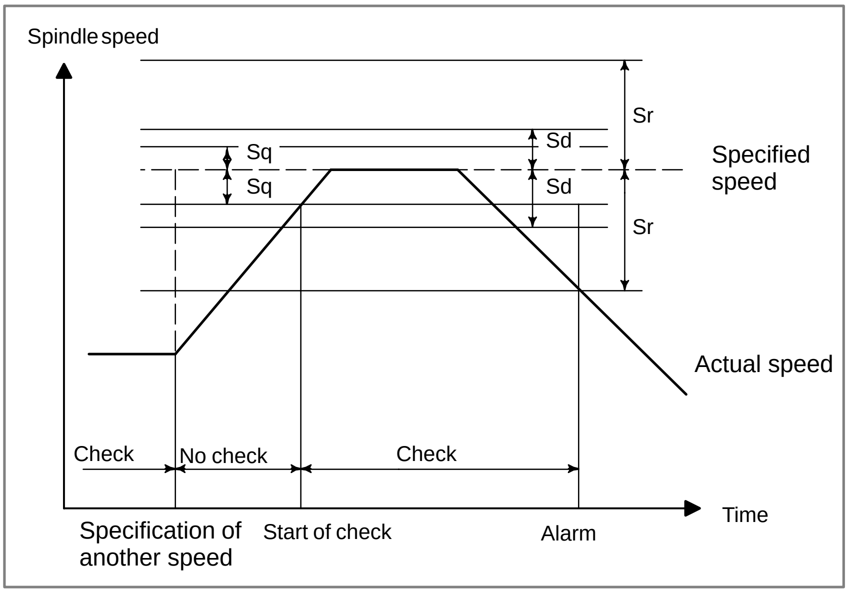
where Sq = Sc $\times$ q/100

q: Percent tolerance of the target spindle speed, specified in parameter No. 4911. If the actual spindle speed is in a range of the specified speed $\pm$q, it is assumed that the actual speed has reached the specified speed.

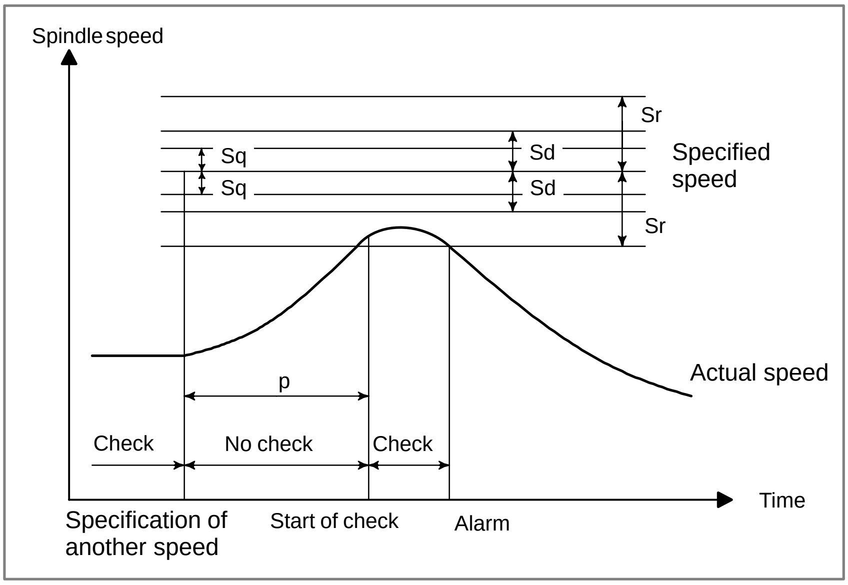
· When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

Parameter No. 4914, 4911, and 4912 can be rewritten also by program (G26 PpQqRr).

1. When an alarm is issued after a specified spindle speed is reached



2. When an alarm is issued before a specified spindle speed is reached



Specified speed :
(Speed specified by address S and five–digit value)×(spindle override)

Actual speed : Speed detected with a position coder

Signal

Spindle fluctuation detection alarm signal SPAL <F035#0>

[Classification] Output signal

[Function] This signal indicates that the actual spindle speed is not within a tolerance to the specified speed.

[Output condition] The signal becomes logical "1" when:

· The actual spindle speed goes out of tolerance to the specified speed.

The signal becomes logical "0" when:

· No alarm condition has been detected for spindle speed fluctuation.

· An alarm condition is cleared by resetting the NC when the signal is logical "1".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F035								SPAL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3708				SVD				

[Data type] Bit

SVD When the SIND signal is on, the detection of spindle speed fluctuation is:

0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
4900								FLR

[Data type] Bit

FLR When the spindle speed fluctuation detection function is used, the rates of allowance (q) and fluctuation (r) those are set in parameter No. 4911 and No. 4912, respectively are set in steps of:

0 : 1%
1 : 0.1%

4911	Percent tolerance (q) of the target spindle speed to begin checking

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

NOTE
Unit of data depends on parameter No. 4900#0 FLR (T series only)

Set the percent tolerance (q) of the target spindle speed to begin checking.

4912	Spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

NOTE
Unit of data depends on parameter No. 4900#0 FLR (T series only).

Set the spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function.

4913	Spindle speed fluctuation value (d) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Set the allowable fluctuation speed (d) for which no alarm is activated in the spindle speed fluctuation detection function.

4914	Time (p) elapsed from when the commanded spindle speed is changed to the start of spindle speed fluctuation detection

[Data type] Two–word

[Unit of data] msec

[Valid data range] 0 to 999999

Set the time (p) elapsed from when the specified spindle speed is changed to the start of spindle speed fluctuation detection. The fluctuation in the spindle speed is not detected until the specified time elapses from when the specified spindle speed is changed.

Alarm and message

Number	Message	Description
704	OVER HEAT : SPINDLE	Spindle overheat in the spindle fluctuation detection (1) If the cutting load is heavy, relieve the cutting condition. (2) Check whether the cutting tool is share. (3) Another possible cause is a faulty spindle amp.

Note

NOTE
1 When an alarm is issued in automatic operation, a single block stop occurs.
2 No check is made during spindle stop state (*SSTP = 0).
3 An alarm is issued one second later if the actual spindle speed is found to be 0 min^{-1}.
4 Issuing the alarm does not cause the spindle to stop automatically.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	II.9.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)

9.6 ACTUAL SPINDLE SPEED OUTPUT (T SERIES)

General

The PMC can read actual spindle speed.

Signal

Actual spindle speed signal AR0 to AR15 <F040, F041>

[Classification] Output signal

[Function] These 16-bit binary code signals output from the CNC to the PMC the actual spindle speed obtained by feedback pulses from the position coder mounted on the spindle.

[Operation] Spindle speed $=\sum_{i=0}^{15}\{2^i \times V_i\}\text{min}^{-1}$

where Vi = 0 when ARi is "0" and Vi = 1 when ARi is "1"

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08

Note

NOTE
1 The AR0 - AR15 signals are always output. Their values change every 64 msec.
2 An absolute error of about 0.5 min^{-1} exists as a measuring error.

9.7 SPINDLE POSITIONING (T SERIES)

General

This function positions the spindle using the spindle motor and position coder.

The function has a coarser least command increment compared with the Cs contour control function and has no interpolation capability with other axes. However, it can be installed more easily because the position detector is a position coder.

Generally, the spindle positioning axes are clamped mechanically except when positioning is under way.

In the turning process, the workpiece is rotated by the spindle to which it is attached (spindle rotation mode), at the speed specified for the spindle motor. The value for the spindle speed is input from the spindle controller to the spindle amplifier.

When the optional spindle positioning function is activated, the spindle is moved to a defined angle, and the workpiece is repositioned at that angle. The specified move distance is input to the error counter, and the velocity command is issued for the spindle motor through the spindle amplifier. The position of the spindle is detected by the installed position coder (Spindle positioning mode).

The spindle positioning function can perform the following operations:

- Release the spindle rotation mode and enter the spindle positioning mode

  Specifying a particular M code sets a reference position in the spindle positioning mode. (This is called spindle orientation.)
- Position the spindle in the spindle positioning mode

  Position an optional angle using address C (H), and position a semi-fixed angle using a specific M code parameter.
- Release the spindle positioning mode and enter the spindle rotation mode

  Specifying a particular M code parameter changes the spindle to the spindle rotation mode.

Also, relationship between M codes and these operations are set by parameters (refer to No. 4950#2 (ISZ), #7 (IMB)).

· Least command increment

$$\frac{360}{4096} \fallingdotseq 0.088 \text{ deg}$$

· Minimum input increment

0.001 deg

· Maximum command value

±9999.999 deg

Selecting a spindle positioning axis

Any axis in the control axis group can be used as the C axis (parameter no. 1020). Specify -1 as its servo axis number (parameter no. 1023).

The spindle subjected to spindle positioning is the first spindle.

Switching to spindle positioning mode (Spindle orientation)

Orientation is required in advance if spindle positioning is first performed after the spindle motor is used as a normal spindle, or when spindle positioning is interrupted. The orientation stops the spindle in a constant position. The orientation position can be sifted in the range of ± 180deg for analog spindle and in the range from 0 to 360° for serial spindle.

To specify orientation, use the M code specified in parameter No. 4960.

The orientation direction is specified by using parameters ZMIx No. 1006 #5 for analog or RETURN No. 4000#5 for serial spindles.

- **Orientation speed**

The spindle moves at rapid traverse (set by parameter No. 1420) until it reaches the orientation enable speed (shown below). After the spindle crosses that speed point, it performs orientation at the speed set by parameter No. 1425. When a serial spindle is used, orientation speed depends on the spindle.

· Orientation enable speed
RPD>9 × (loop gain) KPPM
Loop gain: Parameter No. 4970 (unit: 1/sec)
Set rapid traverse speed at above value.

(Example)

When the loop gain parameter No. 4970 is set to 20 [1/sec], the orientation speed is:

RPD>9 × 20 × 1000 × (360/4096)=15820 [deg /min]

The serial spindle stops at the orientation position as soon as the command is issued. The lower limit to the rapid traverse speed value does not need to be specified for the serial spindle to reach the orientation enable speed.

The analog spindle stops after the spindle speed is changed from rapid traverse to the FL speed. The rapid traverse speed lower limit must be specified for the analog spindle, or obtaining the orientation enable speed need not be specified for the serial spindle. However, it must be specified for the analog spindle.

- **Program origin**

The orientation position is regarded as a program origin. It is possible to modify the program origin using the coordinates system.

Command system

The command system comes in two types: The first positions a semi-fixed angle; the second positions an optional angle.

• Semi-fixed angle positioning by M code

A 2-digit numerical value following the M address is used for the command. There are six positioning angle values (Mα to M(α + 5)), where α is set by parameter No. 4962. Indexing angle β also requires prior parameter setting data No. 4963. Rotation can be done in any direction, using parameter IDM data No. 4950#1.
Also, when extended specification is used (parameter No. 4950#6 ESI=1), max. 256 kinds of values (Mα to M(α+255)) can be specified.

M–code	Indexing angle	eg) when β=30°
Mα	β	30°
M (α+1)	2β	60°
M (α+2)	3β	90°
M (α+3)	4β	120°
M (α+4)	5β	150°
M (α+5)	6β	180°

• Optional angle positioning by C or H address

Numerical value following either the C or H address is used to command the position. C and H addresses are commanded in G00 mode.

(Example) C-1000

H4500

(i) Minimum setting unit :

0.001deg

(ii) Maximum command value:

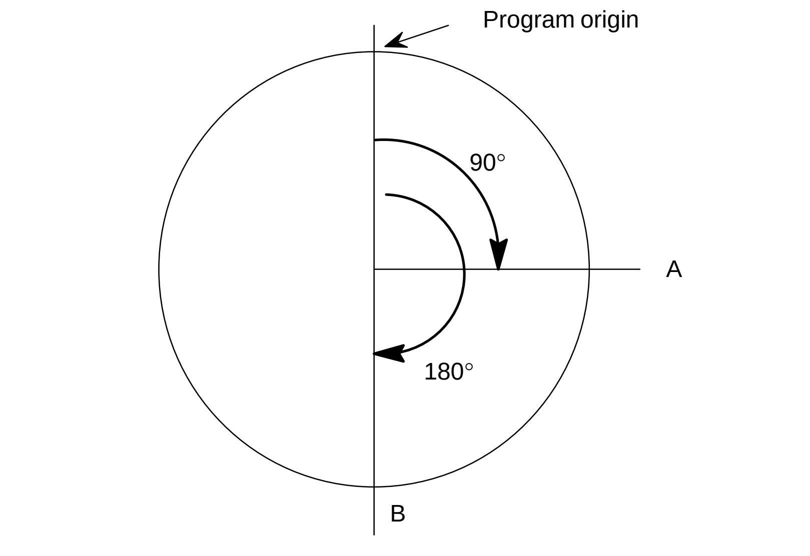
±9999.999 deg

(iii) Decimal point input: A numerical value with decimal point can be entered. The decimal point location is in “degrees”, for instance:

C35.0=C35 degrees

- **Absolute and incremental commands**

Positioning by specifying a semi–fixed angle (by M code) is always incremental. To perform positioning by specifying an optional angle, specify the distance between the program origin and the end point (absolute) with address C and the distance between the start point and the end point (incremental) with address H.



Command method		G code system A		G code system B, C	
		Address used	Command of A→B on the above Fig.	Address used and G–code	Command of A→B on the above Fig.
Absolute command	Direct the end point position by the distance from the program origin.	C	C180.0 ;	G90,C	G90C180.0 ;
Incremental command	Command by the distance between the start and end points.	H	H90.0 ;	G91,C	G91C90.0 ;

- **Spindle positioning feedrate**

Spindle positioning is done at the rapid traverse rate specified by parameter No. 1420, to which linear acceleration deceleration are applied. Overrides of 100%, 50%, 25% and F0 (parameter No. 1421) are also applied.

- **Spindle positioning reset**

A specific M code (parameter no. 4961) must be set when the mode is changed from spindle positioning to normal spindle rotation.

Signal

Spindle stop complete signal SPSTP <G028#6>

[Classification] Input signal

[Function] When this signal is 1, the CNC orients and positions the spindle.

Spindle unclamp signal SUCLP <F038#1>

[Classification] Output signal

[Function] This signal specifies that spindle mechanical clamping be released in a spindle positioning sequence.

When this signal is output, unclamp the spindle on the machine (release the brakes or extract the pin).

[Output condition] Refer to the sequence (time chart) in this section.

Spindle unclamp completion signal *SUCPF <G028#4>

[Classification] Input signal

[Function] This signal indicates that unclamping the spindle is complete in response to the spindle unclamp signal SUCLP.

Spindle clamp signal SCLP <F038#0>

[Classification] Output signal

[Function] This signal specifies that the spindle be clamped mechanically in a spindle positioning sequence.

When this signal turns to 1, clamp the spindle on the machine (apply the brakes or insert the pin).

[Output condition] Refer to the sequence (time chart) in this section.

Spindle clamp completion signal *SCPF <G028#5>

[Classification] Input signal

[Function] This signal indicates that clamping the spindle is complete in response to the spindle clamp signal SCLP.

Other signals

Gear selection signal GR1, GR2, <G028#1, #2>

Refer to 9.3 “Spindle Speed Control.”

Gear selection signal CTH1A, CTH2A <G070#3, #2>

Refer to the manual for serial spindle.

The spindle loop gain multiplier corresponding to the gear currently selected by this signal is used. When the serial spindle is used, input gear selection signals CTH1A and CTH2A, as well.

Relationship between the selected gear and spindle gear selection signal

Analog spindle		
GR2	GR1	Selected gear
0	0	1st gear
0	1	2nd gear
1	0	3rd gear
1	1	4th gear

Serial spindle		
CTH1A	CTH2A	Selected gear
0	0	HIGH
0	1	MEDIUM HIGH
1	0	MEDIUM LOW
1	1	LOW

Spindle orientation completion signal ZPx<F094>

[Classification] Output signal

[Function] This signal indicates that the spindle orientation for the spindle positioning has been completed.

[Output condition] When spindle orientation is complete, this signal turns to 1. When spindle positioning is performed or cleared, it turns to 0.

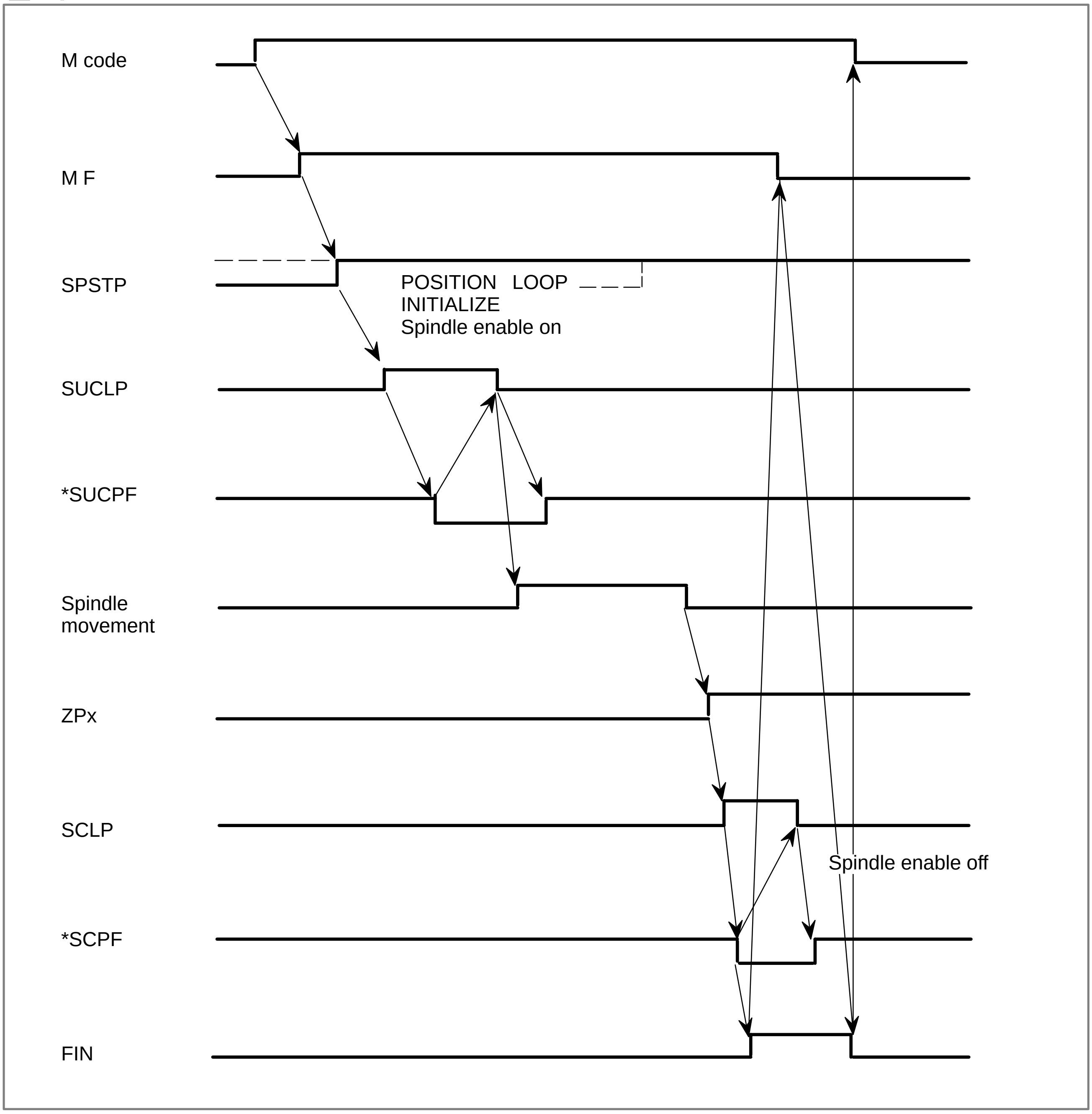
Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G028		SPSTP	*SCPF	*SUCPF		GR2	GR1	
G070					CTH1A	CTH2A		

	#7	#6	#5	#4	#3	#2	#1	#0
F038							SUCLP	SCLP
F094					ZP4	ZP3	ZP2	ZP1

Sequence (Time chart)

❑ Spindle Orientation



⇒ POSITION LOOP INITIALIZE is performed within the CNC.

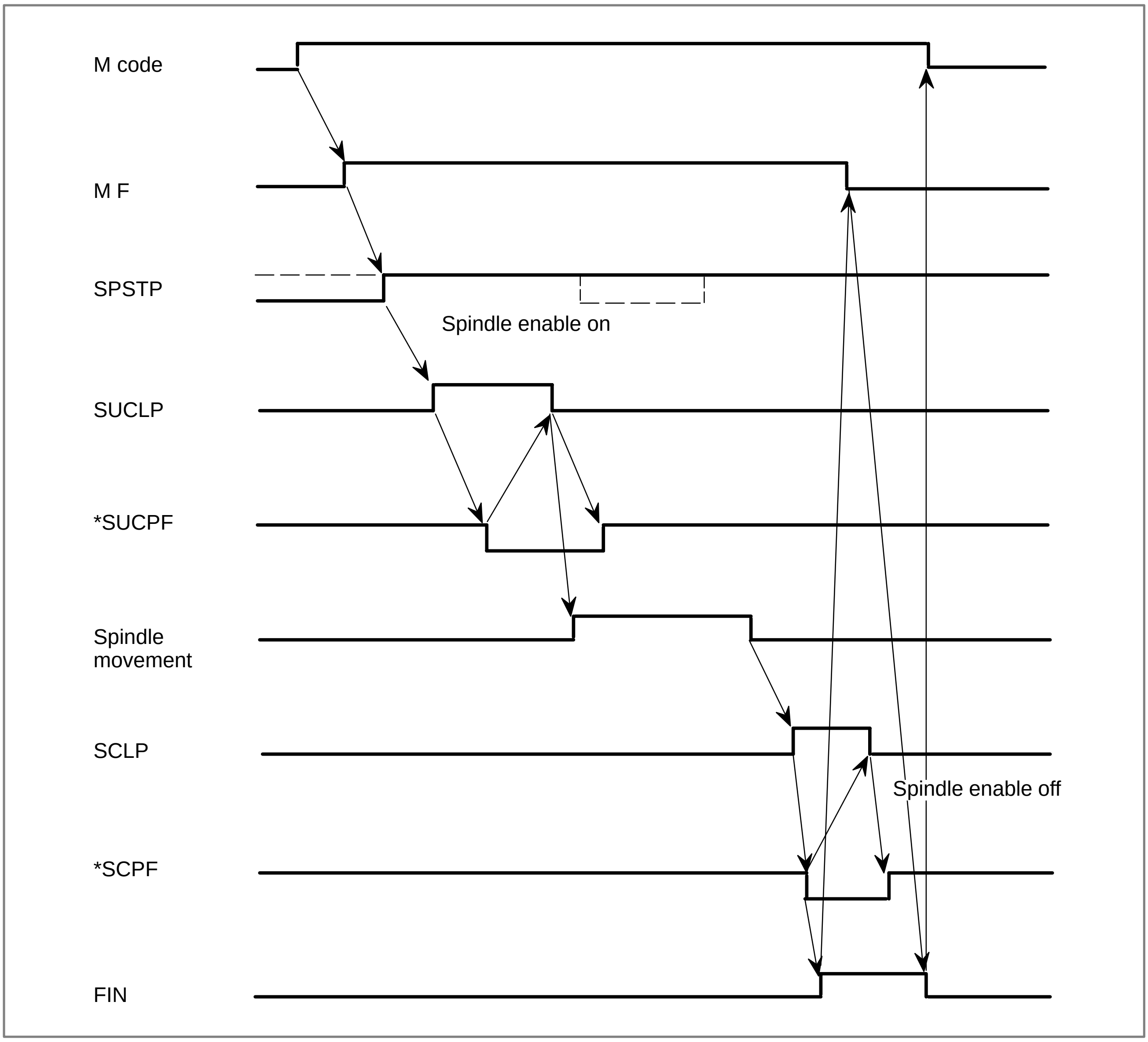
⇒ Spindle ENABLE ON/OFF specifies that the PMC ladder direct the spindle control unit to run or stop the spindle motor.

(Example) For serial spindles, the ladder should contain the following command or something like that:

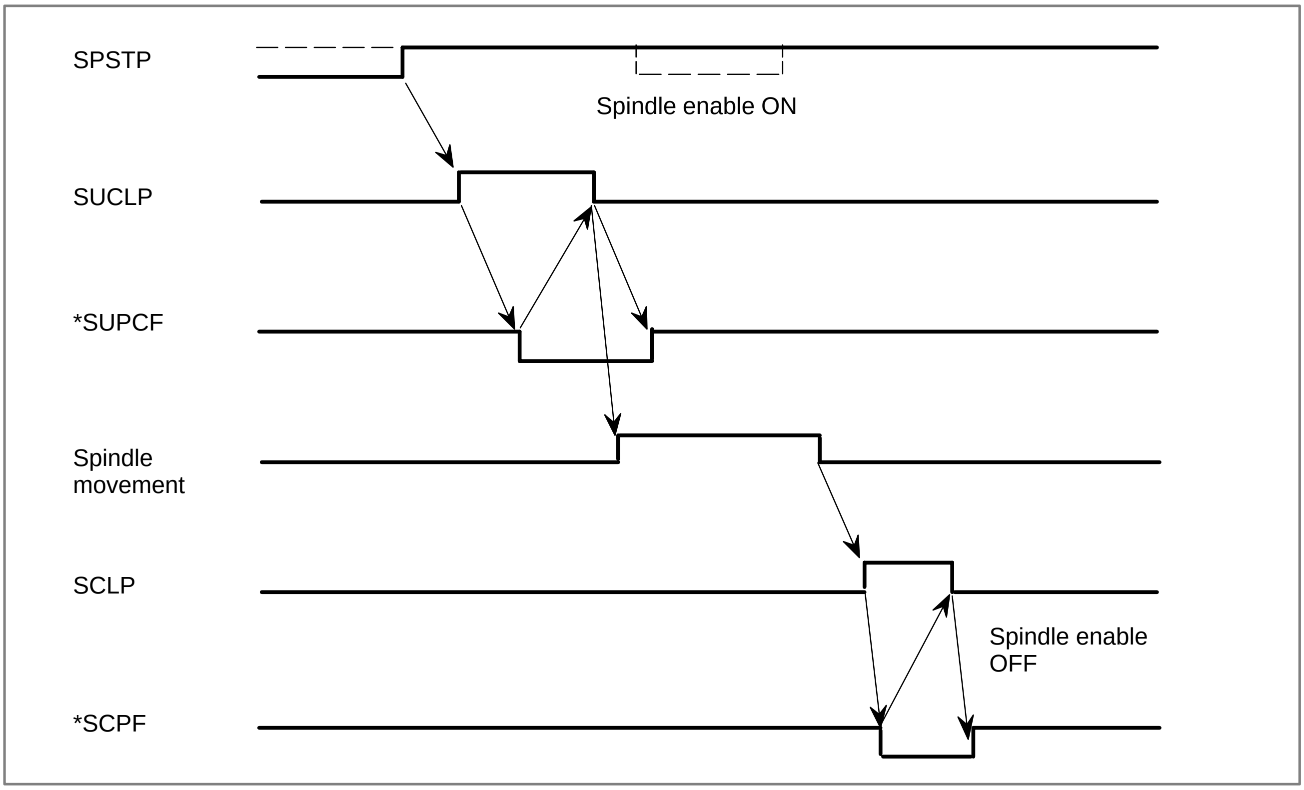
ENABLE ON, and SFRA<G070#5> ⇒ 1
ENABLE OFF, and SFRA<G070#5> ⇒ 0

For details, refer to the manual for the spindle control unit you actually use.

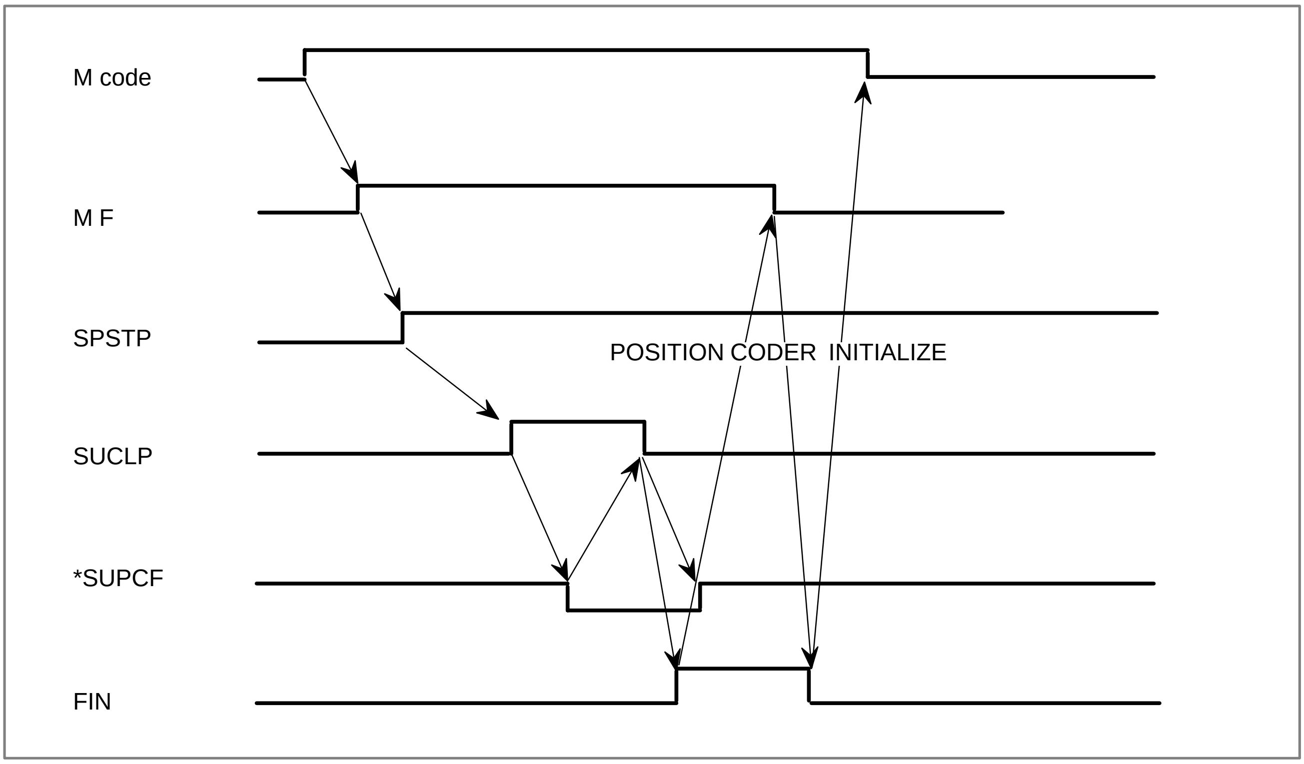
❑ Spindle Positioning by M code



❑ Spindle Positioning by Address C,H



❑ Spindle Positioning Reset



⇒ POSITION CODER INITIALIZE is performed only in the CNC.

Parameter

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction
1 : Negative direction

> **NOTE**
> When the serial spindle is being used, this parameter is invalid for the spindle positioning axis.

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

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

> **NOTE**
> 1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
> 2 The same axis name cannot be assigned to more than one axis.

<u>The axis name of spindle positioning is C axis.</u>

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

<u>Set –1 to the C axis when spindle positioning function is used.</u>

1420	Rapid traverse rate for each axis

[Data type] Word axis

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

For spindle positioning.

[Unit of data] 10 deg/min

[Valid data range] 30 to 12000

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

Set the F0 rate of the rapid traverse override for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

1425	FL rate of the reference position return for each axis

[Data type] Word axis

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

NOTE

When serial spindle is used, this parameter becomes invalid.

1620	Time constant of rapid traverse linear acceleration/deceleration for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set time constant of rapid traverse linear acceleration/deceleration for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				

[Data type] Bit axis

DM1x to DM3x Setting of detection multiply

Set this parameter to “111” (=4) for spindle positioning.

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value} = \frac{1}{\text{(Command multiply)}} + 100 \quad \text{[Valid data range: 102 to 127]}$$

- When command multiply is 0.5 to 48

Set value = 2 × command multiply [Valid data range: 1 to 96]

Set this parameter to 2 for spindle positioning.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

Set this parameter to 10000 for spindle positioning.

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for each axis.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

Set a grid shift for each axis.

> **NOTE**
> Set this parameter when the analog spindle is used. When the serial spindle is used, set the value to No. 4073.

1851	Backlash compensating value for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR				

[Data type] Bit

CCR Addresses used for chamfering and corner rounding

0 : Address used for chamfering and corner rounding is I or K, not C. In direct drawing dimension programming, addresses ’C, ’R, and ’A (with comma) are used in stead of C, R, and A.

1 : Addresses used for chamfering, corner rounding, and direct drawing dimension programming are C and R without comma. Thus, addresses C cannot be used as the names of axes.

<u>Always set this parameter to “0” for spindle positioning.</u>

	#7	#6	#5	#4	#3	#2	#1	#0
4000					RETRN			

[Data type] Bit

RETRN Reference position return direction of spindle.

0 : CCW (Counter clockwise)

1 : CW (Clockwise)

> **NOTE**
> The direction for spindle orientation (or reference position return) in spindle positioning using a serial spindle is determined by this parameter.

4044	Velocity loop proportion gain in servo mode (High gear)
4045	Velocity loop proportion gain in servo mode (Low gear)

[Data type] Word

[Valid data range] 0 to 32767

This parameter sets a velocity loop proportional gain in servo mode (spindle positioning, rigid tapping, etc.)

NOTE
Set this parameter when serial spindle is used.

4052	Velocity loop integral gain in servo mode (High gear)
4053	Velocity loop integral gain in servo mode (Low gear)

[Data type] Word

[Valid data range] 0 to 32767

This parameter sets a velocity loop integral gain in servo mode (spindle positioning, rigid tapping, etc.)

NOTE
Set this parameter when serial spindle is used.

4056	Gear ratio (HIGH)
4057	Gear ration (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word

[Unit of data] Motor speed per spindle rotation $\times$ 100

[Valid data range] 0 to 32767

These parameters set the gear ration between the spindle and AC spindle motor.

NOTE
Set the gear ration between spindle and AC spindle motor when the spindle positioning is performed with serial spindle. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH1B.

4065	Position gain in servo mode (HIGH)
4066	Position gain in servo mode (MEDIUM HIGH)
4067	Position gain in servo mode (MEDIUM LOW)
4068	Position gain in servo mode (LOW)

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

This parameter sets a servo loop gain in servo mode.
(spindle positioning, rigid tapping, etc.)

> **NOTE**
> When the spindle positioning by a serial spindle is performed, set the position control loop gain in place of parameter No. 4970. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH1B.

	#7	#6	#5	#4	#3	#2	#1	#0
4950	IMB	ESI	TRV			ISZ	IDM	IOR

[Data type] Bit

IOR Resetting the system in the spindle positioning mode

0 : Does not releases the mode.
1 : Releases the mode

IDM The positioning direction for the spindle using a M code is

0 : The positive direction
1 : The negative direction

ISZ When an M code for spindle orientation is specified in spindle positioning:

0 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode, and spindle orientation operation is performed.
1 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode but spindle orientation operation is not performed.

TRV Direction of rotation of spindle positioning

0 : Normal
1 : Reverse

ESI Selection of a spindle positioning specification

0 : The conventional specification is used.
1 : The extended specification is used.

NOTE

The extended specification includes the following two extensions:

- With the conventional specification, the number of M codes for specifying a spindle positioning angle is always 6. With the extended specification, an arbitrary number of such M codes from 1 to 256 can be selected by parameter setting (See parameter No. 4964.)
- The maximum feedrate for spindle positioning (setting of parameter No. 1420) can be extended from 240000 to 269000 (unit: 10 deg/min).

IMB When the spindle positioning function is used, semi–fixed angle positioning based on M codes uses:

0 : Specification A
1 : Specification B

NOTE

In the case of semi–fixed angle positioning based on M codes, three types of spindle positioning operations can occur:

(1) The spindle rotation mode is cleared, then the mode is switched to the spindle positioning mode.
(2) Spindle positioning is performed in the spindle positioning mode.
(3) The spindle positioning mode is cleared, then the mode is switched to the spindle rotation mode.

In the case of specification A:
Operations (1) to (3) are specified using separate M codes.

(1)–Specified using M codes for performing spindle orientation.
(See parameter No. 4960)
(2)–Specified using M codes for specifying a spindle positioning angle. (See parameter No. 4962)
(3)–Specified using M codes for clearing spindle positioning operation. (See parameter No. 4961.)

In the case of specification B:
When M codes for specifying a spindle positioning angle are specified, operations (1) to (3) are performed successively. (See parameter No. 4962.)

4960	M code specifying spindle orientation

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 97

Set an M code to change the spindle rotating mode to the spindle positioning mode. Setting the M code performs the spindle orientation. Spindle positioning can be specified from the next block.

4961	M code releasing the spindle positioning mode

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 97

Set the M code to release the spindle positioning mode and to change the mode to the spindle rotating mode.

4962	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 92

Two methods are available for specifying spindle positioning. One method uses address C for arbitrary–angle positioning. The other use an M code for semi–fixed angle positioning. This parameter sets an M code for the latter method.

- When bit 6 (ESI) of parameter No. 4950=0
  Six M code from M α to M(α+5) are used for semi–fixed angle positioning, when α is the value of this parameter.
- When bit 6(ESI) of parameter No. 4950=1
  Set the start M code in this parameter, and set the number of M codes in parameter No. 4964. Then β M codes from Mα to M(α+β–1) are used for semi–fixed angle positioning.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M (α+1)	2θ	60°
M (α+2)	3θ	90°
M (α+3)	4θ	120°
M (α+4)	5θ	150°
M (α+5)	6θ	180°
⋮	⋮	⋮
M (α+n)	(n+1)θ	

> **NOTE**
> θ represents the basic angular diplacement set in pamrameter No. 4963.

4963	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] deg

[Valid data range] 1 to 60

This parameter sets a basic angular displacement used for semi–fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Data type] Byte

[Unit of data] Integer

[Valid data range] 0, 1 to 255

This parameter sets the number of M codes used for semi–fixed angle positioning using M codes.
As many M codes as the number specified in this parameter, starting with the M code specified in parameter No. 4962, are used to specify semi–fixed angle positioning.
Let α be the value of parameter No. 4962, and let β be the value of parameter No. 4964. That is, M codes from Mα to M (α+β–1) are used for semi–fixed angle positioning.

> **WARNING**
> Make sure that M codes from Mα to M (α+β–1) do not duplicate other M codes.

> **NOTE**
> 1 This parameter is valid when bit 6 (ESI) of parameter No. 4950=1.
> 2 Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M (α+5) are used for half–fixed angle positioning.

4970	Servo loop gain of the spindle

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

Set the servo loop gain of the spindle in the spindle positioning mode.

4971	Servo loop gain multiplier of the spindle for gear 1
4972	Servo loop gain multiplier of the spindle for gear 2
4973	Servo loop gain multiplier of the spindle for gear 3
4974	Servo loop gain multiplier of the spindle for gear 4

[Data type] Word

Set the servo loop gain multipliers of the spindle for gears 1 to 4.

The multipliers are used to convert the amount of the position deviation to the voltage used in the velocity command. Assign the data obtained from the following equation to the parameters.

Loop gain multiplier = 2048000 $\times$ E $\times$ A/L

where;

E: Voltage required to rotate the spindle motor at 1000 min^{-1} in the velocity command

L: Rotation angle of the spindle per one motor rotation (normally 360)

A: Unit used for the detection (degree)

Example) Let E be 2.2 V, L be 360 degrees, and A be 0.088 degrees/pulse.

Loop gain multiplier = 2048000 $\times$ 2.2 $\times$ 0.088/360 = 1101

NOTE
1. When the voltage specified for the spindle motor is 10 V at a spindle speed of 4500 min^{-1}, E is regarded as 2.2 V.
2. The above parameters No. 4970 to No. 4974 are for analog spindle.

Alarm and message

Number	Message	Description
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Or comma (",") was specified with parameter No. 3405#4=1. Modify the program.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering or corner R command, I(K) is commanded for the X(Z) axis. Modify the program.
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	C/H–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program.
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
751	SPINDLE–1 ALARM DETECT (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	SPINDLE–1 MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Caution

CAUTION

1 Feed hold is invalid during spindle positioning.
2 Spindle positioning stops when emergency stop is applied; restart with orientation operation.
3 Dry run, machine lock, and auxiliary function lock are not available during spindle positioning.
4 The spindle positioning function and the serial spindle Cs contour control function cannot be used together. If both functions are specified, positioning has priority.
5 Specify parameter No. 4962 even if semi-fixed angle positioning is not used; otherwise M codes (M00 to M05) do not work.

Note

NOTE

1 Command spindle positioning with an independent block. X- and Y-axis positioning cannot be commanded to the same block.
2 Spindle positioning cannot be done by manual operation.
3 Automatic drift compensation is not effective for spindle positioning. To adjust the amount of drift compensation for each axis, set values manually and adjust the spindle amplifier to minimize the spindle motor rotation at a voltage of 0V. (parameter No. 3731). Insufficient adjustment causes poor positioning accuracy. Drift compensation is not needed with a serial spindle.
4 The machine coordinates for the spindle positioning axis are displayed in pulses units.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	II.9.5	SPINDLE POSITIONING FUNCTION
Series 0*i* Mate–C	OPERATOR’S MANUAL (T series) (B–64134EN)	II.9.4	SPINDLE POSITIONING FUNCTION

9.8 Cs CONTOUR CONTROL

9.8.1 Cs Contour Control

General

The Cs contour control function positions the serial spindle using the spindle motor in conjunction with a dedicated detector mounted on the spindle.

This function can perform more accurate positioning than the spindle positioning function, and has an interpolation capability with other servo axes.

- **Increment system**

Least input increment: 0.001 [deg]
Least command increment: 0.001 [deg]

- **Maximum command value**

±99999.999 [deg]

- **Feedrate**

Rapid traverse rate: 6 to 240000 [deg/min] (parameter no.1420)

Cutting feedrate (feed per minute):
1 to 240000 [deg/min] (rotation axis)

Explanations

The speed of the serial spindle is controlled by the spindle speed control function, while the spindle positioning is controlled by the Cs contouring control function ("spindle contour control"). Spindle speed control rotates the spindle using the velocity command, while the spindle contour control rotates the spindle using the move command.

Switching between spindle speed control and spindle contour control is performed by the DI signal from the PMC.

In the Cs contour control mode, the Cs contour control axis can be operated either manually or automatically, in the same way as normal servo axes.

(For a reference position return, see the relevant description in this section.)

Setting the Cs contour control axis

The axis used for Cs contour control must be set as an axis of the axes controlled by the CNC. Using parameter no. 1023, assign "–1" in the field corresponding to the chosen servo axis. Also set the spindle contour control axis as a rotation axis by setting ROTx of parameter No. 1006#0 and No. 1022.

Only one set of this setting can be used. The spindle that operates under Cs contour control is a serial spindle as the first spindle.

Command Address

The address for the move command in Cs contour control is the axis name specified in parameter no.1020. This address is arbitrary.

When the second auxiliary function is provided, address B cannot be used for the name of the contour axis. For the T series machines, when either address A or C is used for the name of the contour axis, clear CCR (parameter no. 3405#4) to “0”.

Setting Axes that interpolate with Cs contour axis

Up to three servo axes can be specified for linear interpolation against the Cs contour control axis, by setting defined parameters :

- When no servo axis is used for interpolation, specify “0” in parameter nos. 3900, 3910, 3920, 3930, 3940.
- When one or more servo axes are used for interpolation, set the parameter for each as follows :

(1) Assign the axis number (1 to 4) to each of the servo axes used for interpolation in parameter nos. 39n0 (n=0, 1, or 2).

(2) Set the loop gain for each of the servo axes specified in (1) in parameter nos. 39n1, 39n2, 39n3, 39n4. The loop gain must be the position loop gain for the Cs contour control axis or a desired value. Four parameters are provided to correspond to the four gears of the spindle. Use those parameters according to the inputs of the serial spindle clutch /gear signal CTH1A, CTH2A <G70#3, #2>.

> **NOTE**
> The fine acceleration/deceleration function cannot be used between the servo axis and Cs contour control axis.

(3) When the number of servo axes to be used for interpolation is smaller than three, set “0” in remaining parameter nos. 39n0.

Switching spindle speed control/Cs contour control

❑ Switching from spindle speed control to Cs contour control

The serial spindle is put in the Cs contour control mode by setting the DI signal CON (G027#7) to “1”. When the mode is switched while the spindle is rotating, the spindle stops immediately to perform the change.

❑ Switching from Cs contour control to spindle speed control

Turning the DI signal CON (G027#7) to “0” puts the serial spindle in spindle speed control mode. Confirm that the move command for the spindle has been completed, then specify the switch. If it is switched while the spindle is moving, the machine enters interlock, or excessive position deviation alarm occurs.

Reference Position Return of Cs Contour Control Axis

After the serial spindle is switched from spindle speed control to Cs contour control mode, the current position is undefined. Return the spindle to the reference position.

The reference position return of the Cs contour control axis is as follows:

❑ In manual mode

After the serial spindle enters the Cs contour control mode, move the spindle in the direction of the reference position by turning on the feed axis and direction select signal (+Jn (G100) or –Jn (G102)). The spindle starts the movement to the reference position; when it reaches that position, the reference position return completion signal ZPn <F094> turns to “1”.
Turning any feed axis and direction select signal to “1” moves the tool in the reference position direction.

❑ In the automatic mode

After the serial spindle enters the Cs contour control mode, the spindle returns to the reference position when G28 is specified. Under certain conditions, the G00 command returns the spindle to the reference position, depending upon the setting of parameter NRF no. 3700#1:

(i) G00 command

Returning to the reference position using the G00 command differs from using the G28 command or the manual method. The serial spindle can be positioned at any point using the G00 command, while the latter two methods always return the serial spindle to the reference position.

When parameter NRF no. 3700#1 is “0” and the serial spindle is put in the Cs contour control mode, if the G00 command is given before returning the spindle to the reference position, the serial spindle returns to the reference position and indexes it before moving to the commanded position. After positioning at the reference position, the reference position return completion signal ZPn <F094> turns to “1”. When the G00 command is issued after the serial spindle has returned to the reference position at least once, normal positioning operation is executed.

(ii) G28 command

After the serial spindle is put in the Cs contour control mode, issuing the G28 command stops the spindle motor, then moves the spindle to the midpoint. The spindle then returns to the reference position. At this point, the reference position return completion signal ZPn <F094> turns to “1”. When the serial spindle has returned to the reference position once while in the Cs contour control mode, the G28 command positions the spindle at the reference position without moving to the midpoint and ZPn comes on.

❑ Interruption of reference position return

(i) Manual operation

Return to the reference position can be interrupted by resetting, emergency stop, or turning off the feed axis and direction select signal. When the interrupted return operation is resumed, start from the beginning.

(ii) Automatic operation

Return to the reference position can be interrupted by resetting, emergency stop, or feed hold. When the interrupted return operation is resumed, start from the beginning.

Operation of Cs contour control axis (Manual/Automatic)

If a reference position return is performed on the Cs contour control axis, the axis can be operated in the same way as a normal NC axis.

In the spindle speed control mode, on the other hand, it does not operate as the Cs contour control axis, and P/S alarm 197 occurs during automatic operation.

In the spindle speed mode, inhibit manual operation of the Cs contour control axis using the PMC ladder.

Display of Position Error of Cs Contour Control Axis

418	Position deviation amount of 1st spindle

Position deviation amount of the position loop for the 1st spindle.

This diagnostic display shows information obtained from the serial spindle control unit. This diagnosis displays position error of the spindle contour axis during spindle contour control.

The position error can also be checked using a servo error display (DGN of No. 300x) for an axis under Cs contour control.

Disabling fine acceleration/deceleration during Cs contour control mode

During Cs contour control mode, fine acceleration/deceleration is disabled for any axis selected by a PMC signal (CDF1 to CDF4). Using a parameter (bit 6 of parameter No. 3710) can disable fine acceleration/deceleration for the axis (parameter No. 39n0 where n = 0 to 4) that is interpolated with the Cs contour control axis.
Feed–forward is disabled for the Cs contour control axis.

Signal

Spindle contour control change signal CON <G027#7>

[Classification] Input signal

[Function] This signal specifies that the first serial spindle be switched between the spindle speed control and Cs contour control modes.

When this signal turns to ”1”, the spindle mode changes from speed control to Cs contour control.

If the spindle is moving at the time of the change, it stops immediately. Turning the signal to “0” changes the spindle mode from Cs contour control back to speed control.

Spindle contour control change completion signal FSCSL <F004#1>

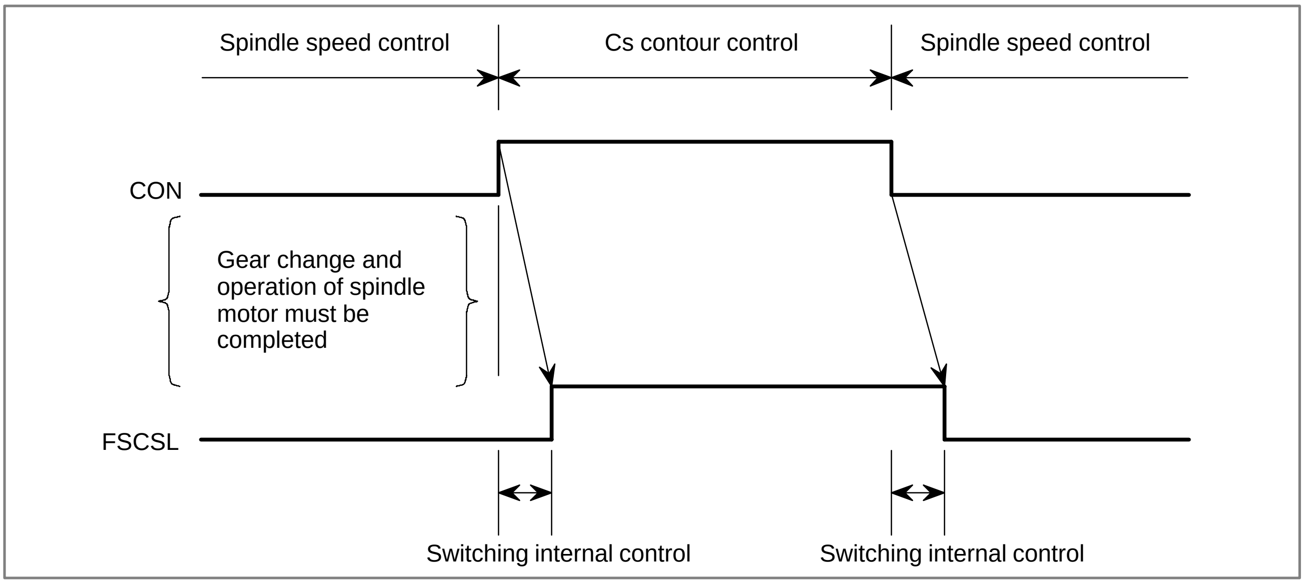
[Classification] Output signal

[Function] This signal indicates the axis is under Cs contour control.

[Output condition] Spindle speed control mode → 0

Cs contour control mode → 1

Time Chart



> **NOTE**
> Any mechanical gear change needed and inputs for GR1, GR2, CTH1A, and CTH2A must be completed before the CON signal selects Cs contour control mode.
> A servo excessive error may be generated if the spindle motor is not ready for operation. (Signal SRVA, SFRA <G070#4, #5> or other required signals must be appropriately processed on the machine side).

Other signals

Gear select signal (Input) GR1, GR2, <G028#1, #2> Gear select signal (Output) GR3O, GR2O, GR1O <F034#2, #1, #0> (M series)

Refer to 9.3 "Spindle speed Control".

Clutch/Gear signal (Serial spindle) CTH1A, CTH2A <G070#3, #2>

Refer to the manual of serial spindle.

These signals determine what parameter (loop gain, etc.) to be used for each gear position.

CTH1A and CTH2A are the gear select signals for the serial spindle, but GR1 and GR2 must also be set. Do not change these signals while in the Cs contour control mode.

Relationship between gears selected and spindle gear select signals

CNC side							Serial spindle		
T/M series with CSSC			M series without CSSC						
GR2	GR1	Gear selection	GR3O	GR2O	GR1O	Gear selection	CTH1A	CTH2A	Gear selection
0	0	1st stage	0	0	1	1st stage	0	0	1st stage
0	1	2nd stage	0	1	0	2nd stage	0	1	2nd stage
1	0	3rd stage	1	0	0	3rd stage	1	0	3rd stage
1	1	4th stage					1	1	4th stage

CSSC: Constant surface speed control

NOTE

1 When the M series does not include the constant surface speed control, and parameter No. 3706#4 GTT=0, GR1 and GR2 do not need to be input. Input CTH1A and CTH2A when gears are changed using GR1O, GR2O and GR3O.

2 The above combination of clutch/gear signals CTH1A and CTH2A is an example.
The serial spindle gear is selected by CHT1A and CHT2A independently of gear selection on the CNC side. So, enter necessary signals, and set the corresponding serial spindle parameters.

Cs contour control axis reference position return completion signal ZPx <F094>

[Classification] Output signal

[Function] This signal indicates that a reference position return has been made for the Cs contour control axis.

[Output condition] If a manual reference position return or automatic reference position return by G28 is performed during the Cs contour control mode, this signal becomes logical 1 when the Cs contour control axis reaches the reference position.

Signals on manual operation

Feed axis and direction select signal +Jn, −Jn <G100, G102> (Input)
Manual handle feed axis select signal HSnA, HSnB, HSnC, HSnD <G018, G019> (Input) (Refer to respective items in this manual)

The Cs contour control axis can be manually operated in the same way as normal servo axes, except for a manual reference position return. In the spindle speed control mode, however, manual operations for the Cs contour control axis must be inhibited using the PMC ladder, etc.

Cs contour control mode fine acceleration/ deceleration disable signals CDF1 to CDF4 <G0127#0 to G0127#3>

[Classification] Input signal

[Function] These signals select the axis for which fine acceleration/deceleration is to be disabled during Cs contour control mode.

[Operation] 0: No fine acceleration/deceleration is disabled for the corresponding axis during Cs contour control mode.

1: Fine acceleration/deceleration is disabled for the corresponding axis during Cs contour control mode.

*1 These signals take effect immediately when they are entered. Do not use the signals during axis movement, or the machine will undergo shocks.

*2 These signals take effect during Cs contour control mode if parameter CSL (bit 6 of parameter No. 3710) = 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027	CON							
G028						GR2	GR1	
G070	MRDYA		SFRA	SRVA	CTH1A	CTH2A		
G127					CDF4	CDF3	CDF2	CDF1

	#7	#6	#5	#4	#3	#2	#1	#0
F034						GR3O	GR2O	GR1O
F044							FSCSL	
F094					ZP4	ZP3	ZP2	ZP1

Parameter

The following describes major parameters.

In addition, parameters such as axis feedrate, acceleration/deceleration, and display can be used. Also, digital servo parameters (No. 2000 or later) for the Cs contour axis are not required to be set.

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

[Data type] Bit axis

Type of controlled axis

0 : Linear axis
1 : Rotation axis

NOTE
Inch/metric conversion cannot be made to the rotation axis. The machine coordinate values are rounded in 0 to 360 deg. Automatic reference position return (G28, G30) is made in the manual reference position return direction and the move amount does not exceed one rotation.

Set 1 as the rotation axis to the Cs contour control axis.

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

Axis name	Set value	Axis name	Set value	Axis name	Set value	Axis name	Set value
X	88	U	85	A	65	E	69
Y	89	V	86	B	66	–	–
Z	90	W	87	C	67	–	–

NOTE
1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
2 The same axis name cannot be assigned to more than one axis.
3 When the optional function of the second auxiliary function is provided, however, the address (B for the T series or the address specified by parameter No. 3460 for the M series) for the second auxiliary function command cannot be used. In the T series, when address A or C is used, set parameter CCR (No. 3405#4) to 0.
4 If you use letter E as an axis name in format F10/11, be sure to use address F in the major–axis direction lead command for equal–lead threading (G32).

Any axis name can be used for Cs contour control axis except for above limitation.

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

Set 0 to the Cs contour control axis.

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

Set –1 as the number of servo axis to the Cs contour control axis.

1420	Rapid traverse rate for each axis

[Data type] Word axis

[Unit of data] 1 deg/min

[Valid data range] 30 to 240000 (IS–A, IS–B)
6 to 100000 (IS–C)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/ deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used in linear acceleration/deceleration in rapid traverse for each axis.

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value}=\frac{1}{(\text{Command multiply})}+100 \text{ [Valid data range: 102 to 127]}$$

- When command multiply is 0.5 to 48

Set value = 2 × command multiply [Valid data range: 1 to 96]

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for each axis.

1851	Backlash compensation value used for rapid traverse for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] −9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Data type] Bit

NRF The first positioning command by G00 after the serial spindle is switched to Cs axis contouring control performs:

0 : Positioning after returning to the reference position.
1 : Normal positioning

	#7	#6	#5	#4	#3	#2	#1	#0
3710		CSL						

[Data type] Bit

CSL In Cs contour control mode, fine acceleration/deceleration is disabled for:

0 : An axis selected by the signal (CDFn <G0127>) issued from the PMC. (n = 1 to 4)
1 : An axis for which interpolation is performed with the Cs contour controlled axis (parameter No. 39n0). (n = 0 to 2)

3900	The number of servo axis that interpolates with Cs contour control axis

[Data type] Byte

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (1st group)

NOTE
Set 0 when there is no servo axis that interpolates with Cs contour control axis.

3901	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3902	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3903	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3904	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (1st group)

3910	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (2nd group)

NOTE
When there is no servo axis or only one servo axis that interpolates with Cs contour control axis, set this parameter to 0.

3911	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3912	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3913	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3914	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (2nd group)

3920	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (3rd group)

> **NOTE**
> When there is no servo axis or less than three servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3921	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3922	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3923	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3924	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (3rd group)

3930	Number of servo axis that interpolates with Cs contour control

[Data type] Byte

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (4th group)

> **NOTE**
> Set this parameter to 0.

3931	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3932	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3933	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3934	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (4th group)

3940	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (5th group)

NOTE
Set this parameter to 0.

3941	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3942	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3943	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3944	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (5th group)

4056	Gear ratio (HIGH)
4057	Gear ratio (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word axis

[Unit of data] (Number of motor rotations to one spindle rotation) × 100

[Valid data range] 0 to 32767

Set the gear ratio between spindle and AC spindle motor.

> **NOTE**
> For which gear ratio is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4069	Position gain at Cs contour control (High gear)
4070	Position gain at Cs contour control (Medium High gear)
4071	Position gain at Cs contour control (Medium Low gear)
4072	Position gain at Cs contour control (Low gear)

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 32767

Set the position gain at Cs contour control.

> **NOTE**
> For which position gain is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4135	Grid shift value at Cs contour control

[Data type] Two–word

[Unit of data] 1 pulse unit (360000 p/rev)

[Valid data range] –360000 to +360000

Set the number of pulses from an issue of one–rotation signal to the machine zero point in Cs contour control.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs contour control axis when the signal CON(G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
751	First spindle alarm detection (AL–XX)	In a system with serial spindles attached, this is a warning alarm message that indicates an alarm on the spindle amplifier side to be displayed on the CRT of the NC. An alarm on the spindle amplifier side is displayed on the spindle amplifier, as AL–XX (where XX represents a number). This alarm number (XX) is displayed by latching the spindle alarm number detected by the CNC as the cause of this alarm.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning,rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Warning

WARNING

In the spindle contour control mode, do not switch the spindle gears. When the gears need to be changed put the system in the spindle speed control mode first.

Note

NOTE

In the T series machines, the spindle contour control function and the spindle positioning function cannot be used at the same time. If both functions are specified simultaneously, the spindle positioning function takes precedence.

Reference item

FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B–65280EN)	2.4	Cs Contour Control

9.8.2 Cs Axis Coordinate Setup Function

The machine loses its current position when the serial spindle is switched from spindle rotation control to Cs contour control.
Setting the Cs axis coordinate setup request signal CSFI <G274#4> to ’1’ enables the machine to set up its current position without making a reference position return.

> **NOTE**
> This function stays in effect until the power is turned off after the power is turned on and a reference position return of the Cs contour axis is made.
> This function is part of the ”Cs axis contour control function.”

Setting

This function is put in effect by setting the CSF parameter (bit 2 of parameter No. 3712) and the CSPTRE parameter (bit 5 of parameter No. 4353) to 1.
(Using this function requires resetting the RFCHK3 parameter (bit 7 of parameter No. 4016) to 0.)
The M code for setting the Cs axis contour control coordinate setup request signal CSFI <G274#4> to ’1’ should be set up as buffer inhibit M codes (parameter Nos. 3411 to 3432).

Cs axis coordinate setup procedure

(1) Trigger the M code to put the machine in the Cs contour control mode.
(2) Make sure that the Cs axis origin setup state signal CSPENA is ’1’, and set the Cs axis coordinate setup request signal CSFI1 <G274#4> to ’1’.
(3) The absolute, relative, and machine coordinates of the Cs axis are incremented for coordinate system setup.
(4) Once the coordinate system is set up, the reference position setup signal ZRFx <F120#0 to F120#3> becomes ’1’. So, reset the Cs axis coordinate setup request signal CSFI1 <G274#4> to ’0’.
(5) Perform FIN processing for the M code.

If an attempt to set up Cs axis coordinates fails

If an attempt to set up Cs axis coordinates fails, the P/S alarm 5346 is issued.

Also, the Cs axis coordinate setup alarm signal CSFO1 <F274#4> becomes ’1’. So, reset the Cs axis coordinate setup request signal CSFI1 <G274#4> to ’0’.
Resetting the Cs axis coordinate setup request signal CSFI1 <G274#4> to ’0’ causes the Cs axis coordinate setup alarm signal CSFO1 <F274#4> to be reset to ’0’.

The following conditions can make the attempt to setup the Cs axis coordinates fail.

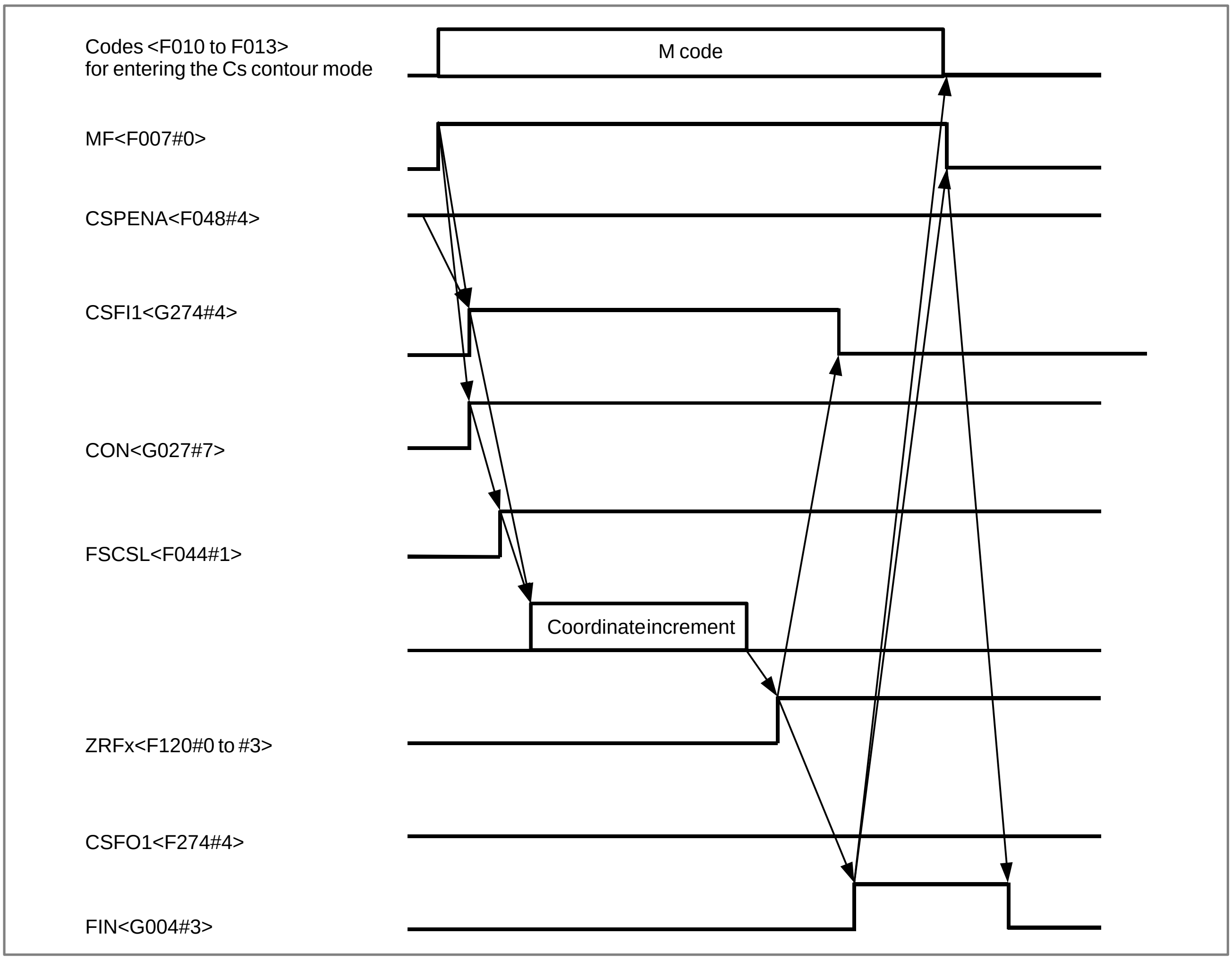
– The Cs axis origin setup state signal CSPENA is ’0’.
– The Cs axis is under synchronization control.
– The machine is brought to an emergency stop during coordinate setup.
– The Cs axis motor is de–energized during coordinate setup (such as servo alarm or servo off).
– Bidirectional pitch error compensation is in effect for the Cs axis.

Clearing P/S alarm 5346

P/S alarm 5346 can be cleared only after the reference position of the Cs axis has been set up.
If P/S alarm 5346 has been issued, perform a manual reference position return for the Cs axis. Once the reference position has been setup, a reset clears P/S alarm 5346.
In addition, once the machine has exited the Cs contour control mode, a reset clears P/S alarm 5346.

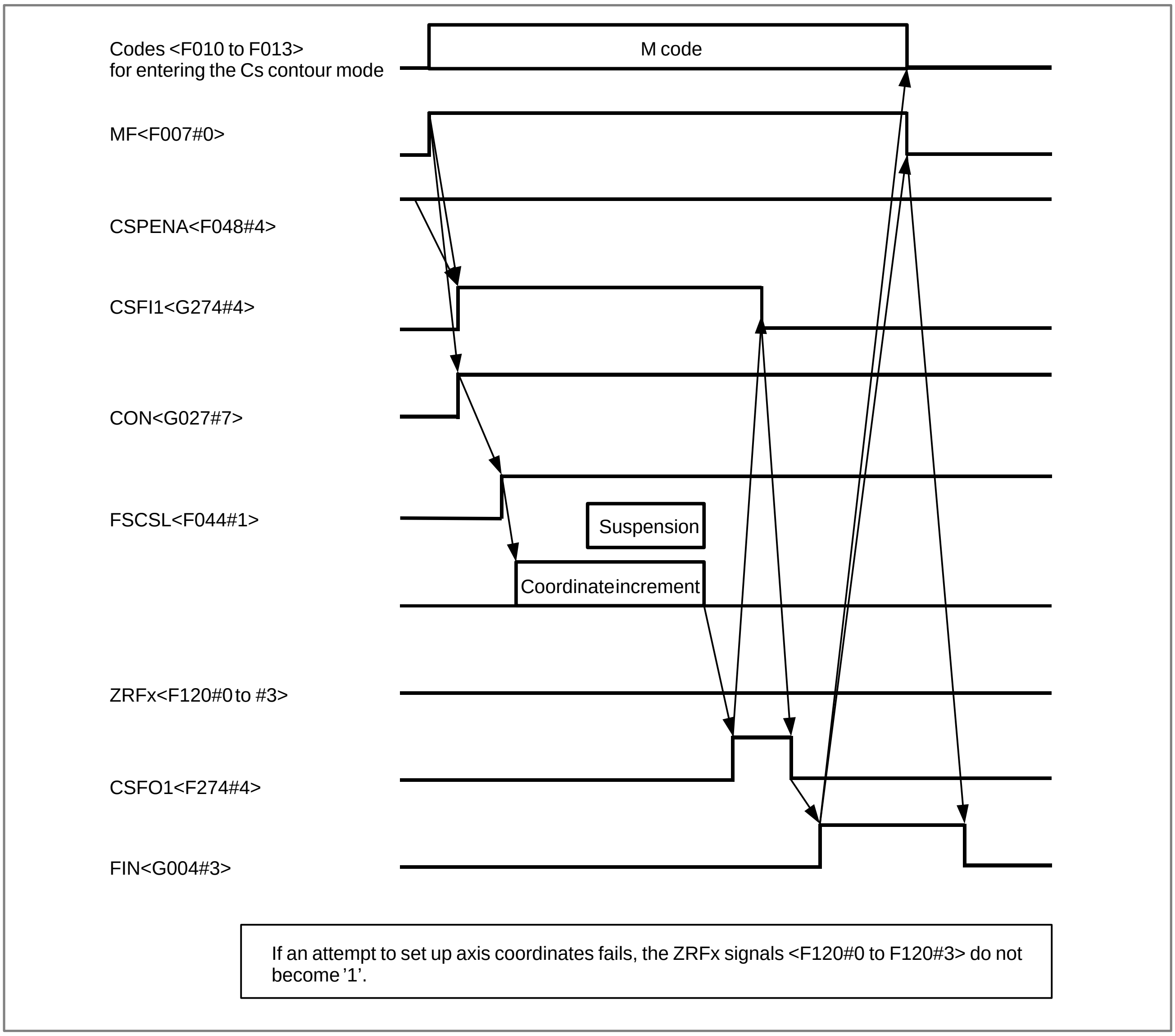
Sequence (timing chart)

When Cs axis coordinate setup is normally completed:



The Cs axis coordinate setup function is valid only during Cs contour mode. If the Cs axis coordinate setup request signal CSFI1 <G274#4> is set to ’1’ during any other mode, it is ignored.

If an attempt to set up Cs axis coordinates fails (because of suspension by an emergency stop, for example)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRN

[Data type] Bit

ZRN If no reference position return has been made, an attempt to start automatic operation:

0 : Is responded with an alarm (P/S 224).
1 : Is responded with no alarm.

To use the Cs axis coordinate setup function, it is recommended to specify ZRN = 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Data type] Bit

NRF Once the Cs contour control mode has been entered, the first positioning command G00:

0 : Makes a reference position return.
1 : Performs normal positioning.

To use the Cs axis coordinate setup function, it is recommended to specify NRF = 1.

	#7	#6	#5	#4	#3	#2	#1	#0
3712						CSF		

[Data type] Bit

CSF If the origin has been set up, automatic setting of the machine and work coordinates according to the machine position of the spindle during Cs contour control mode is:

0 : Disabled.
1 : Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
4353			CSPTRE					

[Data type] Bit axis

CSPTRE Cs axis position data transfer is:

0 : Disabled.
1 : Enabled.

To use this function, reset the RFOHK3 parameter (bit 7 of parameter No. 4016) to 0.

Signal

Cs axis coordinate setup request signal CSFI1 <G274#4>

[Classification] Input signal

[Function] The Cs axis coordinate setup function is executed.

[Operation] If this signal is ”1”, the control unit behaves as follows:

If the signal is ”1” during Cs contour control mode, the work and machine coordinates are set up according to the machine position of the Cs axis.
If this signal is ”0”, the control unit behaves as follows:
The Cs axis coordinate setup alarm signal CSFO1 <F274#4> is reset to ’0’.

Cs axis coordinate setup alarm signal CSFO1 <F274#4>

[Classification] Output signal

[Function] This signal indicates that Cs axis coordinate setup has not normally been completed.

[Output condition] The signal becomes '1' under the following condition:

– Cs axis coordinate setup is not normally completed.

The signal becomes '0' under the following conditions:

– The Cs contour control mode is exited.

– The corresponding Cs axis coordinate set request signal CSFI1 <G274#4> becomes '0'.

Cs axis origin setup status signal CSPENA <F048#4>

[Classification] Output signal

[Function] This signal indicates that it is possible to perform Cs axis coordinate setup processing.

[Output condition] The signal becomes '1' under the following condition:

- A reference position return is normally completed during Cs contour control mode.

The signal becomes '0' under the following condition:

- A spindle alarm is detected or the spindle motor rotates faster than the maximum allowable rotation speed (No. 4020).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G274				CSFI1				
F048				CSPENA				
F274				CSFO1				

Alarm and message

Number	Message	Description
5346	RETURN TO REFERENCE POINT	The Cs coordinate setup function was suspended. Make a manual reference position return. 1. An attempt was made to perform Cs axis coordinate setup for the Cs axis for which CSPEN = '0'. 2. No position information has been transferred from the spindle amplifier. 3. The machine was brought to an emergency stop during coordinate setup. 4. Cs axis coordinate setup was started for the Cs axis for which bidirectional pitch error compensation was in effect. 5. Cs axis coordinate setup was started for the Cs axis whose servo was off.

Caution

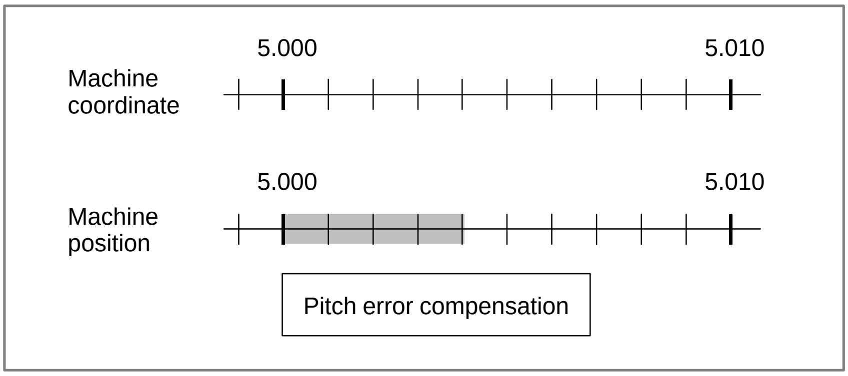
CMR

If the CMR is greater than 1 (parameter No. 1820 > 2), the Cs axis may move through several pulses' worth of distance. This is intended to align the Cs axis machine position to the least command increment. (1 to 9 pulses in terms of detection units if CMR = 10.)

Stored pitch error compensation

If the Cs axis machine position meets the following condition, the Cs axis moves by a pitch error compensation during coordinate setup.
|Nearest pitch compensation point – Cs axis machine position|
< pitch error compensation

Example:



Assuming that a pitch error compensation of +0.004 is specified for a pitch error compensation interval of 10.000 degrees and a machine coordinate of 5.000, the Cs axis will not stop at a machine position of 5.000 to 5.003.

Machine coordinate 4.999 → 5.000 → 5.001
Machine position 4.999 → 5.004 → 5.005

Whereas the Cs axis may stop at a machine position of 5.000 to 5.003 if the Cs contour control mode is entered after rotation in the spindle speed control mode.
In this case, the relationship between the machine coordinates and machine position is adjusted by shifting the machine position of the Cs axis by a pitch error compensation.

Synchronization control

If the Cs axis coordinate setup request signal CSFI1 <G274#4> is set to ’1’ during synchronization control, P/S alarm 5346 occurs.
Be sure to set up the coordinates before starting synchronization control.

Manual handle interrupt

Manual handle interrupts are disabled during Cs axis coordinate setup.

External machine coordinate system shifting

External machine coordinate system shifting is disabled for the axis that is undergoing Cs axis coordinate setup.

Mirror image

The mirror image signals MI1 to MI4 <G106> and a mirror image based on the MIRx parameter <bit 0 of parameter No. 0012> are disabled for the Cs axis coordinate setup function.

Bidirectional pitch error compensation

This function does not support bidirectional pitch error compensation. P/S alarm No. 5346 is issued if an attempt is made to perform Cs axis coordinate setup for the axis for which bidirectional pitch error compensation is in effect.

Reference item

FANUC AC SPINDLE MOTOR αi/βi series PARAMETER MANUAL (B–65280EN)	2.4	Cs Contour Control

9.9 MULTI–SPINDLE CONTROL

General

In addition to the conventional (first) spindle, second and third spindles can be controlled. These additional spindles allow two-stage gear changes. A single S code is used to command to any of these spindles. The spindle to be commanded by S code is determined by a signal from the PMC. The second and third spindle can change gears in 2 stages.

Also, the maximum clamp spindle speed can be individualy set to each spindle to clamp the spindle speed of each spindle (set by parameters No. 3772, 3802, 3822).

When the second spindle is used, a second position coder interface channel is added. Selection between 1st position coder to 4th position coder is made by a signal from PMC. (The conventional and additional position coders are referred to as the first position coder and second position coder, respectively, throughout the remainder of this discussion.)

The signal from the PMC selects one of the first, second and third position coders.

The spindle serial output function is required to use multi-spindle control.

If the third spindle (serial spindle) are needed, specify the third spindle serial output function.

Difference in multi–spindle control between the M and T series

- For the M series, multi–spindle control is possible only when spindle gear selection type T is selected (when the constant surface speed control is provided, or when GTT (bit 4 of parameter No. 3706) is set to 1).
- For the M series, rigid tapping spindle selection signals RGTSP1 and RGTSP2 (G061#4 and #5, when bit 7 of parameter No. 5200 is set to 1) cannot be used. For details of rigid tapping, see Section 9.10.

Control

Two multi-spindle control methods are available. Type A allows the SIND function (controlling the spindle motor speed based on the PMC) to be used only for the first spindle. Type B allows the SIND function to be used for each of the two spindles independently.

Basic control (Common to TYPE-A and TYPE-B)

An S command is sent as a speed command to each spindle selected, using a spindle selection signal (SWS1 to SWS3 <G027#0 to #2>). Each spindle rotates at the specified speed. If a spindle is not sent a spindle selection signal, it continues to rotate at its previous speed. This allows the spindles to rotate at different speeds at the same time.

Each spindle also has a spindle stop signal (*SSTP1 to *SSTP3 <G027#3 to #5>) to stop its rotation; an unused spindle can still be stopped.

There is a spindle enable signal to control each spindle; ENB <F001#4> controls the first spindle, while ENB2 and ENB3 <F038#2 and #3> control the second and third spindles, respectively.

The PMC signal PC2SLC <G028#7> is used to select between the first and second position coders.

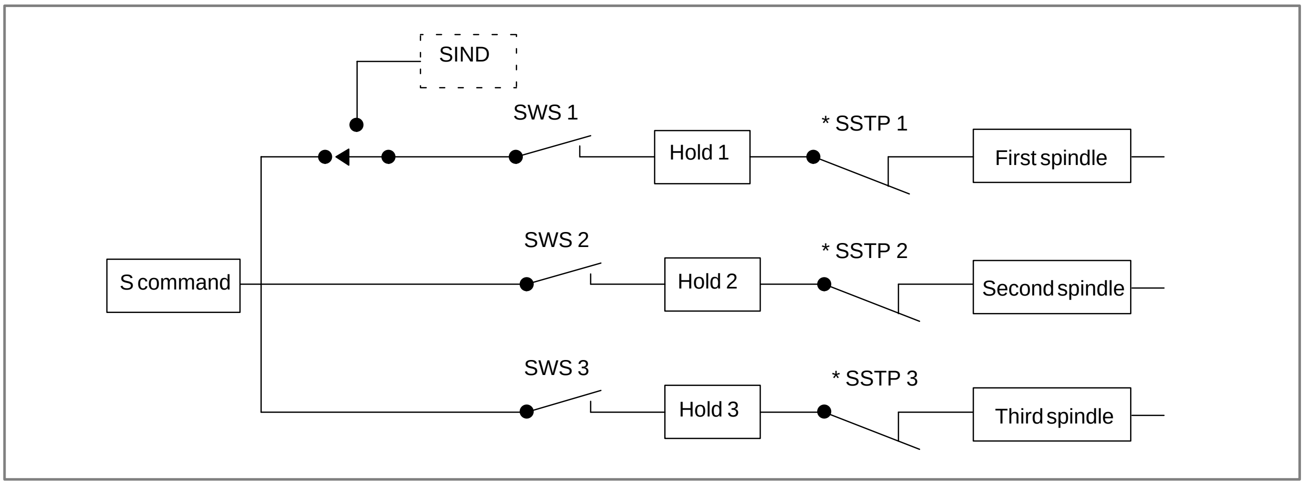
Multi-spindle control (TYPE-A)

When parameter MSI (No. 3709#2)=0, TYPE-A is used.

When the first spindle is selected with the SWS1 signal, the SIND signal <G033#7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I <G0033#3 to G0032#0> are used to set that spindle’s analog voltage. These signals do not affect the second to third spindles.

The PMC-based polarity (rotation direction) control signals SGN and SSIN <G033#5,#6> will function for any spindle selected by SWS1 to SWS3.

The concept of Type A multi-spindle control is outlined below.



> **NOTE**
> When using an analog spindle, the third spindle is selected. In this case, the second spindle cannot be used.

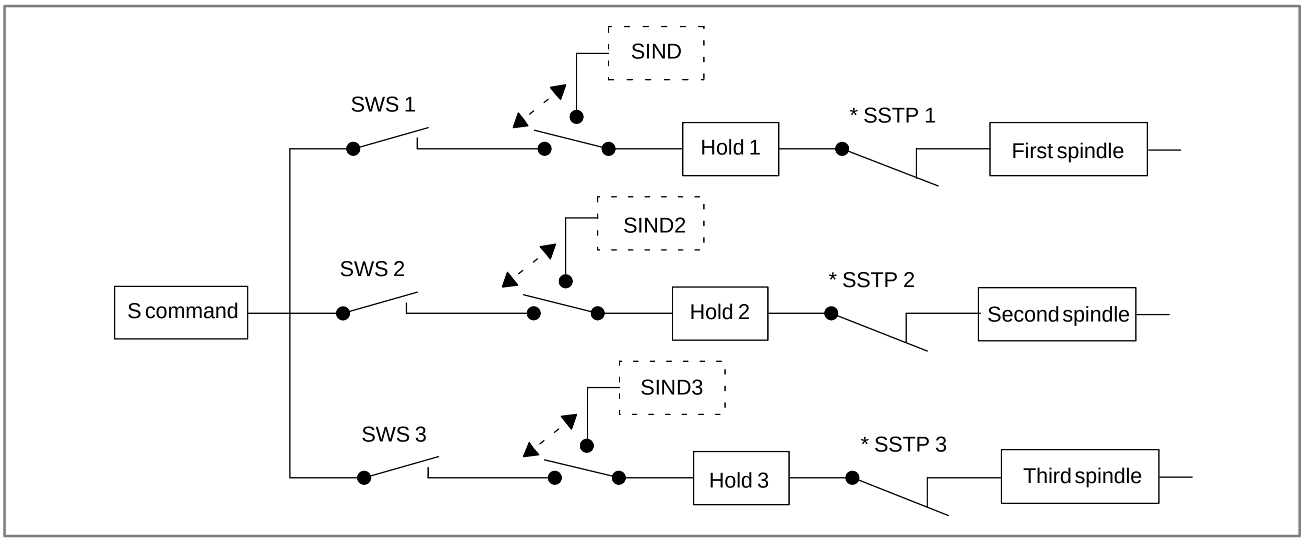
Multi-spindle control (TYPE-B)

Select Type B control by setting parameter MSI No. 3709#2 to “1”.

Each spindle has its own SIND, SSIN and SGN signals.
Each of these signals functions regardless of selection state of the spindle selection signals (SWS1 to SWS3).

When either the spindle selection signals (SWS1 to SWS3) or the SIND signal for the first, second, or third spindle is set to “1”, the polarity control signals SSIN, SGN will function.

The concept of Type B multi-spindle control outlined below.



NOTE
When using an analog spindle, the third spindle is selected. In this case, the second spindle cannot be used.

Spindles to be controlled

In multi-spindle control, the first spindle is the first serial spindle, the second spindle is the second serial spindle, and the third spindle is an analog spindle.
The second and third spindles cannot be connected at the same time.

Connection of spindle

Spindle configuration when multi-spindle control is used:

Necessary parameter	· Multi-spindle control · Spindle serial output · Parameter SS2 (No. 3701#4) = 1 (to use second spindle) · Spindle analog output (to use third spindle)
Connection of each spindle	First spindle (JA7B) → Connect on JA41 (CNC main CPU board). Second spindle (JA7B) → Connect on JA7A (first spindle control unit). Third spindle → Connect on JA40 (CNC main CPU board).
Connection of each position coder	First position coder: Feedback information obtained by position coder or equivalent sensor connected to first spindle control unit is fed to CNC via serial interface. Second position coder: Feedback information obtained by position coder or equivalent sensor connected to second spindle control unit is fed to CNC via serial interface through 1st spindle control unit Note) When second spindle is not connected, second position coder cannot be used.

For detailed information about serial spindle connection, refer to the manuals on the serial spindles.

Relationship with other functions

- **Constant surface speed control**

The control function for keeping the surface speed constant can be used with any of the three spindles, if the spindle speed is within the range allowable for this function. When the position coder is required, it can be installed on the 1st and 2nd spindle. The spindle selection signal (SWS 1–3) for the spindle must stay set at “1” during machining using this function.

- **Spindle speed fluctuation detection**

If multi–spindle control is combined, multiple position coders can be selected. To carry out spindle speed fluctuation detection, select the first spindle (first position coder). Monitor the states of the second position coder selection signal (PC2SLC) and spindle selection signals (SWS1 and SWS2). The spindle speed fluctuation detection cannot be executed with the second position coder.

- **Actual spindle speed output**

The actual spindle speed output function conveys speed information obtained from the position coder specified by the second position coder selection signal (PC2SLC) to the PMC.
When the parameter HSO (No. 3709#5)=1, the difference of the feedback pulses between the 1st and 2nd position coder can be output irrespective of the state of 2nd position coder selection signal (PC2SLC).

- **Spindle positioning or Cs contour control**

When the spindle motor is used for positioning, as in the case of spindle positioning or Cs contour control, the first spindle will allows function as the positioning spindle. Switching to the positioning mode and positioning command are possible irrespective of the state of the selection signal of the first spindle (SWS1). This means that the first spindle cannot be controlled as a spindle in positioning mode, but the second spindle can be controlled as usual.

- **Polygon turning (T series)**

Polygon turning rotates a tool axis in phase with the spindle. To perform polygon turning when multi-spindle control issued, select the spindle and the position coder associated with the spindle.
(First and second position coders can be selected.)

- **Spindle synchronization**

During spindle synchronization, the second spindle operates in phase with the first spindle. Multi-spindle control for the first spindle can be used during synchronization control, but multi-spindle control for the second spindle is disabled.

- **Rigid tapping**

Rigid tapping can be performed by selecting one spindle from the first and second spindles by means of the spindle selection signals (SWS1 and SWS2). There are certain restrictions:

- Set the SWS 1 to 2 signals before directing rigid tapping;
- Do not switch the SWS 1 to 2 signals during rigid tapping; and
- Use the appropriate ENB signal (either ENB or ENB2) for the selected spindle as the ENB signal for the rigid tapping PMC sequence.

The spindles not used for rigid tapping can be rotated at a speed specified before rigid tapping starts, or can be stopped.

Signal

Spindle Selection Signal SWS1, SWS2, SWS3 <G027#0, #1, #2>

[Classification] Input signal

[Function] Controls whether S command specified to the NC is output to the spindle or not in multi-spindle.

SWS1 1 : Outputs a speed command to the first spindle.
0 : Outputs no speed command to the first spindle.

SWS2 1 : Outputs a speed command to the second spindle.
0 : Outputs no speed command to the second spindle.

SWS3 1 : Outputs a speed command to the third spindle.
0 : Outputs no speed command to the third spindle.

Individual spindle stop signal *SSTP1, *SSTP2, *SSTP3 <G027#3, #4, #5>

[Classification] Input signal

[Function] Effective only to multi-spindle, each spindle can be stopped by this signal.

***SSTP1** 1 : Does not set 0 min^{-1} for output to the first spindle.
0 : Sets 0 min^{-1} for output to the first spindle.

***SSTP2** 1 : Does not set 0 min^{-1} for output to the second spindle.
0 : Sets 0 min^{-1} for output to the second spindle.

***SSTP3** 1 : Does not set 0 min^{-1} for output to the third spindle.
0 : Sets 0 min^{-1} for output to the third spindle.

Gear select signal GR21 <G029#0> GR31 <G029#2>

[Classification] Input signal

[Function] Gear selection signal for second spindle when multi–spindle is equipped (2–stage). Use GR1 and GR2 <G028#1, #2> for the 1st spindle and up to 4–stage gears can be used.

GR21 1 : Selects the second-stage gear for the second spindle.
0 : Selects the first-stage gear for the second spindle.

GR31 1 : Selects the second-stage gear for the third spindle.
0 : Selects the first-stage gear for the third spindle.

2nd position coder selection signal PC2SLC <G028#7> PC3SLC <G026#0> PC4SLC <G026#1>

[Classification] Input signal

[Function] These signals select the position coder of the serial spindle used for multi–spindle control.

To select a position coder, enter the signals as follows.

Position coder to be selected	PC2SLC
First position coder	0
Second position coder	1
Third position coder	0
Four position coder	0

When the second position coder is not installed, do not switch this signal. Always set “0” for first position coder.

Spindle enable signal ENB2<F038#2> ENB3<F038#3>

[Classification] Output signal

[Function] These signals inform PMC of whether or not to perform output to the second to third spindles in multi–spindle control.

The signals are used as a condition to stop the analog spindle, and are also used for a PMC ladder sequence that is associated with rigid tapping. (See Section 9.10.)

[Output condition]

ENB2 1 : Enables output of command other than 0 to the second spindle control unit.
0 : Outputs 0 to the second spindle control unit.

ENB3 1 : Enables output of command other than 0 to the third spindle control unit.
0 : Outputs 0 to the third spindle control unit.

Spindle control signal by PMC

First spindle	SIND, SSIN, SGN <G033#7, #6, #5> R01I to R12I <G032#0 to G033#3>
Second spindle	SIND2, SSIN2, SGN2 <G035#7, #6, #5> R01I2 to R12I2 <G034#0 to G035#3>
Third spindle	SIND3, SSIN3, SGN3 <G037#7, #6, #5> R01I3 to R12I3 <G036#0 to G037#3>

[Classification] Input signal

[Function] The spindle motor of each spindle can be controlled by issuing commands from the PMC. The speed command and polarity (rotation direction) of the spindle motor can be controlled. Usually, CNC commands are used to specify a speed and polarity. By using these signals, whether commands issued from the CNC or PMC are to be used for this control can be selected. Even when multi–spindle control is not being used, the signals can be used to control the second to third spindles.

When multi–spindle control is being used, and TYPE–A is selected (bit 2 (MSI) of parameter No. 3709 is set to 0), the signals for the second to third spindles cannot be used.

For details of each signal, see Section 15.4.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027			*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029		*SSTP				GR31		GR21
G031								
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3

	#7	#6	#5	#4	#3	#2	#1	#0
F038					ENB3	ENB2		

Parameter

The parameters for the 1st spindle and the 1st position coder are the same as usual. This section describes the parameters which are added by the multi–spindle control function.

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit

SS2 In serial spindle control, the second spindle is:

0 : Not used.

1 : Used.

NOTE

This parameter is valid, when the spindle serial output function is provided and parameter ISI(bit 1 of parameter No.3701)is 0.

1 Confirmation of connection of the second serial spindle amplifier, and communication with it

2 Control of the second spindle during asynchronous control (SIND2)

When this parameter is set, it is also necessary to set the serial spindle parameter for the second spindle.

Parameter setting SS2 (No.3701#4)	Serial spindles to be used
0	First spindle only
1	First and second spindles

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Data type] Bit

EMS Multi–spindle control is

0 : Used
1 : Not used

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PG2	PG1
				GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ration of spindle to first position coder
For the setting, see the description of parameter No.3707.

GTT Selection of a spindle gear selection method

0 : Type M.
1 : Type T.

NOTE
1 Type M:
The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in parameters. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.
2 When the constant surface speed control is selected, type T is selected, regardless of whether this parameter is specified.
3 When type T spindle gear switching is selected, the following parameters have no effect:
No. 3705#2 SGB, No. 3751, No. 3752, No. 3705#3 SGT, No. 3761, No. 3762, No. 3705#6 SFA, No. 3735, No. 3736
However, parameter No. 3744 is valid.

		#7	#6	#5	#4	#3	#2	#1	#0
3707								P22	P21

PG2, PG1 (Bits 1 and 0 of parameter No.3702)
Gear ratio of spindle to first position coder

P22, P21 Gear ratio of spindle to second position coder

Magnification	PG2, P22	PG1, P21
$\times 1$	0	0
$\times 2$	0	1
$\times 4$	1	0
$\times 8$	1	1

$$\text{Magnification}=\frac{\text{(Number of spindle revolutions)}}{\text{(Number of position coder revolutions)}}$$

> **NOTE**
> Second position coder is enabled when the multi–spindle control is selected.

		#7	#6	#5	#4	#3	#2	#1	#0
3709							MSI		

MSI In multi–spindle control, the SIND signal is valid

0 : Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective)

1 : For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal).

3772	Maximum spindle speed

[Data type] Word

[Unit of data] $\min^{-1}$

[Valid data range] 0 to 32767

This parameter sets the maximum spindle speed.
When a command specifying a speed exceeding the maximum speed of the spindle is specified , or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

> **WARNING**
> 1 When 0 is set in this parameter, the speed of the spindle is not clamped.
> 2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the speed of the spindle is not clamped.

NOTE

1 When the constant surface speed control option is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

2 When the multi–spindle control option is selected, set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

3802	Maximum speed of the second spindle

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Parameter sets the maximum speed for the second spindle.
When a command specifying a speed exceeding the maximum speed of the spindle is specified, or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

WARNING

1 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used.
When 0 is set in parameter No. 3772, the spindle speed is not clamped.

2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE

1 This parameter is valid when the multi–spindle control is selected.

2 When the constant surface speed control is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is specified.

3811	Maximum spindle speed for gear 1 of the second spindle
3812	Maximum spindle speed for gear 2 of the second spindle

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the second spindle.

> **NOTE**
> These parameters are used only with multi–spindle control.

3820	Data for adjusting the gain of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] 0.1%

[Valid data range] 700 to 1250

Set the data used for adjusting the gain of the analog output of the third spindle speed.

> **NOTE**
> This parameter is used only with multi–spindle control.

3821	Offset voltage compensation value of the analog output of the third spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset voltage compensation value of the analog output of the third spindle speed.

1) Set 0 (standard setting) to this parameter.
2) Command a spindle speed that makes the spindle speed analog output 0.
3) Measure output voltage.
4) Set the following value to parameter No. 3821.

$$\text{Setting value} = \frac{-8191 \times \text{offset voltage (V)}}{12.5}$$

5) After the parameter has been set, command a spindle speed whose analog output becomes 0 and confirm the voltage becomes 0V.

3822	Maximum speed of the third spindle

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

This parameter sets the maximum speed for the third spindle.

When a command specifying a speed exceeding the maximum spindle speed is specified, or the spindle speed exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

WARNING

1 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used. When 0 is set in parameter No. 3772, the spindle speed is not clamped.
2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the speed of the spindle is not clamped.

NOTE

1 This parameter is valid when the multi–spindle control option is selected.
2 When the constant surface speed control option is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is set.

3831	Maximum spindle speed for gear 1 of the third spindle
3832	Maximum spindle speed for gear 2 of the third spindle

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the third spindle.

NOTE

These parameters are used only with muliti–spindle control.

Warning

WARNING

Do not switch between the first and second position coders while a function that uses position coder feedback information is being executed. That is, PMC signal PC2SLC <G028#7> cannot be used while, for instance, a command for feed per rotation or thread cutting is taking place.

Caution

CAUTION

1 If the primary spindle stop signal *SSTP for stopping all selected (SWS1 to SWS3) spindles' rotation is cleared, the speed command is restored. A spindle not selected by SWS1 to SWS3 and rotating at its previous speed, which is stopped using its respective command *SSTP1 to *SSTP3, cannot be restored to that speed when the signal is cleared.

2 Type A multi-spindle control differs from Type B in the relationship between the SWS1 and SIND signals for the first spindle. In Type B, SIND functions only when SWS1 is set to "1". In Type A, SIND functions whether SWS1 is "1" or "0"; each spindle is selected by either of its respective SWS1 or SIND signals being set to "1".

Note

NOTE

1 The spindle orientation signal, spindle speed override signals, and spindle stop signal *SSTP only function for the selected signals.

2 The S 12-bit code signals R01O to R12O outputs the state of a selected spindle. If two or more spindles are selected at the same time, the states of the first, second, third, and fourth spindles are output in this order.

3 The multi-spindle function allows two position coder interfaces to be used. But the number of actual speed indications on the CNC screen does not change. The speed based on the feedback information of the selected position coder is displayed.

4 An SOR command has priority over S commands and SIND-based rotation control from the PMC, and will cause all selected spindles to perform orientation rotation.

9.10 RIGID TAPPING

9.10.1 General

In a tapping cycle (M series: G84/G74, T series: G84/G88), synchronous control is applied to the tapping operation of a tapping axis and the operation of the spindle.

This capability eliminates the need to use a tool such as a float tapper, thus enabling higher–speed, higher–precision tapping.

Whether a tapping cycle is an ordinary tapping cycle or rigid tapping cycle is determined by the miscellaneous function code for rigid tapping M29. (A different M code can be used by setting the parameters accordingly, but M29 is used in the description given here.)

By setting the parameters, G codes for tapping cycles can be changed to G codes for rigid tapping only. In this case, the CNC specifies M29 internally.

To perform rigid tapping, the following must be added to the ordinary connections:

- Connection of a position coder to the spindles (described in 9.10.2)
- Addition of a sequence to the PMC (described in 9.10.6 and 9.10.7)
- Setting of related parameters (described in 9.10.8)

This section provides an example of M series connection.

To avoid duplicate descriptions, assume the following unless noted otherwise:

- G code for a tapping cycle

  M series: G84 (G74) T series: G84 (G88)
- Gear selection method

  M series: M–type or T–type gear selection method
  T series: T–type gear selection method only
- Parameters used according to the number of gear stages (No. 5221 to No. 5224, No. 5231 to No. 5234, No. 5241 to No. 5244, No. 5261 to No. 5264, No. 5271 to No. 5274, No. 5281 to No. 5284, No. 5291 to 5294, No. 5321 to No. 5324, etc.)
- M series: Up to three stages T series: Up to four stages (The stages are also used for the second spindle, and up to two stages are used with the second spindle.)

CAUTION

1 The description given in this section covers up to the second axis.

2 When M–type gear selection is used for the M series, the maximum spindle speed for rigid tapping (specified with parameters No. 5241 to 5243) must also be set for parameter No. 5243 regardless of the number of gear steps. (For a system having a single gear step, set the same value as that of parameter No. 5241 for parameter No. 5243. For a system having two gear steps, set the same value as that of parameter No. 5242 for parameter No. 5243.)

The descriptions given in this section (such as spindle gear switching and M–type/T–type) are based on the explanation given in Section 9.3. Refer to Section 9.3 as necessary.

Specification of M series/T series

The differences in the specifications for rigid tapping for the M series and T series are described below.

- **Rigid tapping of M series**

The tapping cycle G84 and the reverse tapping cycle G74 can be used to specify M series rigid tapping.

A tapping axis can be arbitrarily selected from the basic axes X, Y, Z, as well as axes parallel to the basic axes, by setting the corresponding parameters accordingly (bit 0 (FXY) of parameter No. 5101).

The spindle operations of G84 and G74 are reversed with respect to each other.

If multi–spindle control is added as well as rigid tapping by the first spindle (analog or serial), rigid tapping by selecting the second spindle (serial) is also enabled.

- **Rigid tapping of T series**

The face tapping cycle G84 and the side tapping cycle G88 can be used to specify T series rigid tapping.

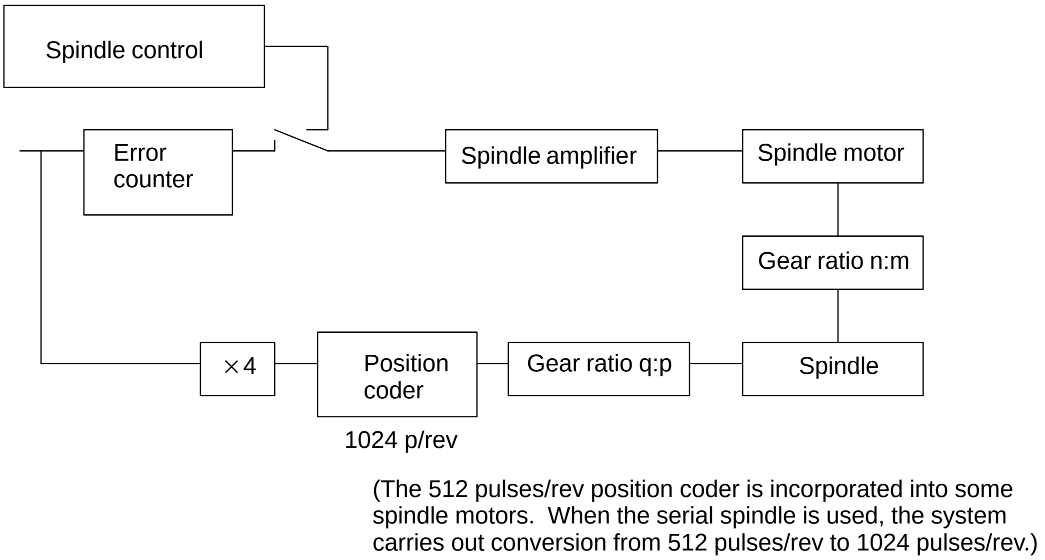
Depending on the rigid tapping command, rigid tapping can be performed along the Z–axis (when G84 is used) or the X–axis (when G88 is used).

A reverse tapping cycle, like that supported by M series, is not available.

If multi–spindle control is added as well as rigid tapping by the first spindle (analog or serial), rigid tapping by selecting the second spindle (serial) is also enabled.

9.10.2 Connection Among Spindle, Spindle Motor, and Position Coder

As shown in the figure below a gear ratio can be inserted between the spindle and spindle motor, and between the spindle and position coder.



(1) Gear between spindle and spindle motor

Up to three gear stages (1st spindle of M series) or four gear stages (1st spindle of T series) can be provided between the spindle and the spindle motor. The gear ratio is arbitrary. Up to two gear stages (2nd spindle) can be provided between the spindle and the spindle motor. The distance of spindle rotation per revolution of spindle motor is different, based on the gear ratio. The speed command to the spindle motor will be adjusted. See (2), below, for additional information regarding a spindle motor incorporating a position coder.

(2) Gear between spindle and position coder

The position coder is used to detect the position of the spindle.
The gear ratio for the spindle and position coder is specified in the parameter sets No. 5221 to No. 5223 and No. 5231 to No. 5233, or parameter set PG1 and PG2 No. 3706 #0, #1, parameter P21, P22 (No. 3707 #0, #1) for 2nd spindlee. Which parameter is set for use is specified by parameter VGR No. 5200 #1.

- **Arbitrary gear ratio (VGR=1)**

This is used if the gear ratio for the spindle motor and position coder (built-in or separate) is not 1:1, 1:2, 1:4, or 1:8, set VGR to 1 and set the gear ratio using parameter No. 5221 to 5224.

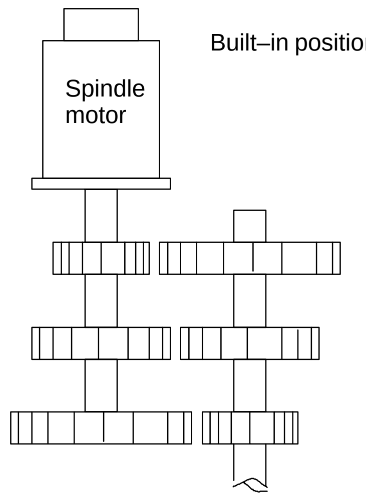
When position coder is mounted on a spindle, the gear ratio for the spindle motor and position coder cannot be changed by shifting the spindle motor and spindle gears. Parameters No. 5221 to 5224 must all specify the same value for the teeth of the individual spindle gears. Parameters No. 5231 to 5234 must all specify the same value for the teeth of individual position coder gears.

The 1024 or 512 pulses/rev position coder is built into the spindle motor. For the 512 pulses/rev version, specify double the number of teeth on each gear for the position coder. (Double the number of teeth need not be specified for the serial spindle.)

1st spindle of M series allows up to three stages, regardless of which gear selection method has been selected. (Parameter Nos. 5224 and 5234 cannot be used.)

1st spindle of T series supports up to four stages. (Set parameter Nos. 5221 to 5224 and 5231 to 5234.) When the multi–spindle function is used to perform rigid tapping with the second spindle, up to two stages are supported. (Set parameter Nos. 5221, 5222, 5231, and 5232.)

Example)



Parameter No.	Set value		Meaning
	512p/rev Position coder	1024p/rev Position coder	
5221	70		Number of teeth of the 1st gear for the spindle side
5222	50		Number of teeth of the 2nd gear for the spindle side
5223	30		Number of teeth of the 3rd gear for the spindle side
5231	80 Note)	30	Number of teeth of the 1st gear for the position coder side
5232	100 Note)	50	Number of teeth of the 2nd gear for the position coder side
5233	140 Note)	70	Number of teeth of the 3rd gear for the position coder side

NOTE

Double value setting is not required for serial spindle.

- **Gear ratio is 1:1, 1:2, 1:4, 1:8 (VGR=0)**

If the gear ratio is either 1:1, 1:2, 1:4, and 1:8, it is set using parameters PG1 and PG2 (No. 3706 #0, #1). This applies if the position coder is mounted in a spindle or built into a spindle motor when only one stage gear is provided. .
For 2nd spindle, set it to parameter P21, P22 (No.3707#0, #1).

Parameter		Gear ratio		Detection unit
PG2	PG1	Spindle	Position coder	
0	0	1	1	360/4096=0.08789 deg
0	1	1	2	360/4096×2=0.17578 deg
1	0	1	4	360/4096×4=0.35156 deg
1	1	1	8	360/4096×8=0.70313 deg

The spindle motor building in the 512 pulses/rev position coder uses the values set forth in the following table. A serial spindle does not require double-value setting; use the same values as for the spindle motor building in the 1024 pulses/rev position coder.

	Gear ratio		Parameter		Gear ratio of spindle to position coder	Detection unit (deg)
	Spindle motor	Spindle	PG2	PG1		
Built–in position coder 512p/rev	1	1	0	1	1:2	0.17578
	2	1	1	0	1:4	0.35156
	4	1	1	0	1:8	0.70313

(3) Rigid tapping and machines with multiple gears

If the M type gear selection method is selected, the CNC determines whether gears need changing using the gear change specification mentioned in section 9.3. If the gears need to be changed, the CNC generates the S function code read signal SF (F007#2) and gear selection signals GR1O, GR2O, and GR3O (F034#0-#2) to notify the PMC. Change gears using the PMC, based on these signals.

If the T type gear selection method is selected, the CNC does not process gear changes. When the CNC has the S function code, it outputs signal SF and S function code signals S00 to S31 (F022#0-F025#7) to the PMC. (However, parameter No. 3705 and its related parameters need to be set for S code and SF signal output). Using the PMC, determine whether gears need changing, and make the change if needed. Input gear selection signals GR1 and GR2 <G028#1,#2> or GR21 <G029#0> for 2nd spindle for the selected gear, and notify the CNC of them.

To perform rigid tapping with the serial spindle, enter the clutch/gear selection signals CTH1 and CTH2 (G070#3,#2 for the first spindle and G074#3, #2 for the second spindle) from the PMC. Notify the serial spindle control unit of these signals via the CNC, irrespective of the gear selection method.

Changing gears during rigid tapping requires a different process from that for gear changes during normal machining. As described above, changing gears conforms to the gear change specifications mentioned in section 9.3 when the M type gear selection method has been selected. With the T type gear selection method, changing gears conforms to the logic programmed in the PMC.

Regardless of the option's selection, if the range in which the spindle speed specified by the S function code does not correspond to the currently selected gear, the gears are changed.

The following tables list the spindle speed ranges for each gear during normal machining (assuming no machine restrictions) and rigid tapping:

Gear	Spindle speed range (normal machining)	
	Lower limit	**Upper limit**
Low–speed gear	1 revolution	Maximum low–speed gear speed = $\frac{\text{Maximum spindle motor speed}}{\text{Low–speed gear ratio}} \times L\%$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed = $\frac{\text{Maximum spindle motor speed}}{\text{Medium speed gear ratio}} \times L\%$
High–speed gear	Maximum medium–speed gear speed +1 revolution	Maximum high–speed gear speed = $\frac{\text{Maximum spindle motor speed}}{\text{High–speed gear ratio}} \times L\%$

NOTE

This table shows an example of three gears. L% indicates a spindle motor protection constant (up to 100). L can be specified for each gear using method B for changing in M type gear selection method (bit 2 (SGB) of parameter No. 3705 =1).

Gear	Spindle speed range (during rigid tapping)	
	Lower limit	**Upper limit**
Low–speed gear	1 revolution	Maximum low–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Medium–speed gear ratio}}$
High–speed gear	Maximum medium–speed gear speed +1 revolution	Maximum high–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{High–speed gear ratio}}$

NOTE
This table show an example of three gears. For the basic spindle motor speed, refer to the spindle motor description manual. "+ α" means that the spindle motor speed may slightly exceed the basic spindle motor speed.

If the M type gear selection method is used, use gear change method B (bit 3 (SGT) of parameter No. 3705 = 1) in the tapping cycle to specify the following:

The table above shows the maximum low-speed gear speed during rigid tapping for low-/medium-speed gear change position D (parameter No. 3761).

The table above shows the maximum medium-speed gear speed during rigid tapping for medium-/high-speed gear change position E (parameter No. 3762).

If the T type gear selection method is used, add the rigid tapping logic to the logic programmed in the PMC.
See Section 9.3, "Spindle Control" for details of the spindle gear change specifications.

The loop gain can be specified for each gear. Specify "0" for parameter No. 5280 and specify loop gains for each gear for parameter Nos. 5281 to 5284. Unless "0" is specified for parameter No. 5280, the loop gains for each gear are disabled, and the loop gain common to all gears, the value of parameter No. 5280, is enabled.

Specify the time constant and the maximum spindle speed for each gear. Use parameters Nos. 5261 to 5264 to specify the time constant.
Use parameters Nos. 5241 to 5244 to specify the maximum spindle speed.

For M type gear selection method, set the maximum spindle speed to parameter No. 5243, irrespective of the number of gear stages used.

Setting bit 2 (TDR) of parameter No. 5201 to "1" enables setting of the time constants used during extraction for each gear set. Specify the extraction time constant for each gear in parameter Nos. 5271 to 5274.

If bit 1 (VGR) of parameter No. 5200 is set to "1", the gear ratio for the spindle and position coder can be set to anywhere between 1:32767 and 32767:1 in one-increment units for three gear sets with 1st spindle of M series, four gear sets with 1st spindle of T series, or two gear sets with 2nd spindle. However 1:8 to 8:1 is the recommended value.

9.10.3 Rigid Tapping Specification

- **Feed rate**

In rigid tapping mode, the tapping axis is fed at a rate specified by F. The spindle speed is specified by S × 360(deg/min). Use of override is invalid for both of them. An override of 200% can be applied to withdrawal operations by setting bit 4 (DOV) of parameter No. 5200 to "1", and setting an appropriate override value in parameter RGOVR of No. 5211. Using the OVU parameter (bit 3 of parameter No. 5201) enables an override value of up to 2000% to be applied. The time constant for withdrawal operations can be modified by bit 2 (TDR) of parameter No. 5201; when it is set to "1", the values in parameter Nos. 5271 to 5274 are used as the time constant for withdrawal.

- **Acceleration and deceleration**

Linear acceleration/deceleration is valid for rigid tapping.

- **Override**

Override is invalid during rigid tapping. Fixed override can be applied to withdrawal operations by setting bit 4 (DOV) of parameters No. 5200 or RGOVR of No. 5211.

- **Dry run**

Dry run is valid for G84 (G74). When the dry run is applied to the tapping axis speed of G84 (G74), tapping is performed. The spindle speed will match the dry run speed.

- **Machine lock**

Machine lock is valid for G84 (G74).
When G84 (G74) is executed with the machine locked, however the tapping axis and the spindle do not move.

- **Reset**

When the reset operation is performed during rigid tapping, the mode is reset. The spindle motor goes to the ordinary mode, but G84 (G74) mode is not reset.

- **Feed hold and single block**

The feed hold and single block functions are nullified for G84 (G74).
The feed hold and single block functions in rigid tapping mode can be effective by setting bit 6 (FHD) of parameter No. 5200 to "1".
As with the machine lock signal, the feed hold and single block functions are also effective for the spindle indirectly, through tapping axis operations.

- **Operation mode**

G84 (G74) can be executed only in the MEM and MDI modes.

- **Manual feed**

Rigid tapping cannot be performed in the manual feed mode.

- **Backlash compensation**

In rigid tapping mode, the backlash is compensated for the lost motion at forward and reverse spindle rotations. Set it using parameter No. 5321 to No 5324. The normal backlash compensation is inserted on the tapping axis.

9.10.4 Display Data on the Diagnosis Screen

For rigid tapping adjustment, the diagnosis screen displays information related to rigid tapping.

For part of the display data, the user can choose between two sets of data items relating to the synchronization of the spindle and tapping axis by setting bit 0 (DGN) of parameter No. 5204.

- **Common display data**

The following information items are always displayed, regardless of the setting of bit 0 (DGN) of parameter No. 5204:

· Spindle position deviation → Diagnosis No. 450

· Number of command pulses distributed to the spindle (momentary value) → Diagnosis No. 451

· Cumulative number of command pulses distributed to the spindle → Diagnosis No. 454

- **Display of rigid tapping synchronization error (When DGN = 0)**

When bit 0 (DGN) of parameter No. 5204 is set to 0, the following items of information are displayed.

(Diagnosis Nos. 452 and 453 are not displayed.)

· Spindle–converted move command difference → Diagnosis No. 455

· Spindle–converted position deviation difference → Diagnosis No. 456

· Synchronization error range → Diagnosis No. 457

Spindle–converted move command difference

$$= \Sigma \frac{\text{spindle move command}}{\text{gear ratio}} - \Sigma \frac{(\text{tapping axis move command}) \times 4096}{\text{thread lead}}$$

Spindle–converted position deviation difference

$$= \frac{\text{spindle position deviation}}{\text{gear ratio}} - \frac{(\text{tapping axis position deviation}) \times 4096}{\text{thread lead}}$$

Synchronization error range

= (maximum spindle–converted move position deviation difference on the positive side)

– (maximum spindle–converted position deviation difference on the negative side)

If a maximum allowable synchronization error range is set in parameter No. 5214, the position deviation alarm during spindle movement (alarm No. 741) is issued to indicate that the set synchronization error range has been exceeded. (If 0 is set in parameter No. 5214, no check is performed to detect whether the synchronization error range has been exceeded.)

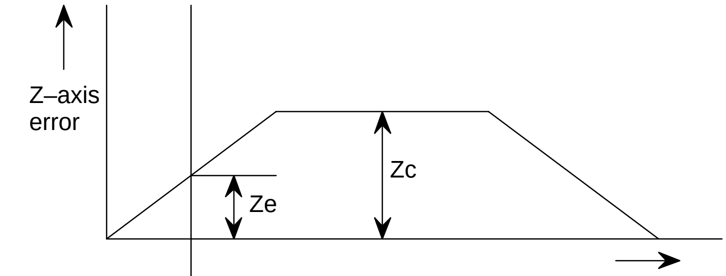
- **Rigid tapping error display (error difference display) (When DGN = 1)**

When bit 0 (DGN) of parameter No. 5204 is set to 1, the following items of information are displayed. (Diagnosis Nos. 455, 456, and 457 are not displayed.)

· Momentary error difference between the spindle and tapping axis → Diagnosis No. 452

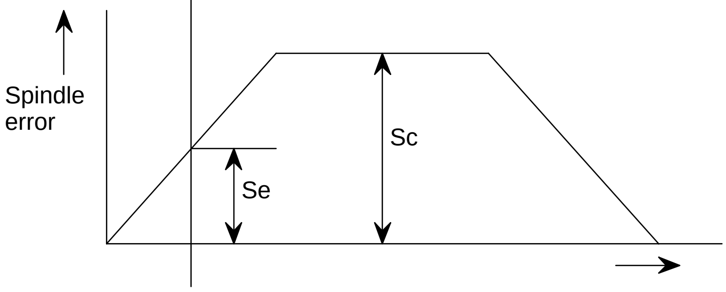
· Maximum error difference between the spindle and tapping axis → Diagnosis No. 453

Diagnosis No. 0452 is cleared to “0” when rigid tapping mode is set or canceled, and diagnosis No. 0453 is cleared to “0” in the positioning of the rigid tapping cycle.
The following figure shows the tapping axis as the Z axis.



$$Zc = \frac{\text{Speed}}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \ \text{(Theoretical value)}$$

Ze = Z–axis error counts (measured value)



$$Sc = \frac{\text{Speed} \times 360}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \ \text{(Theoretical value)}$$

Se = Spindle error counts (measured value)

Speed : mm/min or inch/min
Gain : 0.01 s^{-1}
Detection unit :mm, inch, or deg
Speed : min^{-1}

$$\Delta Z = \frac{Ze}{Zc} \times 100 \ [\%]$$

$$\Delta S = \frac{Se}{Sc} \times 100 \ [\%]$$

The error difference between the spindle and Z axis can be obtained by ΔS–ΔZ. This value is sampled at intervals of 64 ms.

Diagnosis screen

- **Spindle position deviation**

0450	SPINDLE MOTION ERROR

Spindle position deviation during rigid tapping

[Unit] Pulse

- **Number of pulses distributed to the spindle**

0451	SPINDLE MOTION PULSE

Number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Error difference between the spindle and tapping axis (momentary value)**

0452	RIGID ERROR

Momentary error difference between the spindle and tapping axis during rigid tapping (signed)

[Unit] %

NOTE
This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Error difference between the spindle and tapping axis (maximum)**

0453	RIGID ERROR(MAX)

Maximum error difference between the spindle and tapping axis during rigid tapping (absolute value)

[Unit] %

NOTE
This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Cumulative number of pulses distributed to the spindle during rigid tapping**

0454	SPINDLE PULSE(SUM)

Cumulative number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Spindle–converted move command difference during rigid tapping (momentary value)**

0455	SYNC. PULSE(SUM)

Momentary spindle–converted move during command difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Spindle–converted position deviation difference during rigid tapping (momentary value)**

0456	SYNC. ERROR

Momentary spindle–converted position deviation difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Synchronization error range during rigid tapping (momentary value)**

0457	SYNC. WIDTH

Synchronization error range during rigid tapping (maximum value)

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

9.10.5
Command Format

Command format for the T series

The rigid tapping command format for the T series is described below. For an explanation of the command format used with the M series, refer to Section II.13.2.2 of the “Operator’s Manual for Machining Center (B–63534EN) ”.

G□□ X(Z)__ C__ Z(X)__ R__ P__ F__ K__ (M□□) ;

- (M□□): M code for C–axis clamping
- K: Number of repeats
- F: Cutting feedrate
- P: Dwell time at the bottom face of a tapped hole and upon R point return
- R: Specifies the position of an R point
- Z(X): Specifies the position of the bottom face of a tapped hole
- X(Z), C: Specifies a tapping position.
- G□□: G84 — Front tapping cycle; G88 — Side tapping cycle

The rigid tapping mode can be specified by using any of three methods:

- Specification of M29S**** before specifying a tapping cycle
- Specification of M29S****in the same block
- Enabling rigid tapping to be performed without specifying M29S****

When using the third method, specify S**** either before or in a block containing G84 (G88).

Thus, the spindle stops, after which the tapping cycle specified next is placed in rigid tapping mode.

Rigid tapping mode will be canceled by G80;. Note, however, that a G code for another canned cycle, or a group 01 G code will also cancel rigid tapping mode.

When rigid tapping is terminated by a command issued to cancel rigid tapping mode, the spindle stops. (Output to the spindle is equivalent to the specification of S0.)

A reset (by means of the RESET button or an external reset) can also cancel rigid tapping mode. Note, however, that canned cycle mode is not canceled by a reset.

- Specifying M29 before a block containing G84 (G88)

```
M29 S****;
G□□X (Z) __C__Z (X) __R__P__F__K__ (M□□) ;   ┐
X (Z) __C__;                                  │ Rigid
X (Z) __C__;                                  │ tapping
 .                                            │ mode
 .                                            ┘
G80;
```

- Specifying M29 and G84 (G88) in the same block (Note, however, that M29 and M□□ for C–axis clamping cannot be specified in the same block.)

```
G□□X (Z) __Z (X) __R__P__F__K__M29****;   ┐
X (Z) __C__;                               │ Rigid
X (Z) __C__;                               │ tapping
 .                                         │ mode
 .                                         ┘
G80;
```

- Converting G84 (G88) to a G code for rigid tapping (by setting bit 0 (G84) of parameter No. 5200 to 1)

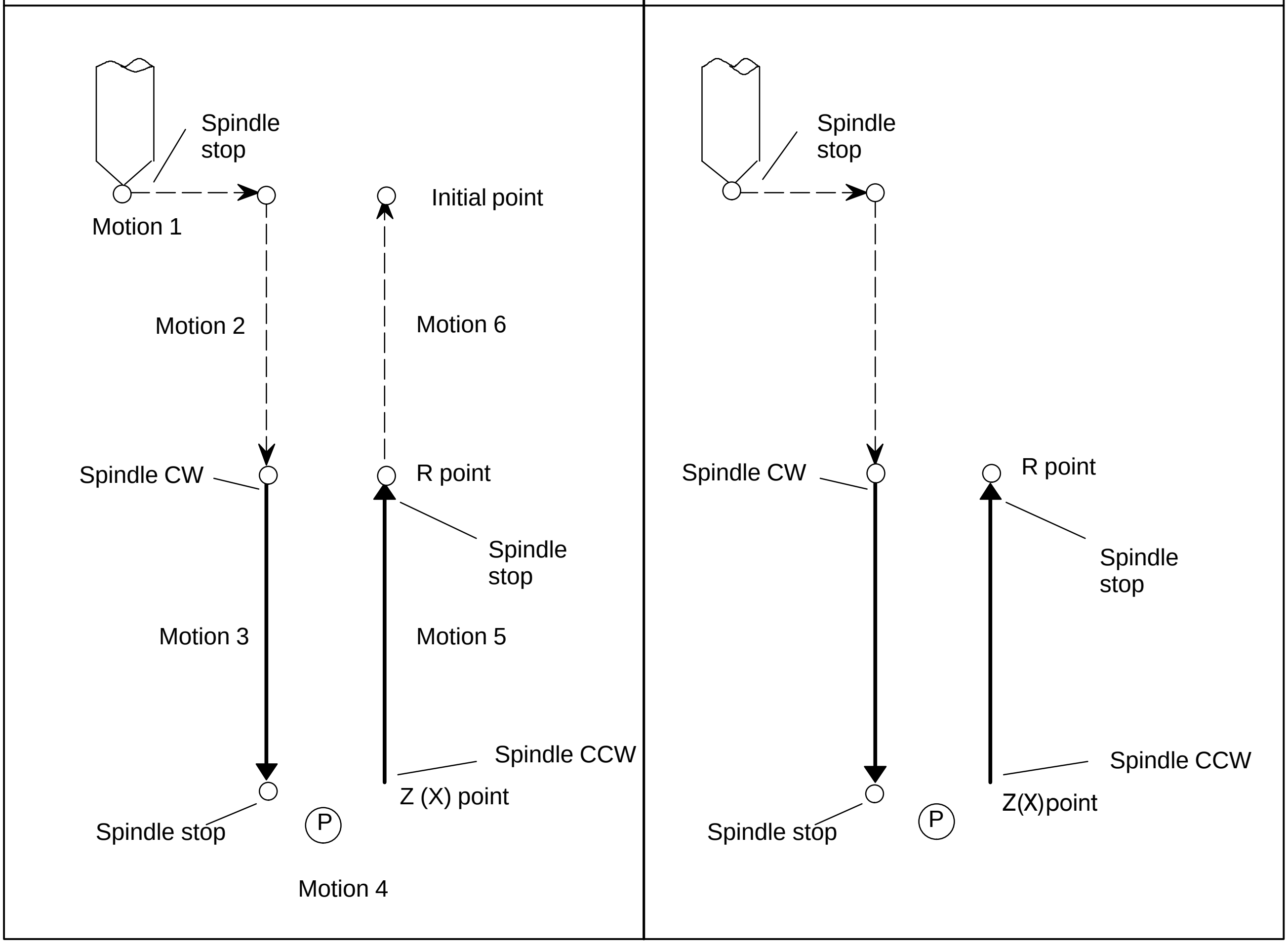
```
G□□X (Z) __C__Z (X) __R_P_F_K_S**** (M□□);   ┐
X (Z) __C__;                                  │ Rigid
X (Z) __C__;                                  │ tapping
 .                                            │ mode
 .                                            ┘
G80;
```

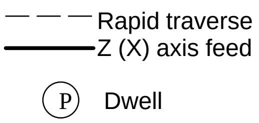
Notes on the T series

> **NOTE**
> 1 In feed per minute mode, F_/S**** determines a thread lead. In feed per rotation mode, F_ specifies a thread lead.
> 2 S**** must specify a value that does not exceed the value set in the maximum spindle speed parameter (No. 5241 to 5244) for the gear to be used. Otherwise, P/S alarm No. 200 is issued in a block containing G84 (G88).
> 3 F_ must specify a value that does not exceed the maximum cutting feedrate. When 0 is specified, P/S alarm No. 201 is issued.
> 4 Between M29 and G84 (G88), S and a command for movement along an axis must not be specified. Further, M29 must not be specified in a tapping cycle. Otherwise, P/S alarm Nos. 203 and 204 are issued, respectively.

G84·G85 (Tapping cycle)

G84 (G88) (G98 mode)	G84 (G88) (G99 mode)





> **CAUTION**
> During cutting feed along the Z–axis (X–axis), the feedrate override is forced to 100%. The spindle speed override is also forced to 100%. For a retract motion (motion 5), a fixed override of up to 2000% can be applied by specifying bit 4 (DOV) of parameter No. 5200, bit 3 (OVU) of parameter No. 5201, and parameter No. 5211 (RGOVR).

> **NOTE**
> G code system A does not include G98 (return to initial level) and G99 (return to R point level). Return to the initial level is always used.

Rigid tapping in feed per rotation mode

Rigid tapping is classified into two types: rigid tapping in feed per rotation mode (G99) and rigid tapping in feed per minute mode (G98).

Example)

The example below specifies rigid tapping in feed per rotation mode for cutting a thread with a lead of 1 mm at a spindle speed of 1,000 min^{-1}.

```
O0001 ;
G99 ;
  .
  .
  .
M29  S1000 ;
G84  Z–100.   R–20.   F1. ;
  .
  .
  .
G80 ;
```

The example below specifies rigid tapping in feed per minute mode for cutting the same thread at the same spindle speed as above. (In feed per minute mode, F/S determines the thread lead.)

```
O0002 ;
G98 ;
  .
  .
  .
M29  S1000 ;
G84  Z–100.   R–20.   F1000 ;
  .
  .
  .
G80 ;
```

Units of F

	Metric input	Inch input	Remarks
G98	1 mm/min	0.01 inch/min	A fractional value can be specified.
G99	0.0001 mm/ rev	0.000001 inch/rev	A fractional value can be specified.

NOTE

1 G98 and G99 are modal G codes. Upon power–up, G99 (feed per rotation mode) is set.

2 Even in feed per rotation mode, a pulse distribution command is converted to a feed per minute command. Thus, feed per rotation mode does not strictly implement feed per rotation. Accordingly, even if the spindle stops for some reason, the tapping axis (Z–axis or X–axis) does not stop.

9.10.6 Signal

9.10.6.1 Signals for the rigid tapping function

Rigid tapping signal RGTAP<G061#0>

[Classification] Input signal

[Function] When M29 (miscellaneous function for preparation for rigid tapping) is specified, the PMC enters rigid tapping mode, then turns on this signal to notify the CNC.

1 : The PMC enters in rigid tapping mode.
0 : The PMC does not enter rigid tapping mode.
For an explanation of placing the PMC in rigid tapping mode, see the description of the interface with the PMC, given later.

This signal posts whether the PMC has entered rigid tapping mode. If this signal is not set to 1, even when M29 is specified, a P/S alarm is issued in a G84 (G74) block.

Spindle rotation direction signals RGSPM, RGSPP <F065#1, #0> (M series)

[Classification] Output signal

[Function] During rigid tapping, these signals notify the PMC of whether the spindle is rotating in the forward or reverse direction.

During rigid tapping, the spindle is:

RGSPP 1 : Rotating in the forward direction (CW).
0 : Not rotating in the forward direction.

RGSPM 1 : Rotating in the reverse direction (CCW).
0 : Not rotating in the reverse direction.

[Output condition] These signals are output when the spindle is rotating in rigid tapping mode. This means that, even in rigid tapping mode, these signals are not output, for example, when the spindle is being positioned to a hole position, or a dwell operation is in progress at the bottom of a hole or at an R point.

These signals are not output in the feed hold state or single block stop state. When the spindle is placed in the interlock stop state, machine lock state, or Z–axis ignore state, however, the spindle is not regarded as having stopped; these signals are output.

These signals are valid only in rigid tapping mode. In normal spindle control, these signals are not output; both RGSPP and RGSPM are set to “0”.

Rigid tapping in–progress signal RTAP<F076#3>

[Classification] Output signal

[Function] This signal notifies the PMC that rigid tapping mode is active.

RTAP 1 : Rigid tapping mode is currently active.

0 : Rigid tapping mode is not currently active.

By latching M29, the PMC knows that rigid tapping mode has been specified, and thus performs the required processing on the PMC side. This signal can substitute for the latching of M29. Even in this case, however, FIN for M29 cannot be omitted.

9.10.6.2 Signals related to S code output

Spindle enable signal ENB<F001#4>

Second spindle enable signal ENB2<F038#2>

[Classification] Output signal

[Function] These signals post whether the spindle output is 0. In rigid tapping mode, these signals are used to cancel rigid tapping in a PMC sequence associated with rigid tapping.

For details, see the explanation of the interface with the PMC, given later.

Spindle–speed function code signals (binary output) S00 to S31 <F022 to F025>

Spindle–speed function strobe signal SF<F007#2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in binary format, to the PMC.

[Output condition] When an S code is specified, the specified value is output, in binary format, with these signals. When the new spindle speed data is fully set, the SF signal is set to “1”.

Before rigid tapping can be performed, however, parameter setting is required to output these signals, as described below.

M series: SF output depends on the gear selection method, as described below.

[1] M–type gear selection method

SF output depends on bit 6 (SFA) of parameter No. 3705.

[2] T–type gear selection method

SF output depends on the setting of bit 5 (NSF) of parameter No. 3705.

T series: The following parameter needs to be set to output S codes and SF: Bit 4 (EVS) of parameter No. 3705 = 1

In rigid tapping, when SF is to be used by the PMC to read an S code output signal for gear switching or output switching, set the above parameters as required.

> **NOTE**
> 1 The timing charts, given later, give examples of gear switching by setting the parameters as follows:
> M series: SFA = 0, NSF = 0
> T series: EVS = 1
> 2 When the constant surface speed control function is being used, an S code (specifying a surface speed) used for constant surface control (G96) is output. Such an S code can be distinguished from an S code used for specifying a rotation speed. One method is to use, for example, the constant surface speed control in–progress signal <F002#2> for the processing performed on the PMC side. Another method is to mask the S code and SF signal, output by setting bit 0 (ESF) of parameter No. 3705.

9.10.6.3 Signals related to gear switching

Gear selection signals (output) GR3O, GR2O, GR1O <F034#2, #1, #0> (M series)

[Classification] Output signal

[Operation] When M–type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.

The signals post, to the PMC, information about a spindle gear to be used, according to the value of S**** specified at the execution of G84 (G74).

When gear switching becomes necessary, the states of the signals change together with the SF signal.

The PMC should perform gear switching according to the information posted by the gear selection signals.

Reference information: The table below indicates the relationship between the output signals and gear selection.

	GR3O	GR2O	GR1O
1st (low) speed gear	×	×	○
2nd (medium) speed gear	×	○	×
3rd (high) speed gear	○	×	×

Gear selection signals (input) GR2, GR1<G028#2, #1>

[Classification] Input signal

[Operation] When T–type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.

The signals post, to the CNC, information about a spindle gear to be used.

Reference information: The table below shows the relationship between the output signals and spindle gear selection.

	GR2	GR1
1st (low) speed gear	×	×
2nd (medium) speed gear	×	○
3rd (high) speed gear	○	×
4th (high) speed gear	○	○

← In M series rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

Gear selection signal (input) GR21<G029#0>

[Classification] Input signal

[Operation] When rigid tapping with the second spindle is being performed, the signal is used in the PMC sequence program.

The signal notifies the CNC of spindle gear information of the selected spindle.

The input signal is related to gear selection as described below.

GR21 1 : The second stage is currently selected as the second spindle gear.
0 : The first stage is currently selected as the second spindle gear.

	GR21
1st speed gear	×
2nd speed gear	○

When a serial spindle is used, the serial spindle clutch/gear selection signals (G070#3, #2 for the first spindle and G074#3, #2 for the second spindle) must be set in addition to the setting of the gear selection signal described above.

9.10.6.4 Signals related to second spindle rigid tapping

Gear selection signal (input) GR21<G029#0>

See the description of the signals related to gear switching, given above.

Signals related to multi–spindle control

Spindle selection signals SWS1, SWS2 <G027#0, #1>

Rigid tapping spindle selection signals RGTSP1, RGTSP2 <G061#6, #5> (T series)

[Classification] Input signal

[Operation] SWS1 and SWS2 are used to transfer spindle commands when the multi–spindle control is used. In rigid tapping, the signals can be shared to select a spindle to be used for rigid tapping. (The signals can be used for this purpose when bit 7 (SRS) of parameter No. 5200 is set to 0.)

RGTSP1 and RGTSP2 are used to select a spindle used for rigid tapping, independently of the SWS1 and SWS2 signals, when the multi–spindle control is being used. (The RGTSP2 and RGTSP1 signals can be used when bit 7 (SRS) of parameter No. 5200 is set to 1. These signals are supported only by the T series.)

See the tables below for details of the settings of these signals.

(T/M series) When bit 7 (SRS) of parameter No. 5200 is set to 0, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	SWS1	SWS2
First spindle	“1”	“1” or “0”
Second spindle	“0”	“1”
P/S alarm No. 205 is issued.	“0”	“0”

(T series) When bit 7 (SRS) of parameter No. 5200 is set to 1, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	RGTSP1	RGTSP2
First spindle	“1”	“1” or “0”
Second spindle	“1”	“1”
P/S alarm No. 205 is issued.	“0”	“0”

WARNING
These signals must be applied before the command for rigid tapping (M29 S....; G84 X...) is specified. The states of these signals must not be changed before rigid tapping has been completed.

Spindle–by–spindle stop signals *SSTP1, *SSTP2 <G027#3, #4>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi–spindle control is used. In a PMC sequence for rigid tapping, the ENB and ENB2 signals are used. Accordingly, the logic of the signals used for a spindle selected to perform rigid tapping must match the logic of the spindle stop signal *SSTP.

***SSTP1** 1 : The output to the first spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to first spindle.

***SSTP2** 1 : The output to the second spindle is not forced to 0 min^{-1}.
0 : 0 min^{-1} is commanded to second spindle.

Second position coder selection signal PC2SLC<G028#7>

[Classification] Input signal

[Operation] This signal is used to select the position coder when the multi–spindle control is being used. Note, however, that it cannot be used with a spindle selected to perform rigid tapping.

For rigid tapping, this signal is not used. Instead, a position loop is constructed by combining the first spindle with the first position coder, by combining the second spindle with the second position coder.

However, the display of the actual speed is switched by this signal, even during rigid tapping (See Section 9.10 for details of this signals).

9.10.6.5
Signal addresses

	#7	#6	#5	#4	#3	#2	#1	#0
G027				*SSTP2	*SSTP1		SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029								GR21
G061			RGTSP2	RGTSP1				RGTAP

	#7	#6	#5	#4	#3	#2	#1	#0
F001				ENB				
F007						SF		
F034						GR3O	GR2O	GR1O
F038						ENB2		
F065							RGSPM	RGSPP
F076					RTAP			

9.10.6.6
Notes on interface with the PMC

The following describes some notes in designing the interface with the PMC.

Rigid tapping mode management and ENB (or ENB2)

The PMC must manage rigid tapping mode as follows: rigid tapping mode is set using M29, and is canceled upon the issue of a reset or at the falling edge of the spindle enable signal ENB in rigid tapping mode. ENB is used during rigid tapping in this way, so the spindle stop signal *SSTP must not be set to “0”.

However, *SSTP and SOR may be used for gear switching. To do so, ensure that the PMC does not cancel rigid tapping mode on a falling edge of ENB while *SSTP is “0”. Rigid tapping mode may be set on a rising edge of the RTAP signal instead of by using M29, and canceled on a falling edge of the RTAP signal instead of the ENB signal.

In rigid tapping using the second spindle, the ENB2 signal must be used for rigid tapping mode management.

Controlling spindle output by the PMC

When the SIND signal is set to “1”, spindle output is controlled by the signals (SSIN, SGN, R1I to R12I) output from the PMC.

At this time, the effect of ENB is as described above. In addition, when rigid tapping mode is canceled in a block containing G80;, the momentary rotation of the spindle, caused by a delay in the PMC processing, can result. Accordingly, the PMC's control over spindle output must be disabled in rigid tapping mode by setting SIND to “0”.

For the same reason, the PMC's control over second spindle output must be disabled in rigid tapping mode by setting SIND2 to “0”.

T–type gear selection method

When T–type gear selection is used, the PMC must determine whether gear switching is to be performed, and subsequently perform gear switching as required. For this purpose, each time a spindle–speed function code is specified, the spindle–speed function code read signal (SF) and spindle–speed function code signals (S00 to S31) must be output to the PMC. The required parameter settings are described below.

· M series: Set bit 5 (NSF) of parameter No. 3705 to 0 to output SF.

· T series: Set bit 4 (EVS) of parameter No. 3705 to 1 to output SF.

Gear switching timing

In general, a block containing M29 (miscellaneous function for preparation for rigid tapping) specifies S****, S**** being output when a block containing G84 (G74) is executed. This means that gear switching is performed in the block specifying G84 (G74).

When rigid tapping mode is specified

M29 (miscellaneous function for preparation for rigid tapping) and S**** specify rigid tapping mode. When M29 is accepted by the PMC, the following processing must be performed:

- Stop the spindle when it is rotating.
- Check that the spindle has stopped completely, then set the rigid tapping signal RGTAP <G061#0> to on.
- Activate the spindle motor. Activate the motor so that a positive speed command rotates the spindle in the forward direction (CCW when viewed from the – side of the tapping axis).
- Return FIN at least 250 ms after activation.

NOTE

The condition “at least 250 ms after activation” results from there being no way of checking the completion of spindle motor activation. Therefore, this wait period serves as an alternative. The time required for activation to be completed varies with the spindle motor and amplifier. Therefore, this value of 250 ms is given as a guideline only.

In an M29 block, S**** is not executed, merely being read in. S**** is executed in a G84 block. Spindle output is equivalent to the specification of S0.

The timing chart is shown in the chart indicating the execution of G84 (G74).

Execution of G84 (G74)

When M29S****; is specified, S**** is read in, spindle output being equivalent to the specification of S0;. S**** is output when G84 (G74) is executed. Thus, the processing described below is performed.

• When M–type gear selection is used

When using a machine that features multiple gear stages for use with the spindle motor and spindle, and the newly programmed S**** is outside the previously selected gear range, the spindle–speed function strobe signal SF <F007#2> and gear selection signals (output) GR3O, GR2O, GR1O <F034#2, #1, #0> are output to the PMC.

At this time, perform gear switching at the PMC.

- **When T–type gear selection is used**

The spindle–speed function strobe signal SF <F007#2> and spindle–speed function code signals S00 to S31 <F022 to F025> are output to the PMC. (However, parameter setting is required to enable output of the S codes and SF signal. See the description of each bit of parameter No. 3705.)

At this time, the PMC must determine whether gear switching is to be performed, and perform gear switching as required. The selected gear must be reflected in the gear selection signals (input) GR2 and GR1 <G028#2, #1> for notification to the CNC.

From GR2 and GR1, the CNC determines which gear is selected.

However, note the difference between the M series and T series, as described below.

M series: Up to three gear stages are supported. If the fourth gear stage is selected, it is assumed that the third gear stage is selected.

T series: Up to four gear stages are supported for the first spindle, and up to two gear stages for the second spindle. (The gear selection for the second spindle/third spindle is notified to the CNC by the signal GR21 <G029#0>.)

An S code is output in the first block (positioning to tapping position) of G84 (G74) execution. However, the spindle motor position loop is closed in the next block (R point positioning). Accordingly, spindle speed offset must be adjusted accurately until the position loop has been closed in the second block of G84 (G74) execution after the PMC activates the spindle motor with M29. Otherwise, the spindle motor may rotate slightly. (This applies only to an analog spindle. No offset adjustment is required for a serial spindle.)

Rigid tapping mode may be specified by specifying M29 before G84, specifying M29 and G84 in the same block, or by specifying G84 as a G code for rigid tapping. In each case, PMC processing is the same. (The M29 code is always output.)

9.10.7 Timing Charts for Rigid Tapping Specification

The timing chart for rigid tapping specification depends on the method used to specify rigid tapping mode, the gear selection method (M–type or T–type), and whether to perform gear switching.

From the table, find the appropriate timing chart (Fig. 9.10.7.1 (a) to Fig. 9.10.7.3 (d)) and apply the information it contains as necessary.

Gear selection method M–type T–type	**Gear switching**	**Specification method**		
		M29 is specified before G84 (G74).	**M29 and G84 (G74) are specified in the same block.**	**By parameter setting, G84 (G74) is specified as a G code for rigid tapping.**
M–type	Not performed	Fig. 9.10.7.1 (a)	Fig. 9.10.7.2 (a)	Fig. 9.10.7.3 (a)
	Performed	Fig. 9.10.7.1 (b)	Fig. 9.10.7.2 (b)	Fig. 9.10.7.3 (b)
T–type	Not performed	Fig. 9.10.7.1 (c)	Fig. 9.10.7.2 (c)	Fig. 9.10.7.3 (c)
	Performed	Fig. 9.10.7.1 (d)	Fig. 9.10.7.2 (d)	Fig. 9.10.7.3 (d)

NOTE

For more information about the M/T type gear selection method, see Section 9.3 SPINDLE CONTROL. Note the following:

T series:	T–type	only
M series:	M–type	when constant surface speed control is not being used and bit 4 (GTT) of parameter No. 3706 is set to 0
	T–type	when constant surface speed control is being used, or bit 4 (GTT) of parameter No. 3706 is set to 1

9.10.7.1
When M29 is specified before G84 (G74) M type gear selection method

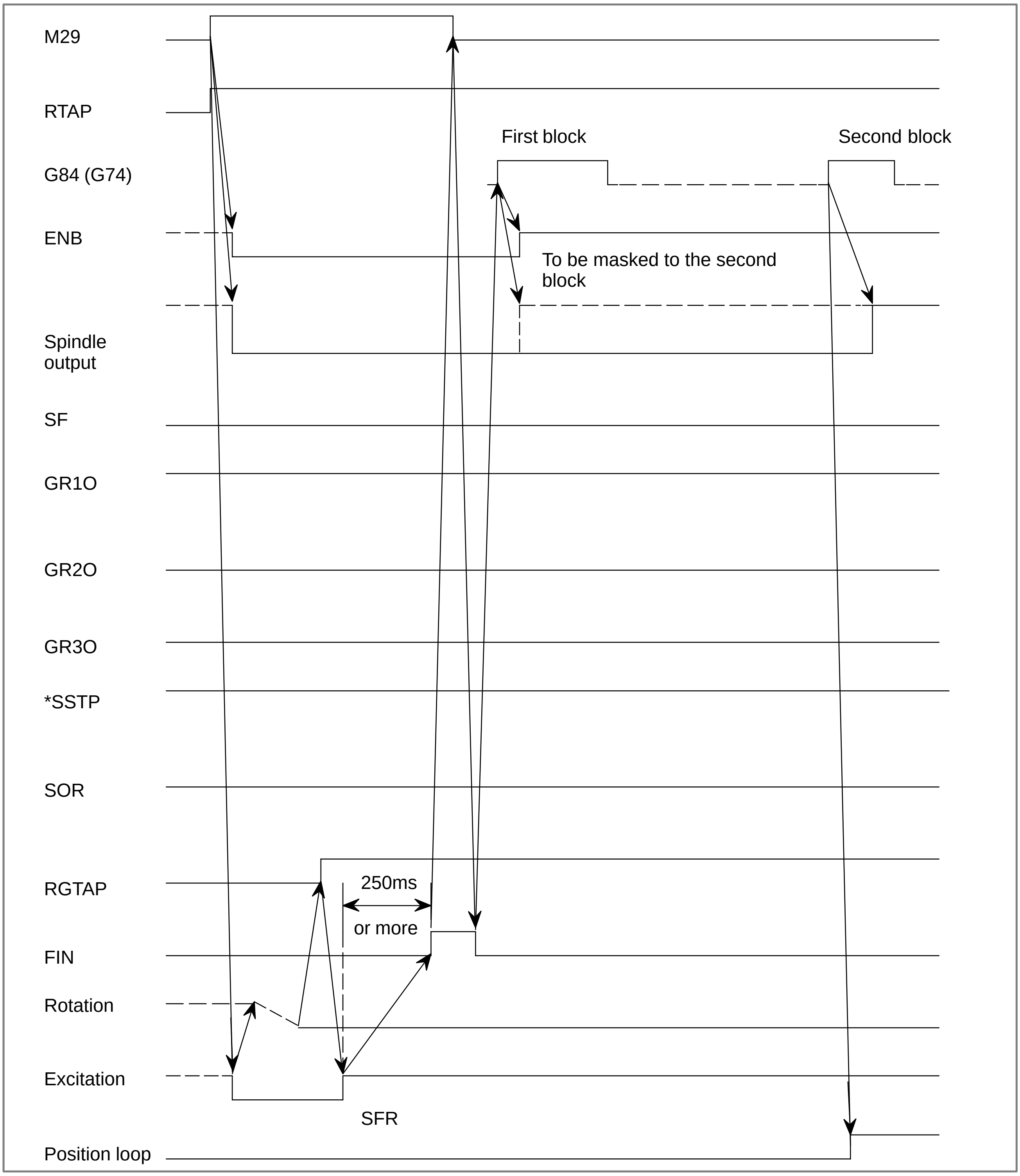
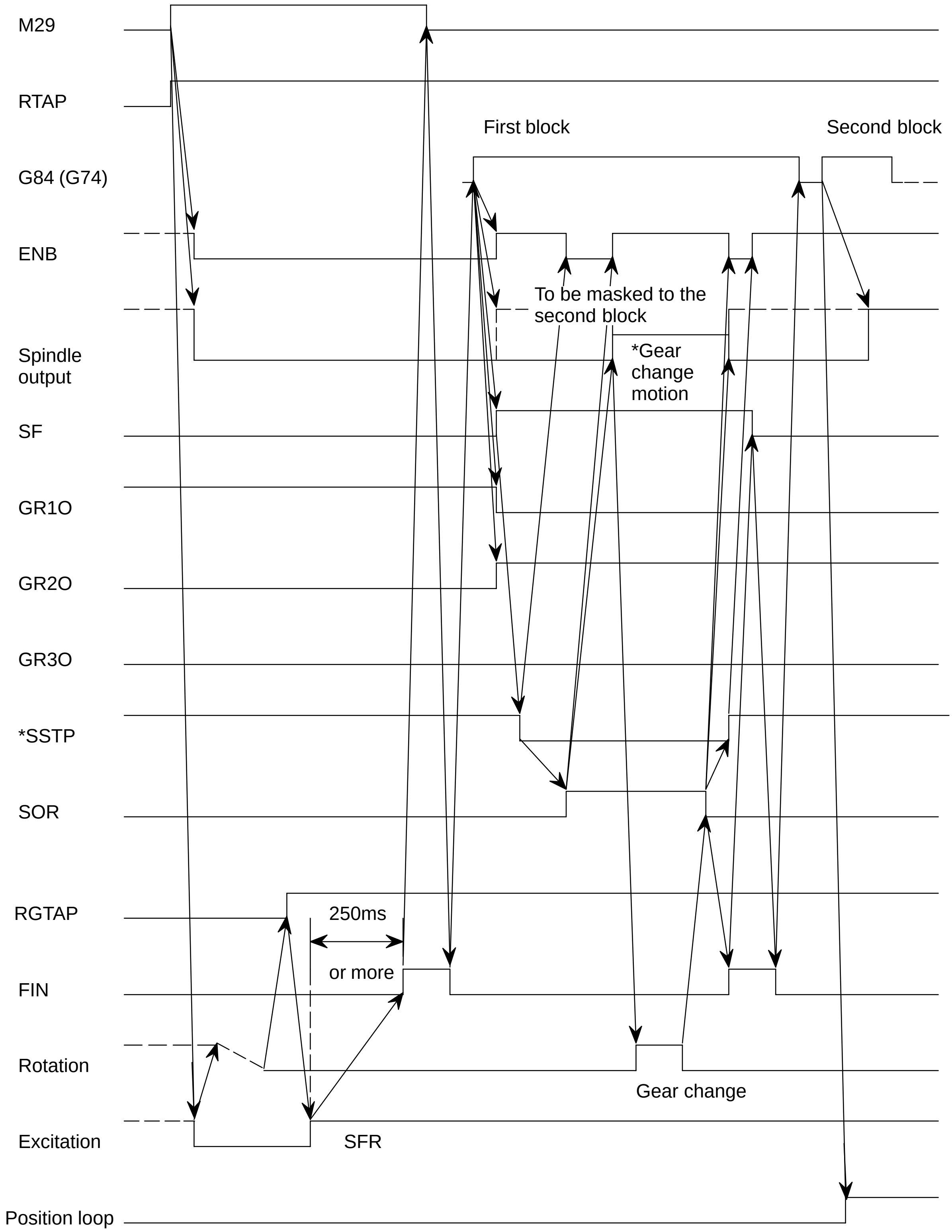


Fig. 9.10.7.1 (a) Gear is not changed



Note This time chart show an example where the gear has shifted from low to middle gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has turned from "0" to "1". This changes the gear.

Fig. 9.10.7.1 (b) When gear change is performed (from low to middle gear)

T type gear selection method

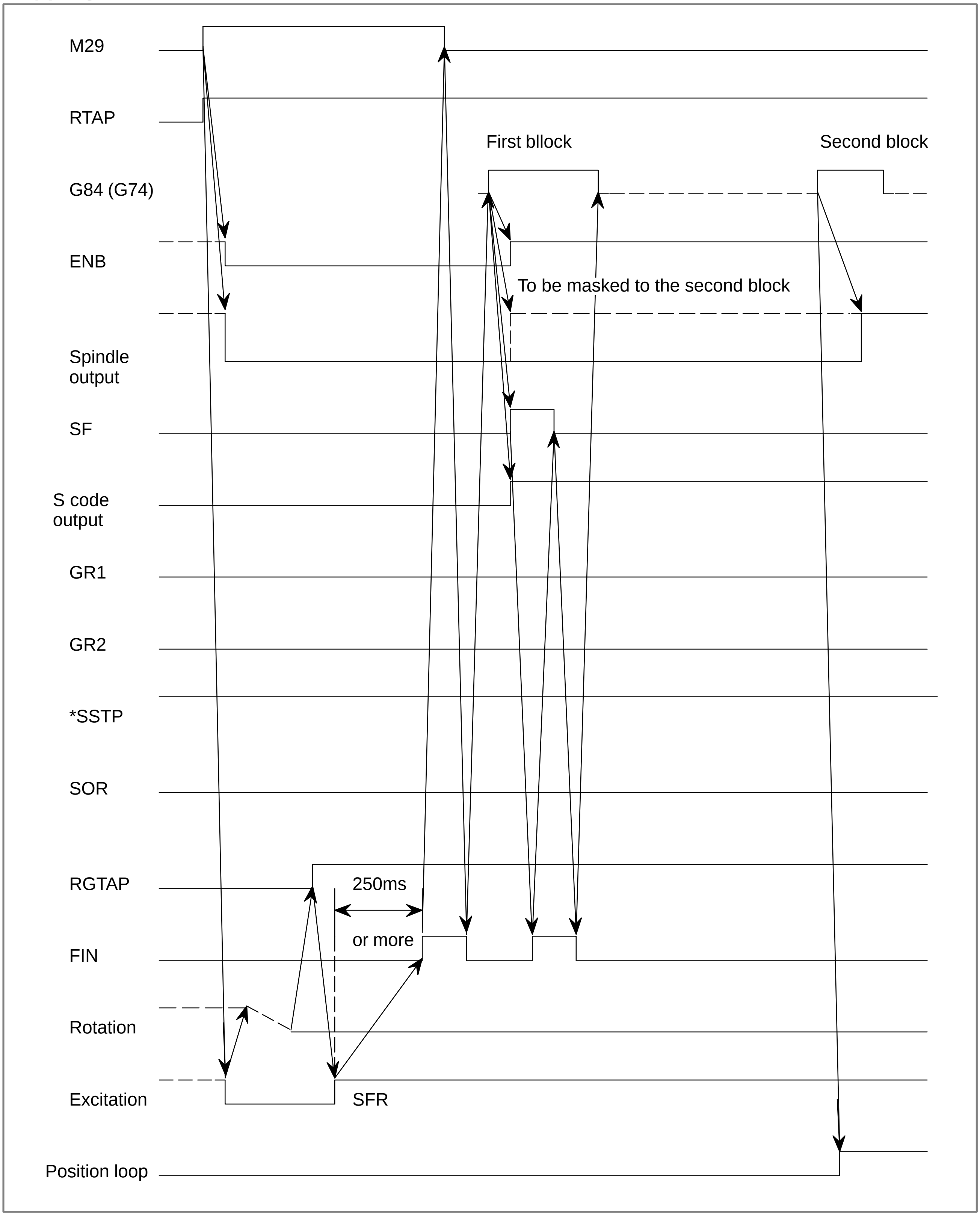
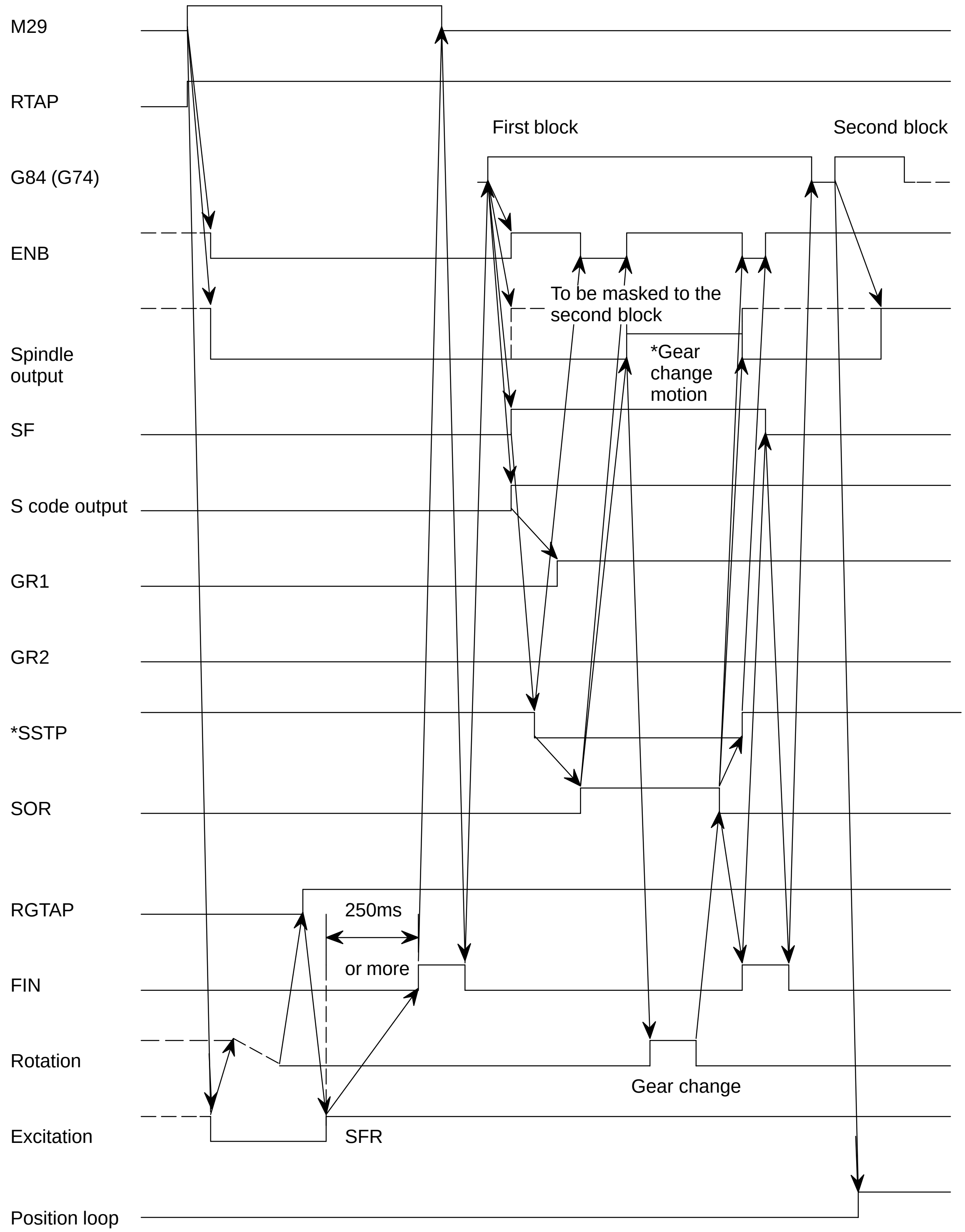


Fig. 9.10.7.1 (c) Gear change is not performed



Note This time chart shows an example where the gear has changed from low to middle gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.

Fig. 9.10.7.1 (d) When gear–change is performed (low to middle gear)

9.10.7.2
M29 and G84 (G74) are specified in the same block
M type gear selection

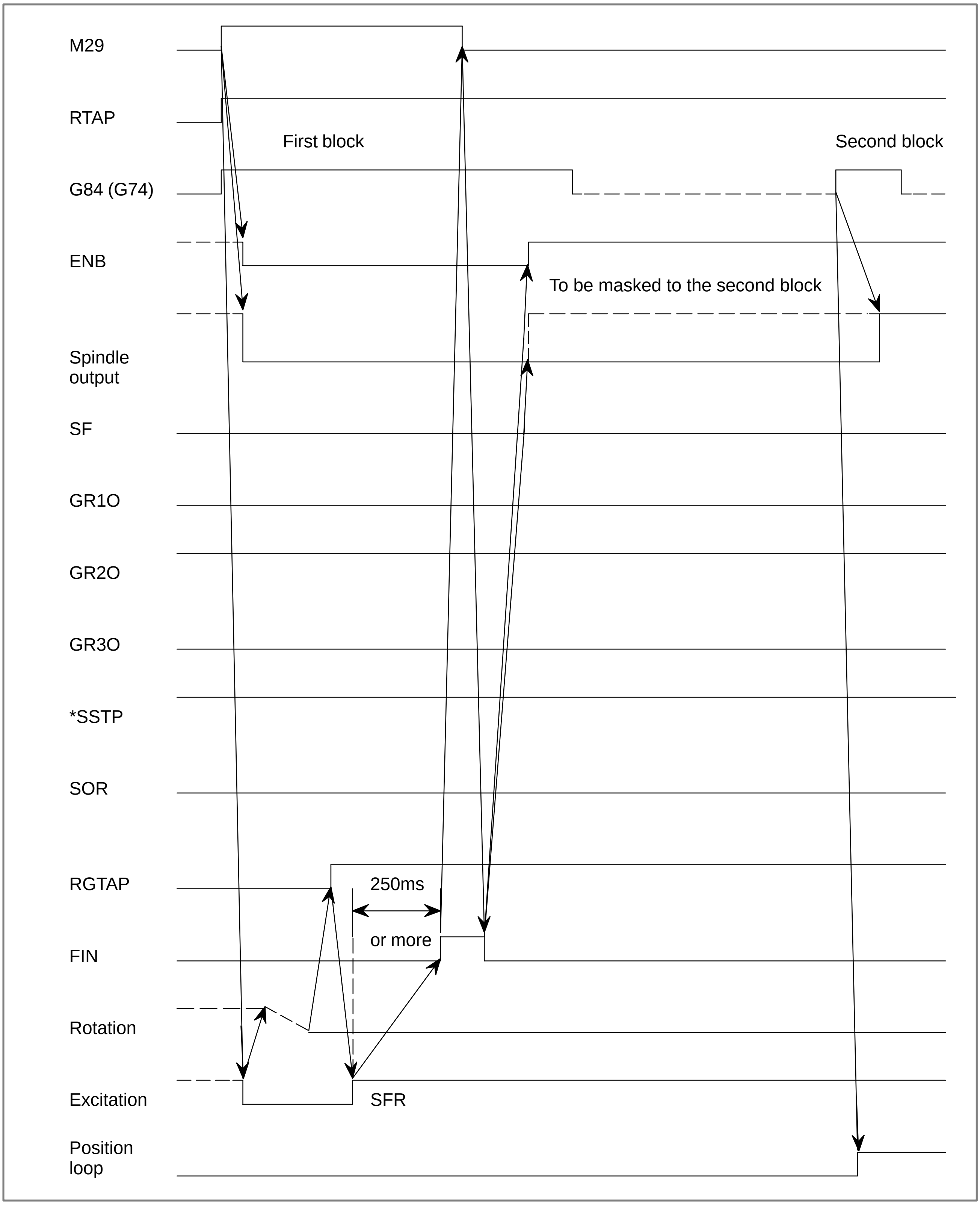
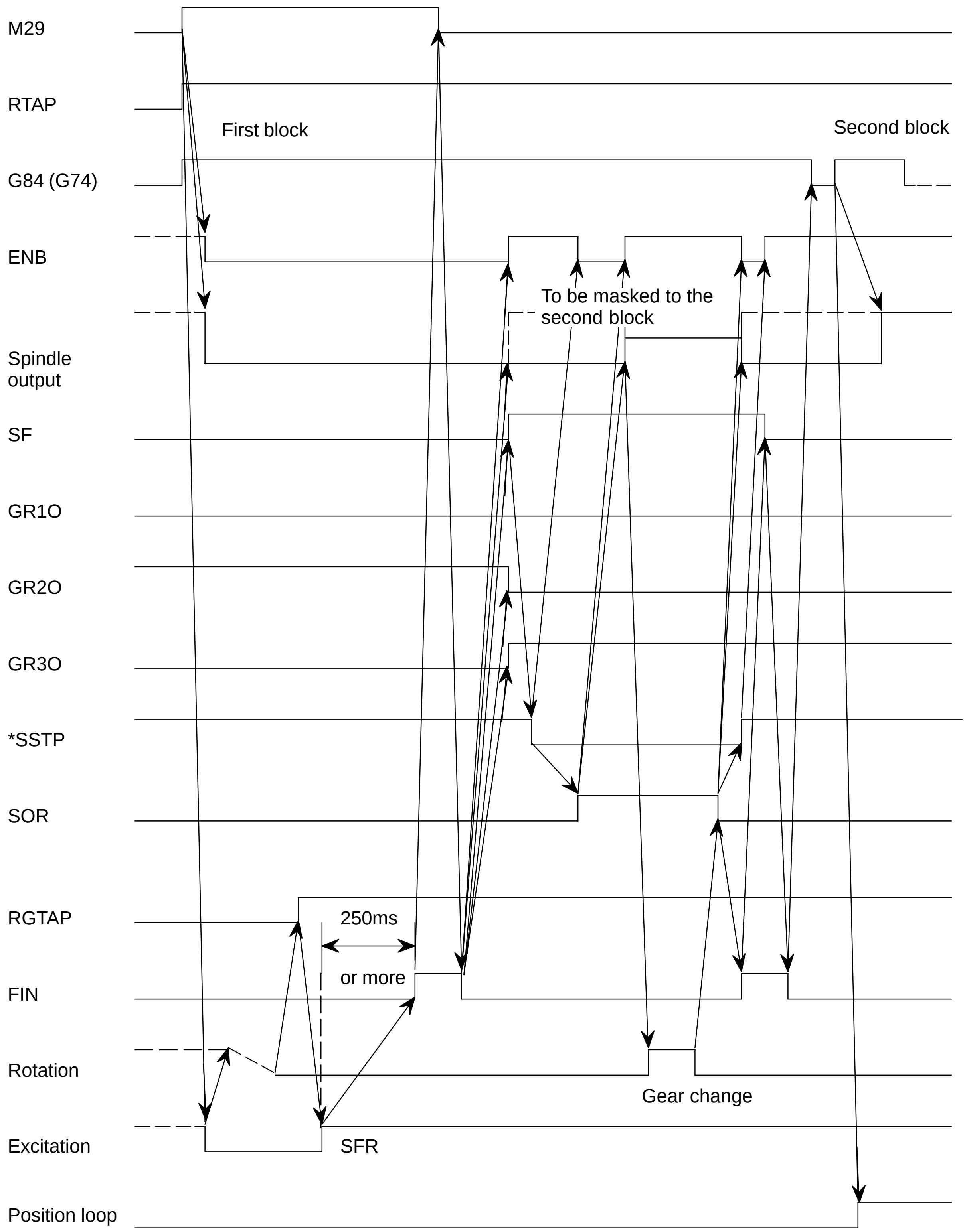


Fig. 9.10.7.2 (a) When gear–change is not performed



Note This time chart shows an example where the gear has shifted from middle high gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from "1" to "0", and one of the two remaining signals has tuned from "0" to "1". This changes the gear.

Fig. 9.10.7.2 (b) When gear–change is performed (middle to high)

T type gear selection method

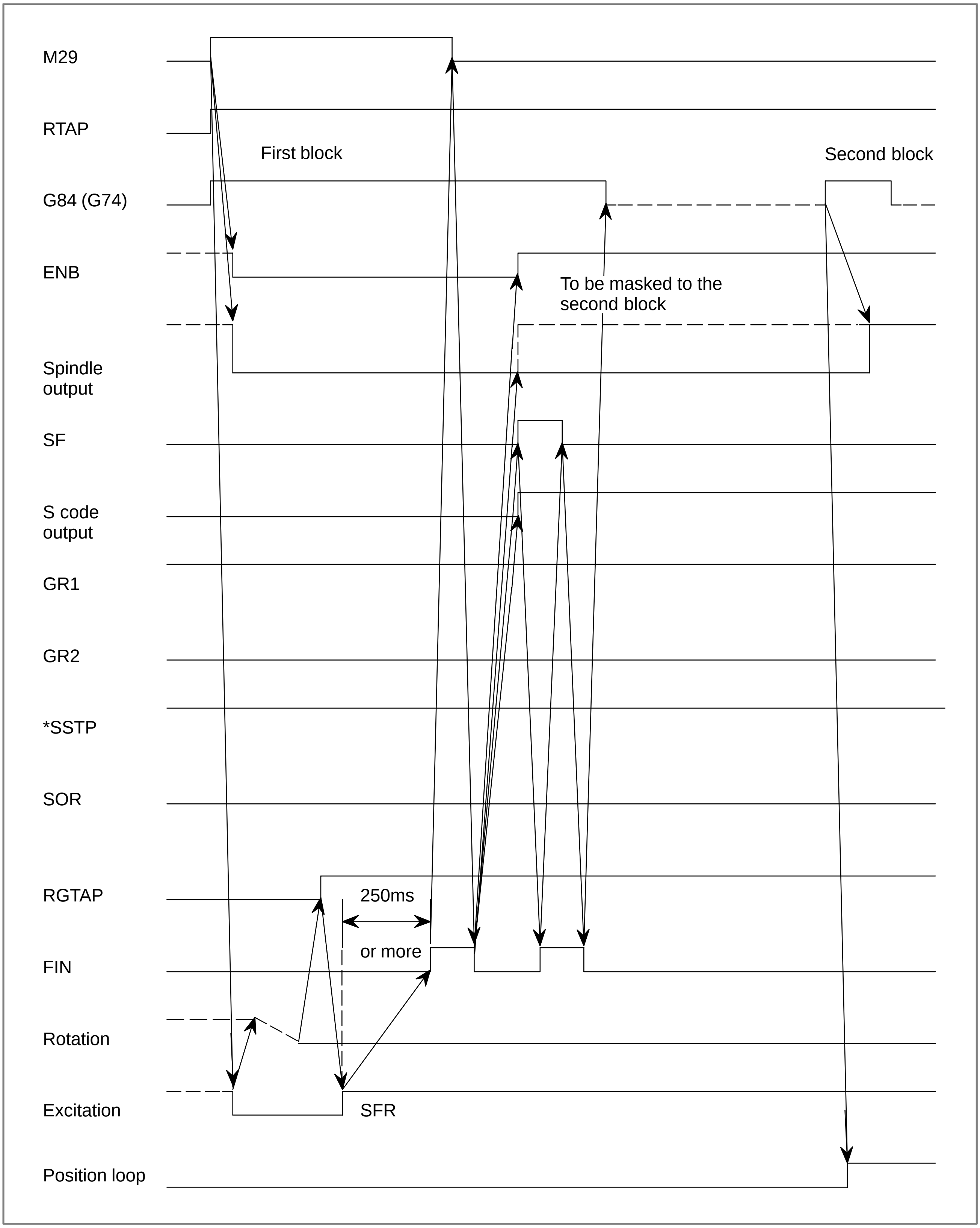
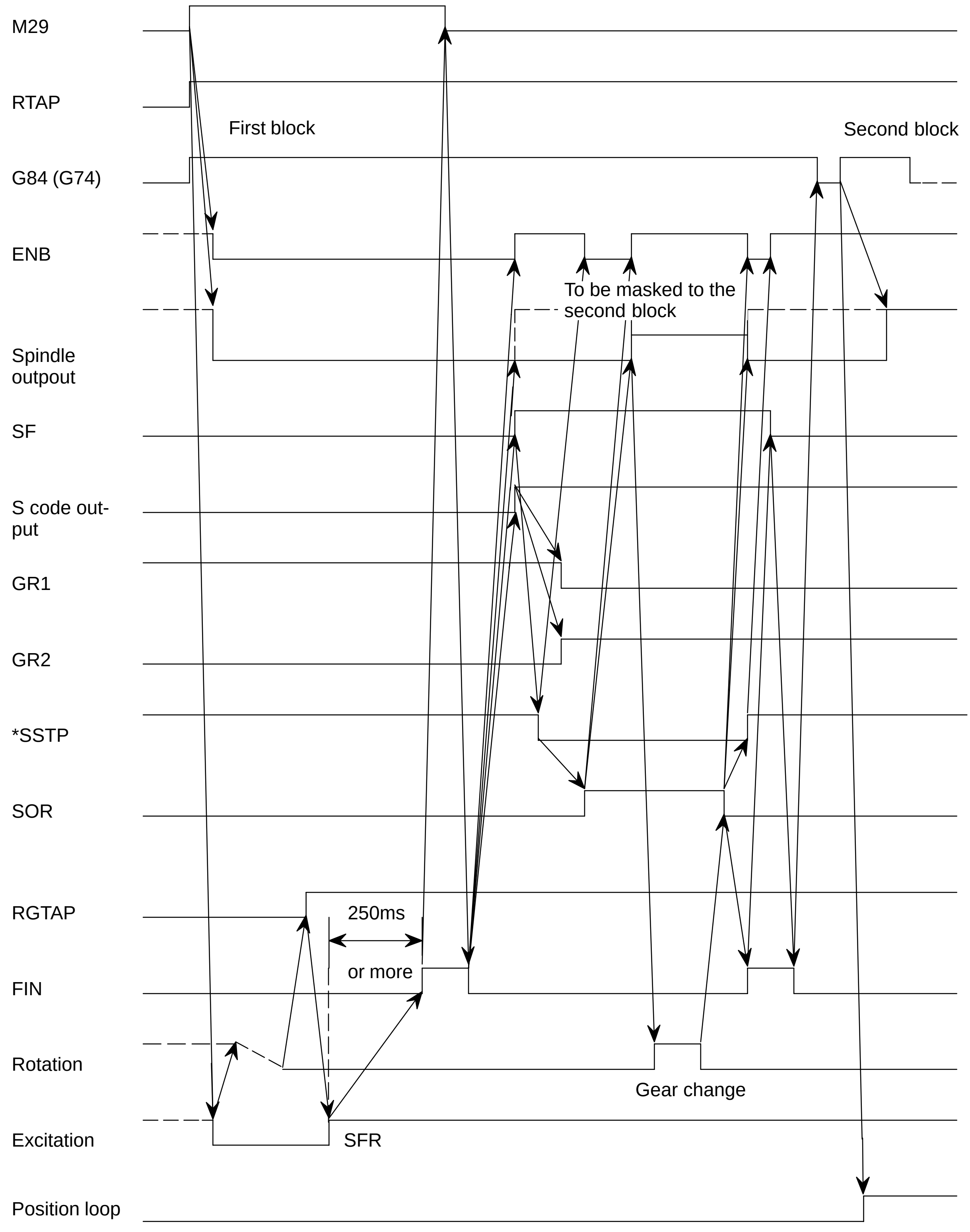


Fig. 9.10.7.2 (c) When gear change is not performed



Note This time chart shows an example where the gear has changed from middle to high gear. The PMC selects a required gear through an S code output and selects the gear using the GR1 and GR2 signals to inform CNC of the selected gear.

Fig 9.10.7.2 (d) When gear–change is performed (middle to high gear)

9.10.7.3

Specifying G84 (G74) for rigid tapping by parameters

M type gear selection

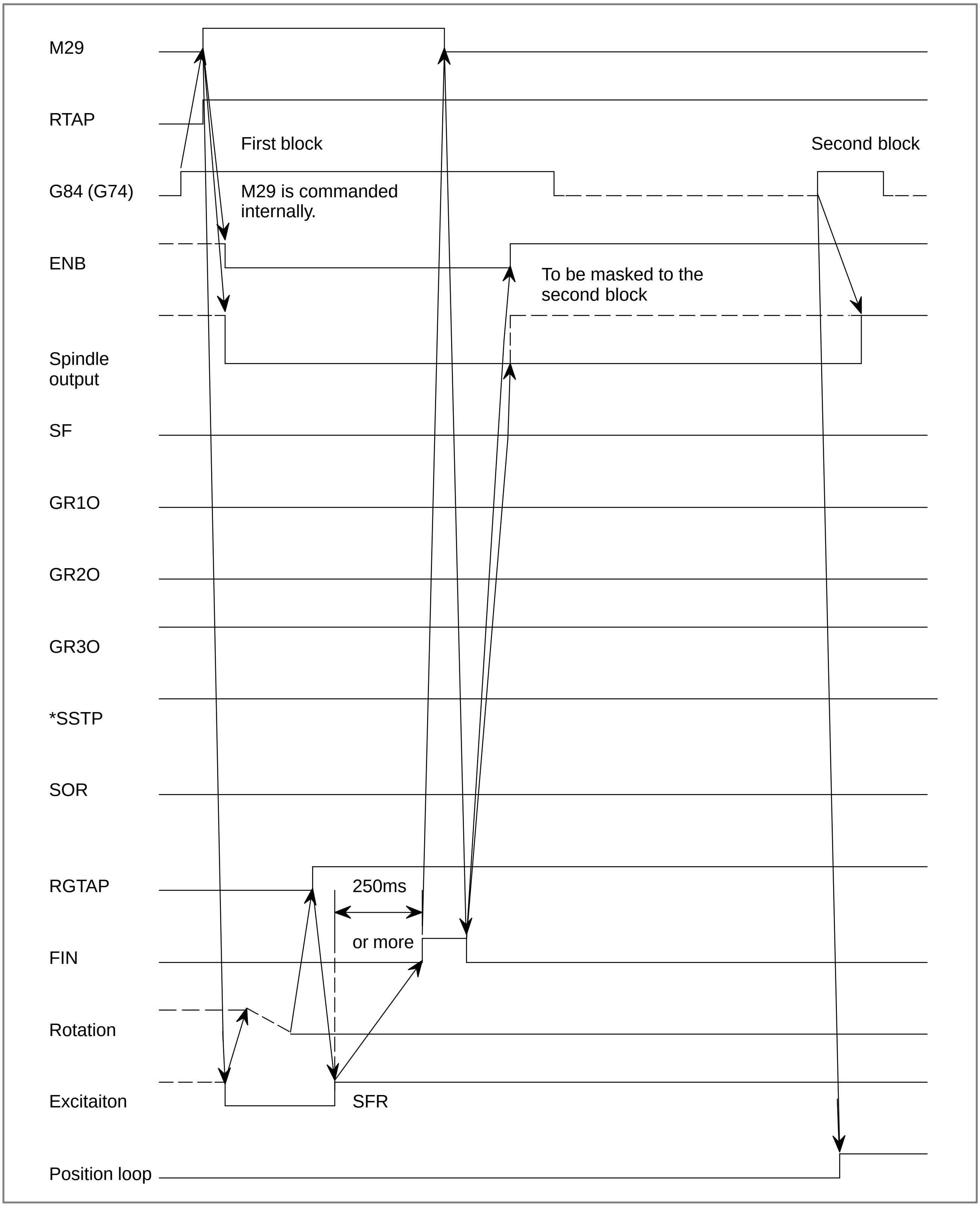
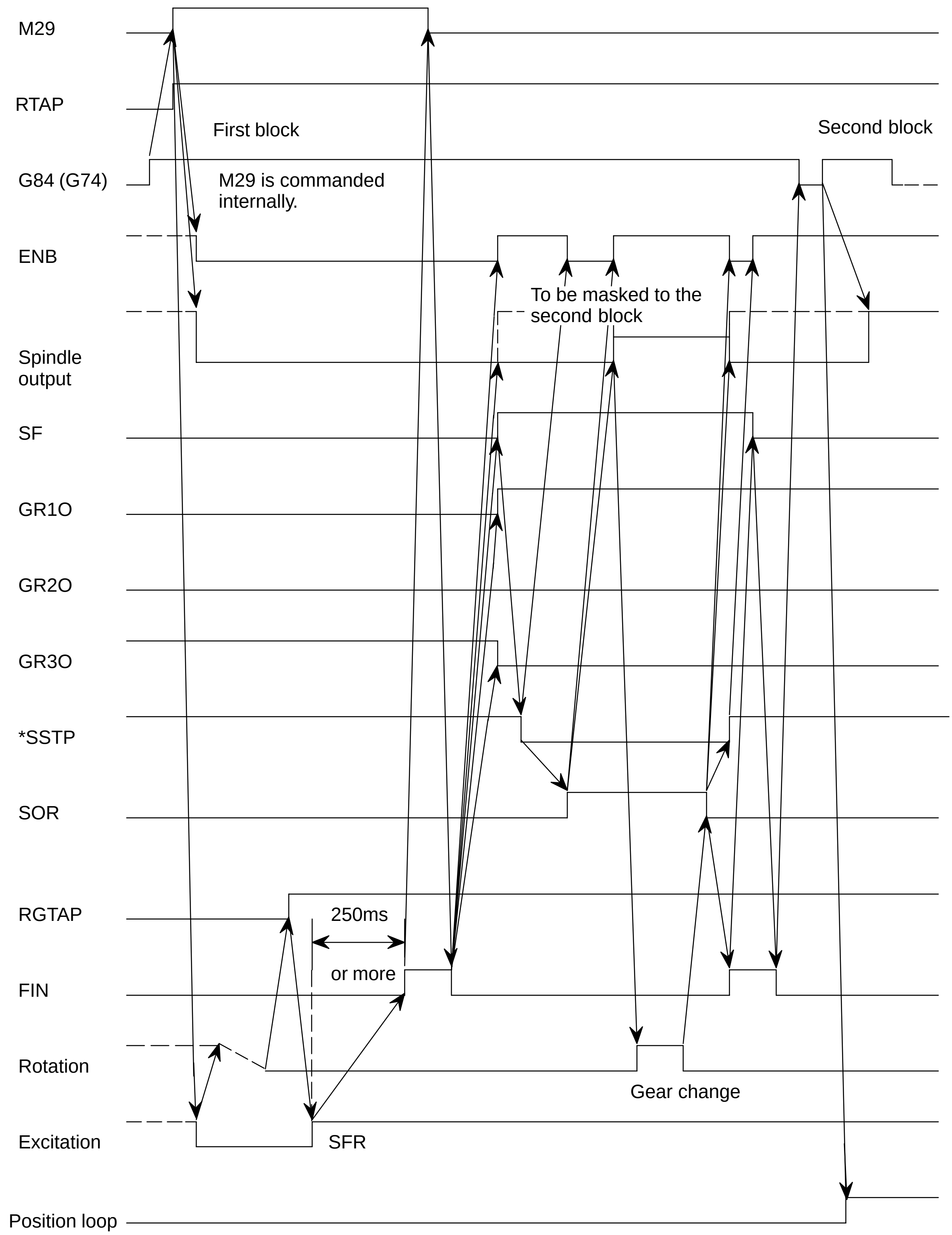


Fig. 9.10.7.3 (a) When gear–change is not performed



Note This time chart shows an example where the gear has shifted from high to low gear. One of the gear select signals (GR1O, GR2O, GR3O) has turned from “1” to “0”, and one of the two remaining signals has turned from “0” to “1”. This changes the gear.

Fig. 9.10.7.3 (b) When gear change is performed (high to low gear)

T type gear selection method

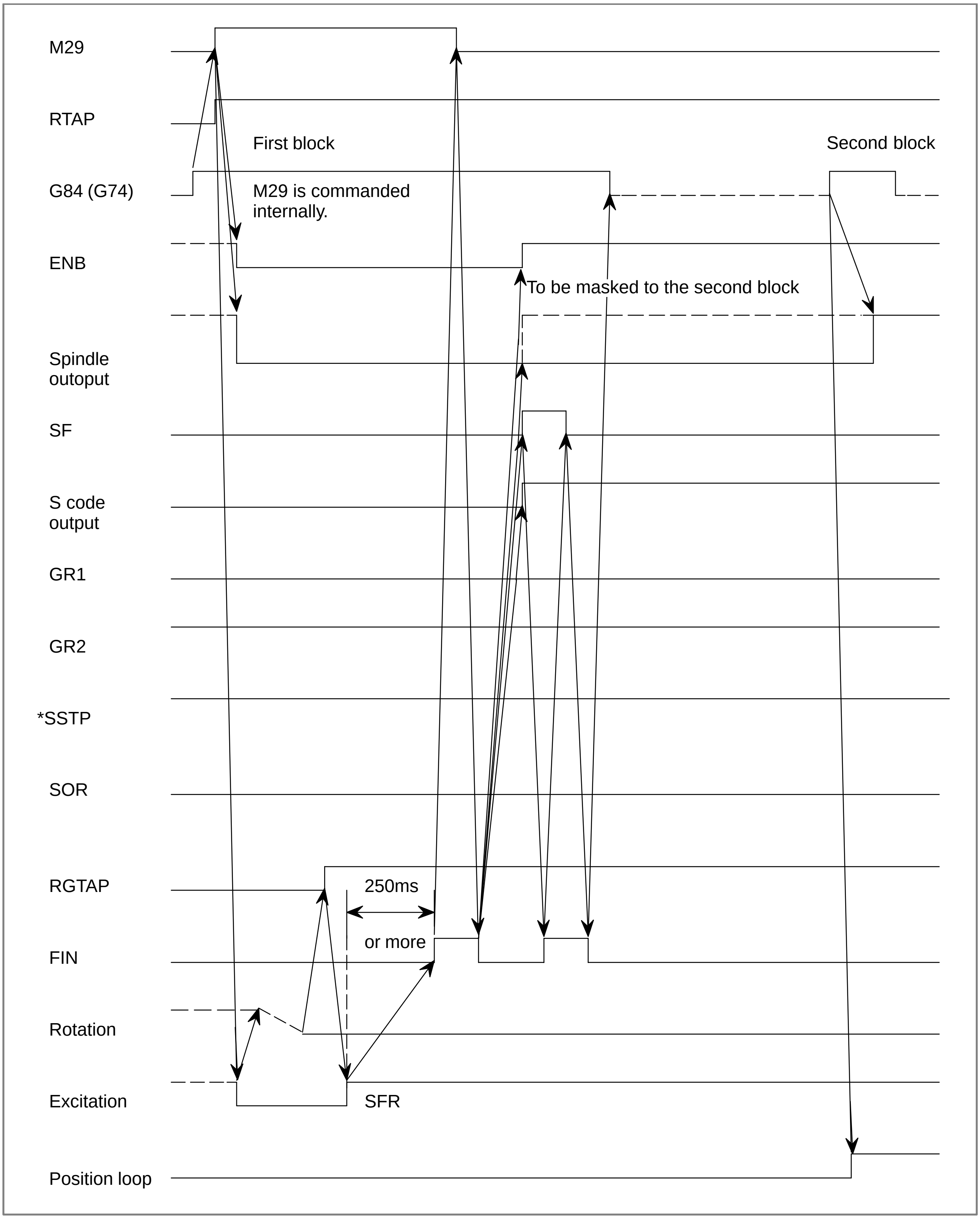
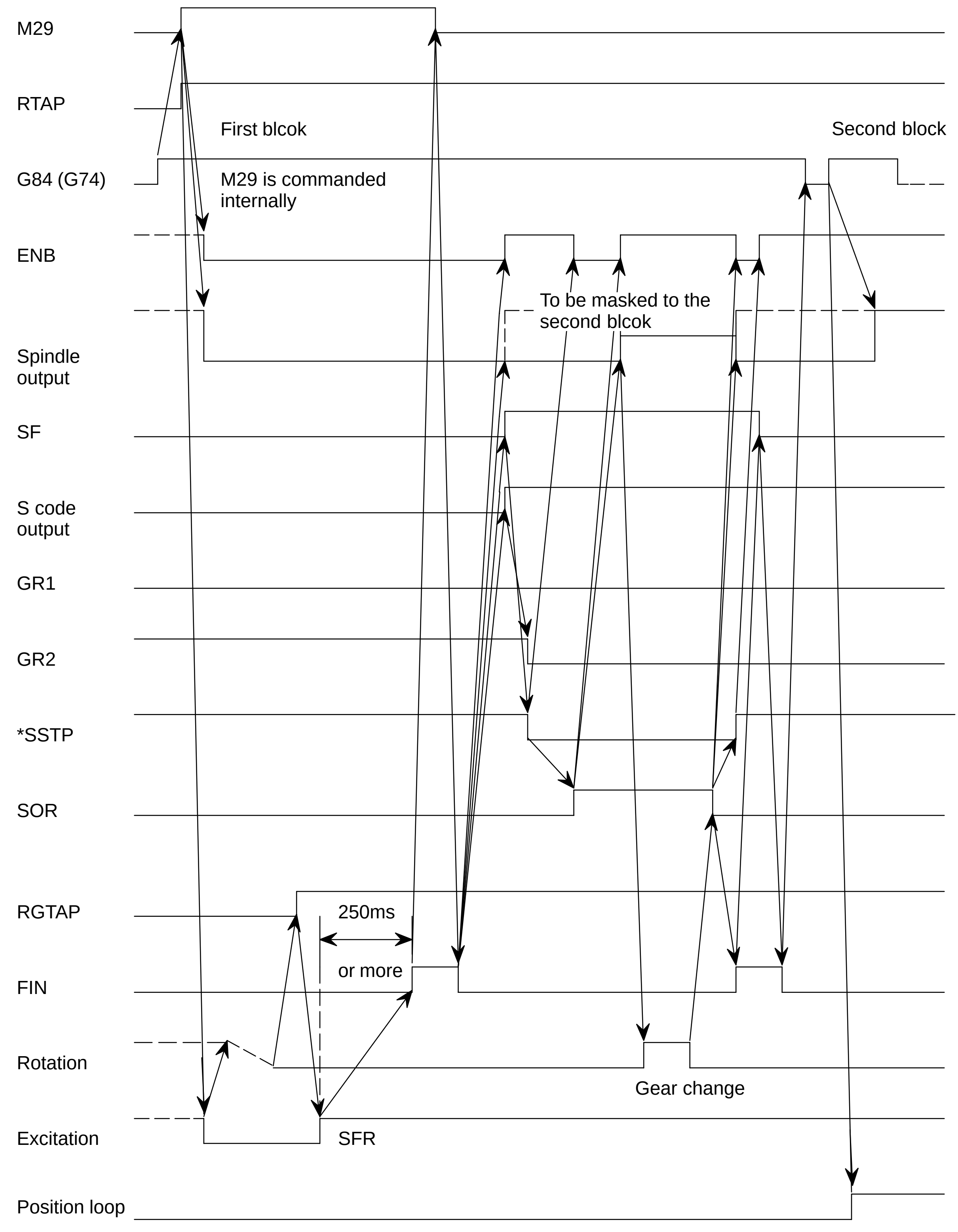


Fig. 9.10.7.3 (c) When gear change is not performed



Note This time chart shows an example where the gear has changed from high to low gear. The PMC selects a required gear through an S code output and inputs the selected gear using the GR1 and GR2 signals to inform CNC of the selected gear.

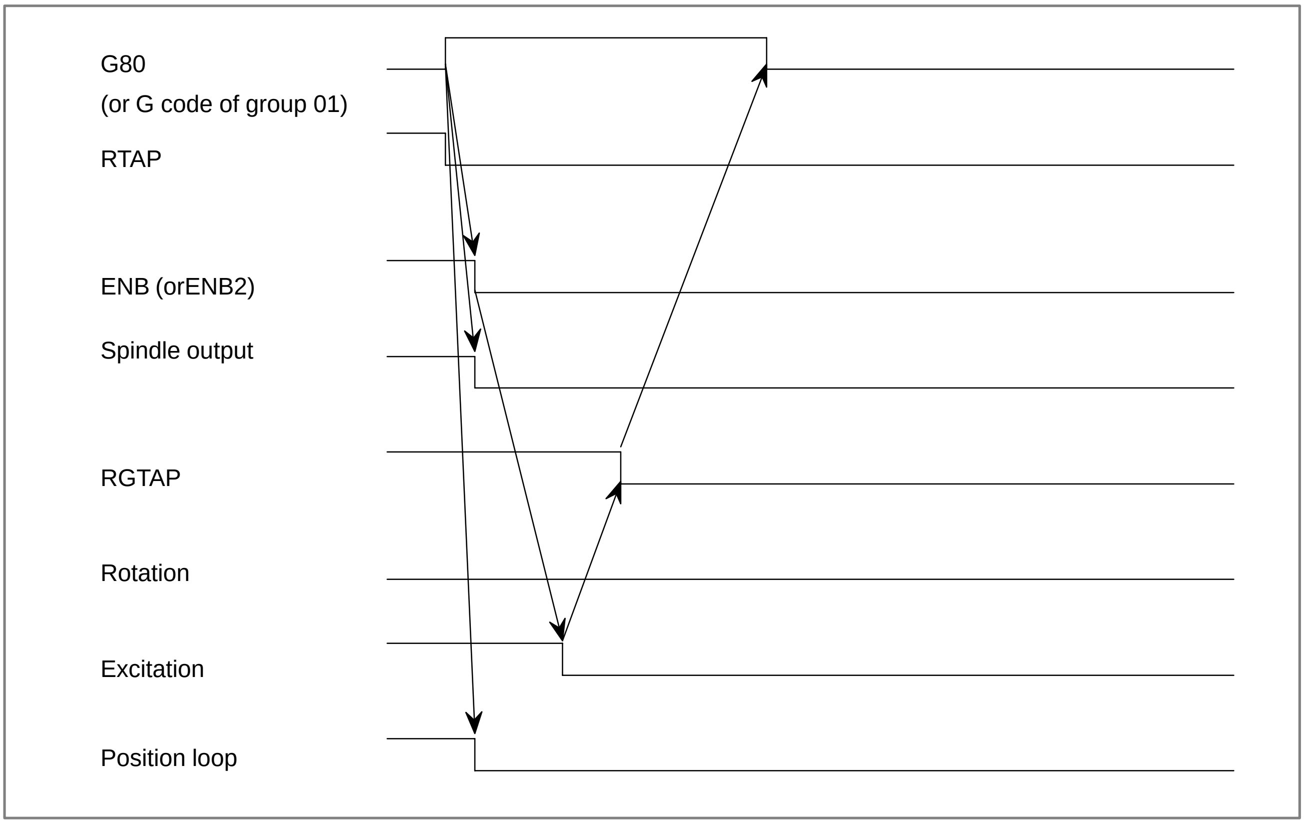
Fig. 9.10.7.3 (d) When gear–change is performed (high to low gear)

9.10.7.4 Timing to cancel rigid tapping mode

When rigid tapping is completed, the mode is canceled if a G code (such as G80, canned cycle G code, or Group 01 G code) is issued.
The S command used during rigid tapping is automatically cleared when rigid mode is cancelled. This reduces the spindle output to 0, placing the system in the state in which the S0 command is specified. Cancel the PMC rigid tapping mode at the falling edge of the ENB signal (ENB2 signal for 2nd spindle) by de-energizing the spindle; then turn off the rigid tapping mode signal. The system goes to the next block after confirming that the signal is off.

When gear change is performed using ∗SSTP and SOR, the ENB signal can be either "1" or "0". Do not cancel the PMC's rigid tapping mode at the falling edge of the ENB signal under these circumstances. The position loop is also canceled.

When the CNC is reset, the PMC's rigid tapping mode must be canceled. When CRG (parameter No. 5200#2) is "1", the system goes directly to the next block without checking that the rigid tapping signal is "0". Set CRG to "1" for systems in which the rigid tapping signal is always "1".



WARNING

1 If rigid tapping mode is canceled by a Group 01 G code, such as G00 or G01, the block containing the G code is executed at the same time the ENB signal is turned to “0”. Therefore, if the block contains an M code for controlling the spindle, an error may occur during processing in the PMC.

2 When CRG (Parameter No. 5200#2) is 1, if the next block contains an M code for controlling the spindle, an error may occur during processing in the PMC, when:
   - Rigid tapping mode is canceled by issuing G80
   - Rigid tapping mode is canceled by issuing a Group 01 G code , such as G00 or G01

NOTE

Rigid tapping mode is canceled as described above regardless of the gear selection method of M–type or T–type.

9.10.8 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT			ESF

[Data type] Bit

ESF When the spindle control function (S analog output or S serial output) is used, and the consatant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:

0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

NOTE
For the T series, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.
For the M series, SF is not output:
- For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode
- When bit 5 (NSF) of parameter No. 3705 is set to 1

SGT Gear switching method during tapping cycle (G84, G74)

0 : Method A (Same as normal gear switching method)
1 : Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters 3761 and 3762)

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.
1 : Output for an S command.

NOTE
The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S–––;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface–speed control,

0 : SF is output.
1 : SF is not output:

SFA: The SF signal is output:

0 : When gears are switched
1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PG2	PG1
				GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method

0: Type M
1 : Type T

NOTE
1 Type M:
The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in a parameter. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.
2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.

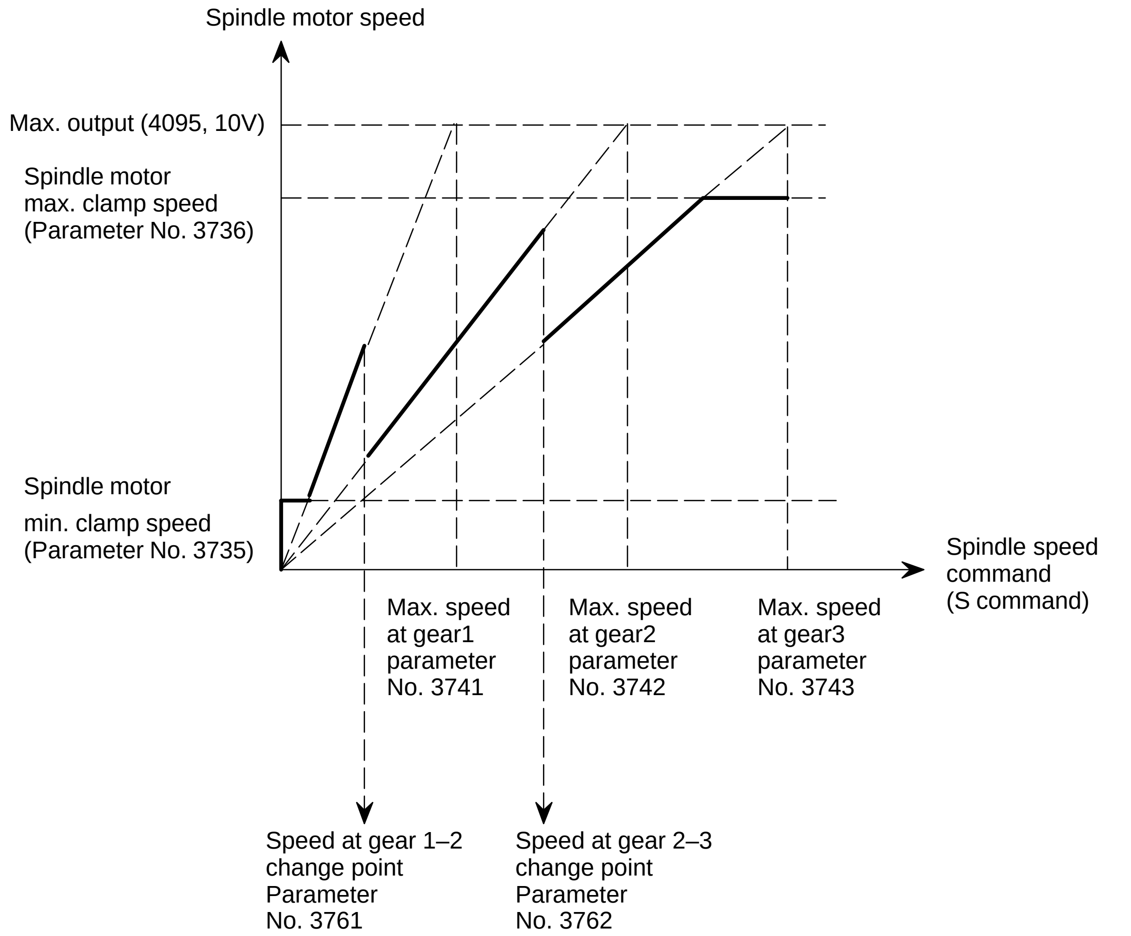
3761	Spindle speed when switching from gear 1 to gear 2 during tapping
3762	Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] min^{-1}

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



	#7	#6	#5	#4	#3	#2	#1	#0
5101								
								FXY

[Data type] Bit

FXY The drilling axis in the drilling canned cycle is:

0 : Always the Z–axis
1 : The axis selected by the program

NOTE
For the M series, this parameter enables rigid tapping by using a basic axis (X, Y, or Z) perpendicular to the program–selected plane, or an axis parallel to that basic axis, as the tapping axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD		DOV	SIG	CRG	VGR	G84
		FHD	PCP	DOV	SIG	CRG	VGR	G84

[Data type] Bit

G84 Method for specifying rigid tapping
0 : An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
1 : An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

VGR Any gear ratio between spindle and position coder in rigid tapping
0 : Not used (The gear ratio is set in parameter No.3706.)
1 : Used (The gear ratio is set by parameters Nos. 5221 through 5224 and 5231 through 5234.)

NOTE
For serial spindles, set this parameter to 0 when using the DMR function for position coder signals on the spindle side.

CRG Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.)
0 : Canceled after rigid tapping signal RGTAP is set to “0”.
1 : Canceled before rigid tapping signal RGTAP is set to “0”.

SIG When gears are changed for rigid tapping, the use of SIND <G032 and G033> is
0 : Not permitted.
1 : Permitted.

DOV Override during extraction in rigid tapping
0 : Invalidated
1 : Validated (The override value is set in parameter No.5211.)

PCP Rigid tapping
0 : Used as a high–speed peck tapping cycle
1 : Not used as a high–speed peck tapping cycle

FHD Feed hold and single block in rigid tapping
0 : Inhibited
1 : Enabled

SRS To select a spindle used for rigid tapping in multi–spindle control:
0 : The spindle selection signals SWS1 and SWS2 (bits 0 and 1 of G027) are used. (These signals are used also for multi–spindle control.)
1 : The rigid tapping spindle selection signals RGTSP1 and RGTSP2 (bits 4 and 5 of G061) are used. (These signals are provided expressly for rigid tapping.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU	TDR		
				OV3	OVU	TDR		NIZ

[Data type] Bit

NIZ Smoothing in rigid tapping is:
0 : Not performed.
1 : Performed.

TDR Cutting time constant in rigid tapping
0 : Uses a same parameter during cutting and extraction (Parameter Nos. 5261 through 5264)
1 : Not use a same parameter during cutting and extraction
Parameter Nos. 5261 to 5264: Time constant during cutting
Parameter Nos. 5271 to 5274: Time constant during extraction

OVU The increment unit of the override parameter (No.5211) for tool rigid tapping extraction is:
0 : 1%
1 : 10%

OV3 A spindle rotation speed for a pull–out operation is specified by program, so the tool is pulled out while the spindle is rotating at the specified spindle rotation speed.
0 : Disabled.
1 : Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5202								
								ORI

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

[Data type] Bit

ORI When rigid tapping is started:
0 : Spindle orientation is not performed.
1 : Spindle orientation is performed.

NOTE
This parameter can be used only for a serial spindle.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS	RGS	RFF		
				OVS		RFF		

[Data type] Bit

REF Feed forward during movement from the initial point to point R in rigid tapping is:
0 : Disabled.
1 : Enabled.

When this parameter is set, the following function is also enabled:

- When rigid tapping is specified in advanced preview control mode, the system automatically exits from advanced preview control mode and executes rigid tapping. After termination of rigid tapping, the system automatically returns to advanced preview control mode.

RGS When bit 0 (MIF) of parameter No. 1403 is set to 1 and rigid tapping is specified in feed–per–minute mode, the spindle speed becomes:
0 : 1/1000 of the specified speed.
1 : 1/1 of the specified speed.

OVS In rigid tapping, override by the feedrate override signal and invalidation of override by the override cancel signal is:
0 : Disabled.
1 : Enabled.

Setting this parameter enables override by the feedrate override signal <G012> to be applied for rigid tapping operation (cutting and extraction) in rigid tapping.

The spindle speed override is fixed to 100%, but override is also applied to the spindle speed in synchronization with the feedrate along the tapping axis by feedrate override.

The override cancel signal OVC <bit 4 of G006> also become available.

NOTE
1 When this parameter is set to override the feedrate, override by parameters (see parameters Nos. 5211 (T/M) and 5381 (M)) is disabled.
2 Regardless of whether this parameter is set, when feedrate override is disabled by the override cancel signal OVC <bit 4 of G006>, override by parameters (see parameters Nos. 5211 (T/M) and 5381 (M)) is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5204							SPR	DGN

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

[Data type] Bit

DGN On the diagnosis screen:
0 : A rigid tapping synchronization error is displayed. (Nos. 455 to 457)
1 : An error difference between the spindle and tapping axis is displayed. (Nos. 452 and 453)

SPR In rigid tapping, the parameters are:
0 : Not changed on a spindle–by–spindle basis.
1 : Changed on a spindle–by–spindle basis.

NOTE
1 When switching between the rigid tapping parameters on a spindle–by–spindle basis in rigid tapping using the second serial spindle, set this parameter to 1. The following parameters are supported for each spindle:

First spindle (4–stage gear)	Second spindle (2–stage gear)
No.5214	No.5215
No.5221 to No.5224	No.5225, No.5226
No.5231 to No.5234	No.5235, No.5236
No.5241 to No.5244	No.5245, No.5246
No.5261 to No.5264	No.5265, No.5266
No.5271 to No.5274	No.5335, No.5336
No.5280	No.5341
No.5281 to No.5284	No.5342, No.5343
No.5300, No.5301	No.5302, No.5303
No.5310 to No.5314	No.5350 to No.5353
No.5321 to No.5324	No.5325, No.5326

2 For rigid tapping using the second serial spindle, the multispindle control is required.

	#7	#6	#5	#4	#3	#2	#1	#0
5205								RCK
						NRV		RCK

[Data type] Bit

RCK In rigid tapping, an excessive error during movement/at stop is:
0 : Checked regardless of whether mode is cutting (tapping) or rapid traverse.
1 : Checked only in cutting (tapping) mode.

NRV For the rigid tapping function, the spindle returns back from the bottom of a hole with:
0 : Rotating opposite to the drilling direction
1 : Rotating in the drilling direction (special purpose)

NOTE
When you want to perform rigid tapping, do not set this parameter.
If rigid tapping is performed with this parameter set, a tapping tool, workpiece, or machine may be damaged.

5210	Rigid tapping mode specification M code

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets an M code that specifies the rigid tapping mode.

NOTE
1 The M code is judged to be 29 (M29) when "0" is set.
2 To use an M code whose number is greater than 255, Specify the code number with parameter No.5212.

5211	Override value during rigid tapping extraction

[Data type] Byte

[Unit of data] 1 % or 10 %

[Valid data range] 0 to 200

The parameter sets the override value during rigid tapping extraction.

NOTE
The override value is valid when DOV in parameter No.5200 #4 is "1".
When OVU (bit 3 of parameter No.5201) is 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

5212	M code that specifies a rigid tapping mode

[Data type] 2–word

[Unit of data] Integer

[Valid data range] 0 to 65535

This parameter sets the M code that specifies the rigid tapping mode.

The M code that specifies the rigid tapping mode is usually set by parameter 5210. To use an M code whose number is greater than 255, specify the code number with parameter 5212.

NOTE
If the setting of this parameter is 0, the M code specifying the rigid tapping mode is determined by the setting of parameter 5210. Otherwise, it is determined by the setting of parameter 5212. The setting of parameter 5212 must always be within the above valid range.

5213	Return or clearance in peck tapping cycle

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter input	0.01	0.001	0.0001	mm
Input in incluse	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the return or clearance in the peck tapping cycle.

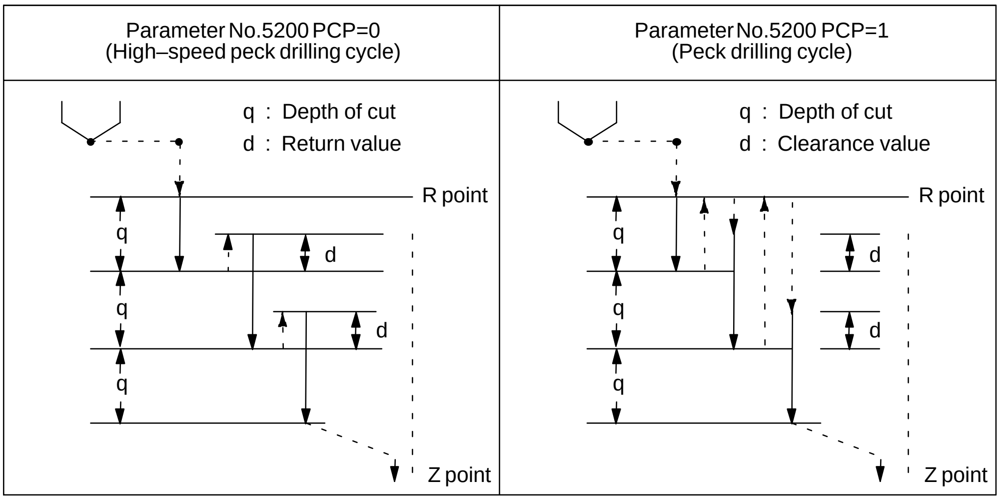


Fig.9.10.8 (a) High–speed Peck Drilling and Peck Drilling Cycles

5214	Setting of an allowable rigid tapping synchronization error range
5215	Setting of an allowable rigid tapping synchronization error range for the second spindle

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Each of these parameters is used to set an allowable synchronization error range between a spindle used for rigid tapping and the tapping axis.

If the value set with each parameter is exceeded, rigid tapping alarm No.741 (excessive error during movement) is issued. When 0 is set, a synchronization error check is not made.

NOTE

When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5214 is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5215 are applied to the second spindle, respectively.

5221	Number of spindle gear teeth (first–stage gear)
5222	Number of spindle gear teeth (second–stage gear)
5223	Number of spindle gear teeth (third–stage gear)
5224	Number of spindle gear teeth (fourth–stage gear)
5225	Number of second spindle gear teeth (first–stage gear)
5226	Number of second spindle gear teeth (second–stage gear)

[Data type] Word

[Valid data range] 1 to 32767

When an arbitrary gear ratio is used in rigid tapping, each of these parameters sets the number of teeth of each spindle gear.

NOTE

1 These parameters are enabled when the VGR parameter (bit 1 of parameter No.5200) is set to 1.

2 When a position coder is attached to the spindle, set the same value for all of parameters No.5221 through No.5224.

3 When the DMR function of the position coder signal is used with a serial spindle, set the VGR parameter (bit 1 of parameter No.5200) to 0, and set these parameters to 0.

4 When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameters No.5221 and No.5222 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5225 and No.5226 are applied to the second spindle.

5231	Number of position coder gear teeth (first–stage gear)
5232	Number of position coder gear teeth (second–stage gear)
5233	Number of position coder gear teeth (third–stage gear)
5234	Number of position coder gear teeth (fourth–stage gear)
5235	Number of position coder gear teeth for the second spindle (first–stage gear)
5236	Number of position coder gear teeth for the second spindle (second–stage gear)

[Data type] Word

[Valid data range] 1 to 32767

When an arbitrary gear ratio is used in rigid tapping, each of these parameters sets the number of teeth of each position coder gear.

NOTE

1 These parameters are enabled when the VGR parameter (bit 1 of parameter No.5200) is set to 1.
When a position coder is attached to the spindle, set the same value for all of parameters No.5231 through No.5234.
When a spindle motor with a built–in position coder is used, a position coder with a resolution of 2048 pulses/rev may be used. In such a case, set the actual number of teeth, multiplied by 2 (for conversion to 4096 pulses/rev).

2 When the DMR function of the position coder signal is used with a serial spindle, set the VGR parameter (bit 1 of parameter No.5200) to 0, and set these parameters to 0.

3 When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameters No.5231 and No.5232 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5235 and No.5236 are applied to the second spindle.

5241	Maximum spindle speed in rigid tapping (first–stage gear)
5242	Maximum spindle speed in rigid tapping (second–stage gear)
5243	Maximum spindle speed in rigid tapping (third–stage gear)
5244	Maximum spindle speed in rigid tapping (fourth–stage gear)
5245	Maximum spindle speed in rigid tapping using the second spindle (first–stage gear)
5246	Maximum spindle speed in rigid tapping using the second spindle (second–stage gear)

[Data type] 2–word

[Unit of data] min^{-1}

[Valid data range] Spindle position coder gear ratio

1:1 0 to 7400
1:2 0 to 9999
1:4 0 to 9999
1:8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.

NOTE

1 For the M series, set the same value for both parameter No.5241 and parameter No.5243 for a one–stage gear system. In a system with two gear stages, set parameter No. 5243 with the value specified in parameter No. 5241 or 5242 whichever is greater. Otherwise, P/S alarm No.200 will be issued.

2 When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameters No.5241 and No.5242 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5245 and No.5246 are applied to the second spindle.

5261	Linear acceleration/deceleration time constant for the spindle and tapping axis (first–stage gear)
5262	Linear acceleration/deceleration time constant for the spindle and tapping axis (second–stage gear)
5263	Linear acceleration/deceleration time constant for the spindle and tapping axis (third–stage gear)
5264	Linear acceleration/deceleration time constant for the spindle and tapping axis (fourth–stage gear)
5265	Linear acceleration/deceleration time constant for the second spindle and tapping axis (first–stage gear)
5266	Linear acceleration/deceleration time constant for the second spindle and tapping axis (second–stage gear)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/deceleration time constant for the spindle of each gear and the tapping axis in rigid tapping.

Set the period required to reach each maximum spindle speed (parameters No.5241 through No.5248). The set time constant, multiplied by the ratio of a specified S value to a maximum spindle speed, is actually used as a time constant.

NOTE

When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameters No.5261 and No.5262 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5265 and No.5266 are applied to the second spindle.

5271	Time constant for the spindle and tapping axis in extraction operation (first–stage gear)
5272	Time constant for the spindle and tapping axis in extraction operation (second–stage gear)
5273	Time constant for the spindle and tapping axis in extraction operation (third–stage gear)
5274	Time constant for the spindle and tapping axis in extraction operation (fourth–stage gear)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/deceleration time constant for the spindle of each gear and tapping axis in extraction operation during rigid tapping.

NOTE

1 These parameters are enabled when the TDR parameter (bit 2 of parameter No.5201) is set to 1.

2 When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameters No.5271 and No.5272 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5335 and No.5336 are applied to the second spindle.

5280	Position control loop gain for the spindle and tapping axis in rigid tapping (common to all gears)
5281	Position control loop gain for the spindle and tapping axis in rigid tapping (first–stage gear)
5282	Position control loop gain for the spindle and tapping axis in rigid tapping (second–stage gear)
5283	Position control loop gain for the spindle and tapping axis in rigid tapping (third–stage gear)
5284	Position control loop gain for the spindle and tapping axis in rigid tapping (fourth–stage gear)

NOTE

Once these parameters have been set, the power must be turned off then back on for the settings to become effective.

[Data type] Word

[Unit of data] 0.01 s^{-1}

[Valid data range] 1 to 9999

Each of these parameters is used to set a position control loop gain for the spindle and tapping axis in rigid tapping. These parameters significantly affect the precision of threading. Optimize these parameters as well as the loop gain multipliers by conducting a cutting test.

NOTE

1 To use a varied loop gain on a gear–by–gear basis, set parameter No.5280 to 0, and set a loop gain for each gear in parameters No.5281 through No.5284. The specification of a loop gain on a gear–by–gear basis is disabled if parameter No.5280 is set to a value other than 0. In such a case, the value set in parameter No.5280 is used as a loop gain that is common to all the gears.

2 When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5280 or the settings of parameters No.5281 and No.5282 are applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5341 through No.5343 are applied to the second spindle.

5291	Spindle loop gain multiplier in the rigid tapping mode (for gear 1)
5292	Spindle loop gain multiplier in the rigid tapping mode (for gear 2)
5293	Spindle loop gain multiplier in the rigid tapping mode (for gear 3)
5294	Spindle loop gain multioplier in the rigid tapping mode (for gear4)

[Data type] Word

[Valid data range] 0 to 32767

Set the spindle loop gain multipliers for gears 1 to 4 in the rigid tapping mode. The thread precision depends on the multipliers. Find the most appropriate multipliers by conducting the cutting test and assign them to the parameters.

NOTE
These parameters are used for analog spindles.

Loop gain multiplier = 2048 $\times$ E/L $\times$ α $\times$ 1000

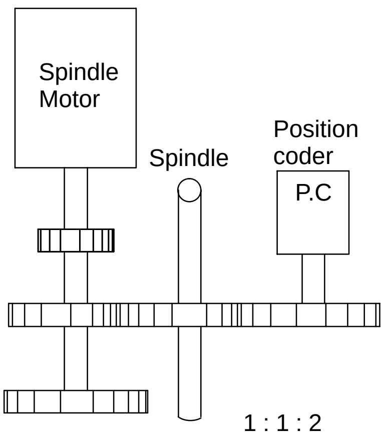
where;

E : Voltage in the velocity command at 1000 min^{-1}

L : Degrees of rotation by the spindle per one rotation of the spindle motor

α : Unit used for the detection

Examples



When the spindle motor, spindle, and position coder are connected as shown left, let the variables be as follows:

E = 1.667 (V) (A motor speed of 6000 min^{-1} corresponds to 10 V.)
L = 360° (One rotation of the spindle corresponds to one rotation of the spindle motor.)
α = La/4096
= 720°/4096
= 0.17578
La = 720° (= 360° $\times$ 2. One rotation of the position coder corresponds to two rotations of the spindle.)
4096 = The number of detected pulses per rotation of the position coder

Gear ratio between the spindle and the position coder
1:1 . . . . 0.08789 degrees
1:2 . . . . 0.17578 degrees
1:4 . . . . 0.35156 degrees
1:8 . . . . 0.70313 degrees

According to above ratio the loop gain multiplier is calculated as 2048 $\times$ 1.667/360 $\times$ 0.17578 $\times$ 1000 = 1667

* When the position coder which is built in a spindle motor sends 512 pulses per rotation, the unit used for the detection, α, is La/2048.

Fig.9.10.8 (b) Connection among the spindle motor, spindle, and position coder

5300	Tapping axis in–position width in rigid tapping
5301	Spindle in–position width in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

These parameters are used to set tapping axis and spindle in–position widths in rigid tapping.

> **NOTE**
> 1 If an excessively large value is specified, the threading precision will deteriorate.
> 2 When rigid tapping is performed using the second spindle
> - When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameter No.5300 and No.5301 are applied to the second spindle, as well as to the first spindle.
> - When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5302 and No.5303 are applied to the second spindle.

5302	Tapping axis in–position width in rigid tapping using the second spindle
5303	Spindle in–position width in rigid tapping using the second spindle

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

These parameters are used to set spindle and tapping axis in–position widths in rigid tapping using the second spindle.

> **NOTE**
> These parameters are enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5308	In–position width at point R in rigid tapping (tapping axis)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter is used to set the tapping axis in–position width at point R in rigid tapping.

5310	Positional deviation limit imposed during tapping axis movement in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit during tapping axis movement in rigid tapping. A value that falls outside the valid data range, described above, can be specified in parameter No.5314.

NOTE
1 When a high–resolution detector is used, the unit must be multiplied by 10.
2 When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5310 (or No.5314) is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5350 is applied to the second spindle.

5311	Limit value of spindle positioning deviation during movement in rigid tapping.

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a spindle positioning deviation during movement in rigidtapping.

Limit value = S $\times$ 360 $\times$ 100 $\times$ 1.5 / (60 $\times$ G $\times$ α)

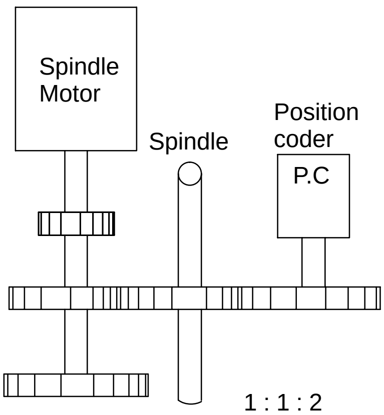
where

S : Maximum spindle speed in rigid tapping
(Setting value of parameter Nos. 5241 and greater)

G : Loop gain of rigid tapping axis
(Setting value of parameter Nos. 5280 and greater)

α : Detection unit

(Calculation example)



S = 3600
G = 3000
L = 360 degrees (One spindle rotation per spindle motor rotation)
α = La/4096
= 720 degrees/4096
= 0.17578 degrees
La = 720 degrees
(One position coder rotation requires two spindle rotations (= 360 degrees $\times$ 2)).
4096 = Detection pulse per position coder rotation

$$\text{Setting value} = \frac{3600 \times 360 \times 100 \times 1.5}{60 \times 3000 \times 0.17578}$$
$$= 6144$$

Fig.9.10.8 (c) Connection Among Spindle Motor, Spindle and Position Coder

NOTE

1 The detection unit is α = La/2048 when the position coder built–in spindle motor uses a position coder of 512 pulses per revolution.

2 When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5311 is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5351 is applied to the second spindle.

5312	Positional deviation limit imposed while the tapping axis is stopped in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit imposed while the tapping axis is stopped in rigid tapping.

NOTE

When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5312 is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5352 is applied to the second spindle.

5313	Positional deviation limit imposed while the spindle is stopped in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit imposed while the spindle is stopped in rigid tapping.

NOTE

When rigid tapping is performed using the second spindle
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5313 is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5353 is applied to the second spindle.

5314	Positional deviation limit imposed during tapping axis movement in rigid tapping

[Data type] 2–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Usually, parameter No.5310 is used to set a positional deviation limit imposed during tapping axis movement in rigid tapping. However, parameter No.5314 can be used to set a value greater than the valid data range of parameter No.5310 because of the resolution of the detector being used.

NOTE

1 When parameter No.5314 is set to 0, the setting of parameter No.5310 is used. When parameter No.5314 is set to a value other than 0, parameter No.5310 is disabled; in this case, the setting of parameter No.5314 is used.

2 When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the setting of parameter No.5314 (or No.5310) is applied to the second spindle, as well as to the first spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameter No.5350 is applied to the second spindle.

5321	Spindle backlash in rigid tapping (first–stage gear)
	Spindle backlash in rigid tapping
5322	Spindle backlash in rigid tapping (second–stage gear)
5323	Spindle backlash in rigid tapping (third–stage gear)
5324	Spindle backlash in rigid tapping (fourth–stage gear)
5325	Spindle backlash in rigid tapping using the second spindle (first–stage gear)
	Spindle backlash in rigid tapping using the second spindle
5326	Spindle backlash in rigid tapping using the second spindle (second–stage gear)

[Data type] Byte

[Unit of data] Detection unit

[Valid data range] 0 to 127

Each of these parameters is used to set a spindle backlash.

NOTE

When rigid tapping is performed using the second spindle

- When the SPR parameter (bit 1 of parameter No.5204) is set to 1, the settings of parameters No.5325 and No.5326 are applied to the second spindle.
- When the SPR parameter (bit 1 of parameter No.5204) is set to 0, the settings of parameter No.5321 is applied to the second spindle, as well as to the first spindle.

5335	Time constant for the spindle and tapping axis in second spindle extraction operation (first–stage gear)
5336	Time constant for the spindle and tapping axis in second spindle extraction operation (second–stage gear)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Each of these parameters is used to set a linear acceleration/deceleration time constant for the spindle and tapping axis in extraction operation during rigid tapping on a gear–by–gear basis.

NOTE

This parameter is enabled when both the TDR parameter (bit 2 of parameter No.5201) and the SPR parameter (bit 1 of parameter No.5204) are set to 1.

5341	Position control loop gain for the spindle and tapping axis in rigid tapping using the second spindle (common to all the gears)
5342	Position control loop gain for the spindle and tapping axis in rigid tapping using the second spindle (first–stage gear)
5343	Position control loop gain for the spindle and tapping axis in rigid tapping using the second spindle (second–stage gear)

NOTE

After these parameters have been set, the power must be turned off then back on for the settings to become effective.

[Data type] Word

[Unit of data] 0.01 s^{-1}

[Valid data range] 1 to 9999

Each of these parameters is used to set a position control loop gain for the spindle and tapping axis in rigid tapping using the second spindle.

NOTE
1 To use a varied loop gain on a gear–by–gear basis, set parameter No.5341 to 0, and set a loop gain for each gear in parameters No.5342 and No.5343.
2 This parameter is enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5350	Positional deviation limit imposed during tapping axis movement in rigid tapping using the second spindle

[Data type] 2–word

[Unit of data] Detection unit

[Valid data range] 1 to 99999999

This parameter sets a positional deviation limit imposed during tapping axis movement in rigid tapping using the second spindle.

NOTE
This parameter is enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5351	Positional deviation limit imposed during spindle movement in rigid tapping using the second spindle

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit imposed during spindle movement in rigid tapping using the second spindle.

NOTE
This parameter is enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5352	Positional deviation limit imposed while the tapping axis is stopped in rigid tapping using the second spindle

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit imposed while the tapping axis is stopped in rigid tapping using the second spindle.

NOTE
This parameter is enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5353	Positional deviation limit imposed while the spindle is stopped in rigid tapping using the second spindle

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter is used to set a positional deviation limit imposed while the spindle is stopped in rigid tapping using the second spindle.

> **NOTE**
> This parameter is enabled when the SPR parameter (bit 1 of parameter No.5204) is set to 1.

5381	Override value during rigid tapping return

[Data type] Byte

[Unit of data] 1% or 10%

[Valid data range] 0 to 200

This parameter is used to set the override value during rigid tapping return.
If the setting is 0, no override is applied.

> **NOTE**
> This parameter is valid when bit 4 (DOV) of parameter No. 5200 is set to 1.
> If bit 3 (OVU) of parameter No.5201 is set to 1, 10% is set as the units of data. Thus, an override of up to 2000% can be applied during extraction.

5382	Amount of return for rigid tapping return

[Data type] 2–word

[Unit of data] Input increments

[Valid data range] 0 to 99999999

During rigid tapping return, the tool can be pulled out, along the tapping axis, going beyond the stored rigid tapping start position by the amount specified with this parameter.

If the tool has already been retracted from rigid tapping, it will be retracted further only by the distance specified in this parameter.

9.10.9 Alarm and Message

Number	Message	Description
200	ILLEGAL S CODE COMMAND	In rigid tapping, an S value is out of the range or is not specified. The maximum value for S which can be specified in rigid tapping is set in parameter (No.5241 to 5243). Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In rigid tapping, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In rigid tapping, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In rigid tapping, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In rigid tapping, an axis movement is specified between the rigid M code (M29) block and G84 or G74 block for M series (G84 or G88 block for T series). Modify the program.
205	RIGID MODE DI SIGNAL OFF	1. Although a rigid M code (M29) is specified in rigid tapping, the rigid mode DI signal (DGN G061.0) is not ON during execution of the G84 (G88) block. 2. In a system with the multi–spindle, the spindle used for rigid tapping is not selected (by DI signal G27#0 and #1, or G61#4 and #5 (T series only). Check the PMC ladder diagram to find the reason why the DI signal is not turned on.
206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Correct the program.
207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–4 of rigid tapping axis) stops is larger than the set value. Note) Limit value must be set to parameter No.5312 for each axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–4 of rigid tapping axis) moves is larger than the set value. Note) Limit value must be set to parameter No. 5310 or 5314 for each axis.

Number	Message	Description
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–4 of rigid tapping axis) are beyond the range of -2^{31} to 2^{31}. This error usually occurs as the result of an improperly set parameters.
740	RIGID TAP ALARM; EXCESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping.
741	RIGID TAP ALARM; EXCESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping or synchronous error exceeded a set value (parameter No. 5214) during rigid tapping.
742	RIGID TAP ALARM; LSI OVER FLOW	LSI overflow has occurred on the spindle side during rigid tapping.

9.10.10 Notes

NOTES ON SPINDLES

CAUTION

1 When using an analog spindle, set the spindle speed offset value parameter (No. 3731) accurately. For the standard system, a value within –8191 to 8191 must be specified in this parameter. To perform rigid tapping, a value within –1023 to 1023 must be specified.
If the spindle speed offset is set inaccurately, the spindle is stopped and placed in in–position wait state when tapping is started.
In rigid tapping with a serial spindle, no setting is required for parameter No. 3731. Be sure to set 0.

2 When the threading and synchronous feed functions are enabled, the actual spindle speed during rigid tapping is indicated correctly. When an arbitrary gear ratio is used (by setting bit 1 (VGR) of parameter No. 5200 to 1), however, the actual spindle speed will not be indicated correctly in normal spindle mode.
When the T series is used, for example, information about the actual spindle speed is important for lathe machining. So, be particularly careful when using an arbitrary gear between the spindle and position coder.

NOTE

1 A spindle pitch error is not compensated for in rigid tapping mode. Drift compensation is not made with an analog spindle.

2 The maximum number of pulses that can be distributed to the spindle is:
- · 32,767 pulses per 8 msec for a serial spindle
- · 4,096 pulses per 8 msec for an analog spindle
  (This information is displayed by selecting No. 451 on the diagnosis screen.)

These values vary with the position coder gear ratio setting and rigid tapping specification. If a value greater than the maximum allowable number is specified, P/S alarm No. 202 is issued.

Notes on using functions such as the spindle positioning function at the same time

CAUTION

1 When the spindle orientation function is to be used at the same time
The spindle orientation function positions the spindle by using sensors and the PMC, without being directly controlled by the CNC.
The CNC has no direct control over this processing, instead following the specifications of the spindle orientation function being used.

2 When the spindle positioning function is to be used at the same time
When the spindle positioning function is to be used together with rigid tapping, rigid tapping mode must not be specified in spindle indexing mode, and spindle indexing mode must not be specified in rigid tapping mode. (Spindle positioning and rigid tapping cannot be performed simultaneously for a single spindle.)
This restriction does not apply, however, when multi–spindle control is applied; rigid tapping can be performed using the second spindle.
The spindle positioning function is effective for the first spindle only. This means that when spindle indexing is performed with the first spindle, rigid tapping can be specified with the second spindle.

CAUTION (continued)

3 When the Cs contouring control function for the serial spindle is used together with the rigid tapping function, the same motor is used for spindle rotation control, Cs contouring control, and rigid tapping modes. The following points must be noted:

(1) Whether to enter Cs contouring control mode or spindle rotation control mode is selected by the CON (Cs contouring control switch signal) signal; however, the system can enter rigid tapping mode regardless of the state of the CON signal. When the rigid tapping mode is canceled the system enters spindle rotation control mode or Cs contouring control mode according to the state of the CON signal.

(2) Since the system can change to rigid tapping mode directly from the Cs contouring control mode, use of the Cs contouring control function enables the tapping tool to be positioned before rigid tapping begins. Accurate positioning is not guaranteed. If the rigid tapping cycle executes gear change or output range changing, positioning is valid.

(3) Although the system can change to rigid tapping mode directly from Cs contouring control mode, positions designated in Cs contouring control mode are not preserved if rigid tapping mode is canceled by G80. When the system is changed to rigid tapping mode from Cs contouring control mode, then returns to the Cs contouring control mode, G00 or G28 must be issued to position the tapping tool.

(4) In systems with the serial spindle Cs contouring control function, the spindle motor is in a state called servo mode when it is operating in rigid tapping mode. In servo mode, it can accept jogging and manual handling feed. To prevent this, inhibit jogging and manual handling feed of the Cs contouring axis in the PMC logic during rigid tapping.

(5) The servo–off signal for the Cs contour control axis is valid also for the spindle during rigid tapping. It should be masked on the PMC side as required.

(6) When the multi–spindle control is also available and the rigid tapping is performed on the 2nd spindle, the rigid tapping can be specified to the 2nd spindle during the Cs contouring control of the 1st spindle.

Position control loop gain switching and serial spindle parameters

In rigid tapping, the loop gain of the tapping axis is switched so that the loop gains for position control of the tapping axis and spindle match each other.

This switching processing is specified by parameter Nos. 5280, and 5281 to 5284. The contents of the processing vary with whether the spindle is an analog or serial spindle, as described below.

· When the spindle is an analog spindle, the loop gains of the spindle and tapping axis are switched according to the values set in these parameters.

· When the spindle is a serial spindle, the loop gain of the tapping axis is switched according to the values set in these parameters. The loop gain of the spindle depends on the values set in the serial spindle parameters and applied gear signals (CTH2, CTH1).

Accordingly, to perform rigid tapping with a serial spindle, the loop gain for position control of the spindle must be set in the serial spindle parameters used for rigid tapping.

When multi–spindle control is being used, rigid tapping can also be performed for the second spindle. For the serial spindle used for rigid tapping, set the parameters indicated below.

The parameters indicated below are the major serial spindle parameters required for the setting and adjustment needed to use a serial spindle.

For details of the serial spindle parameters, refer to the “FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B–65280EN)”.

4044	Proportional gain of the velocity loop in servo mode (gear 1, gear 2)
4045	Proportional gain of the velocity loop in servo mode (gear 3, gear 4)

[Valid data range] 0 to 32767

Set a proportional gain for the velocity loop in a servo mode (such as rigid tapping mode).

4052	Integral gain of the velocity loop in the servo mode (gear 1, gear 2)
4053	Integral gain of the velocity loop in the servo mode(gear 3, gear 4)

[Valid data range] 0 to 32767

Set an integral gain of the velocity loop in a servo mode (such as rigid tapping mode).

4065	Position gain in the servo mode (HIGH)
4066	Position gain in the servo mode (MEDIUM HIGH)
4067	Position gain in the servo mode (MEDIUM LOW)
4068	Position gain in the servo mode (LOW)

[Unit of data] 0.01 sec^{-1}

[Valid data range] 0 to 65535

Set a servo loop gain in a servo mode (such as rigid tapping mode).

CAUTION

1 Set a loop gain for spindle position control in rigid tapping using a serial spindle. In these parameters, basically, set the same values as those set in parameter Nos. 5280 and 5281 to 5284 (loop gains for position control of the tapping axis).
Which serial spindle parameter (i.e., loop gain) is actually used to operate the spindle depends on the serial spindle clutch/gear selection signals CTH1 and CTH2 (G070#3, #2 for the first spindle, and G074#3, #2 for the second spindle). Accordingly, which parameter is to be used must be determined by considering the gear switching and PMC software.

The table below indicates the relationship between the spindle gear selection signals and selected gear numbers.

<table>
<tr><th>CTH1</th><th>CTH2</th><th>Gear selected</th><th colspan="3">Parameter No. to be used</th></tr>
<tr><td>0</td><td>0</td><td>HIGH</td><td>4065</td><td rowspan="2">4044</td><td rowspan="2">4052</td></tr>
<tr><td>0</td><td>1</td><td>MEDIUM HIGH</td><td>4066</td></tr>
<tr><td>1</td><td>0</td><td>MEDIUM LOW</td><td>4067</td><td rowspan="2">4045</td><td rowspan="2">4053</td></tr>
<tr><td>1</td><td>1</td><td>LOW</td><td>4068</td></tr>
</table>

9.10.11 Rigid–Tapping Bell–Shaped Acceleration/ Deceleration (M Series)

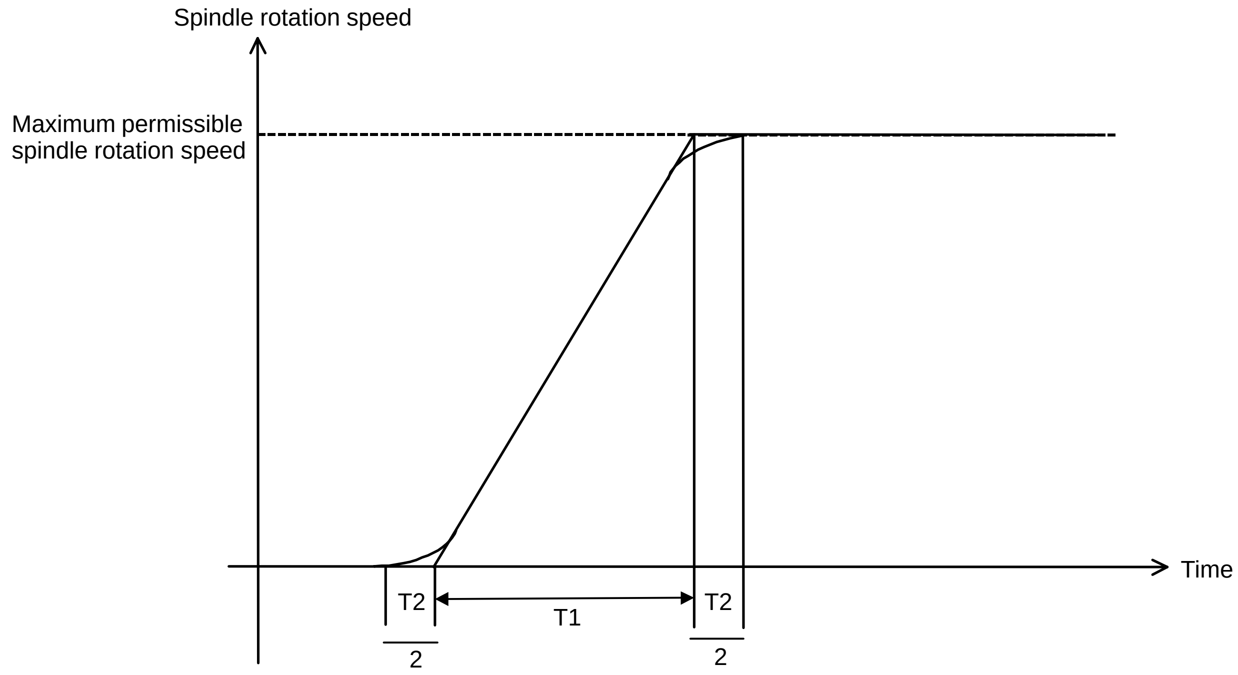
General

Linear acceleration/deceleration and exponential acceleration/ deceleration (M series only) have conventionally been applicable to rigid tapping. However, bell–shaped acceleration/deceleration was added recently. Using bell–shaped acceleration/deceleration enables a smaller time constant to be set for rigid tapping, thereby reducing the time required in acceleration/deceleration. Because this bell–shaped acceleration/deceleration is of rapid traverse bell–shaped acceleration/ deceleration type, parameters are used to specify a linear acceleration/ deceleration time constant and the time for a bell–shaped curved section.

The rigid tapping bell–shaped acceleration/deceleration function is an option.

About bell–shaped acceleration/deceleration

The time required in bell–shaped acceleration/deceleration for rigid tapping is the sum of the linear acceleration/deceleration time constant (value set in the conventional parameter) for the spindle and tapping axis and the time (value set in the new parameter) for a curved section.



T1 = linear acceleration/deceleration time constant for the spindle and tapping axis
T2 = time constant for a bell–shaped acceleration/deceleration curved section
T1 + T2 = time required in acceleration/deceleration

The actual linear acceleration/deceleration time constant, T1, for the spindle and tapping axis is determined according to the ratio of the maximum permissible spindle rotation speed to the actually specified value S. The bell–shaped acceleration/deceleration time constant for the curbed section is not proportional to the actual S command. Instead, the bell–shaped acceleration/deceleration is kept constant (value specified in the parameter).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5203			RGBEL					

[Data type] Bit

RGBEL Specifies the type of acceleration/deceleration to be applied to rigid tapping feed, as follows:

0 : Bell–shaped acceleration/deceleration

1 : Linear acceleration/deceleration

5261	Time constant for the first spindle and tapping axis (first gear stage)
5262	Time constant for the first spindle and tapping axis (second gear stage)
5263	Time constant for the first spindle and tapping axis (third gear stage)
5265	Time constant for the second spindle and tapping axis (first gear stage)
5266	Time constant for the second spindle and tapping axis (second gear stage)

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 4000

The spindle and tapping axis time constant for each gear stage is set up for rigid tapping.

The time needed for the spindle to reach its maximum permissible rotation speed is specified. The actual time constant is the ratio of the maximum permissible spindle speed to the S command.

If bell–shaped acceleration/deceleration is enabled, the time constant for the straight section is set up.

5271	Time constant for the first spindle and tapping axis for a pull–out operation (first gear stage)
5272	Time constant for the first spindle and tapping axis for a pull–out operation (second gear stage)
5273	Time constant for the first spindle and tapping axis for a pull–out operation (third gear stage)
5335	Time constant for the second spindle and tapping axis for a pull–out operation (first gear stage)
5336	Time constant for the second spindle and tapping axis for a pull–out operation (second gear stage)

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 4000

The spindle and tapping axis time constant for each gear stage is set up for a rigid tapping pull–out operation.

If bell–shaped acceleration/deceleration is valid, the time constant for the straight section is set up.

NOTE
These parameters are valid if the TDR parameter (bit 2 of parameter No. 5201) is 1.

5365	First–spindle bell–shaped acceleration/deceleration time constant for rigid tapping (first gear stage)
5366	First–spindle bell–shaped acceleration/deceleration time constant for rigid tapping (second gear stage)
5367	First–spindle bell–shaped acceleration/deceleration time constant for rigid tapping (third gear stage)
5369	Second–spindle bell–shaped acceleration/deceleration time constant for rigid tapping (first gear stage)
5370	Second–spindle bell–shaped acceleration/deceleration time constant for rigid tapping (second gear stage)

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 512

The time required in bell–shaped acceleration/deceleration for a curved section is set up for rigid tapping.

If these parameters are set to 0, linear acceleration/deceleration is applied.

NOTE

1 These parameters are valid if the RGDBEL parameter (bit 5 of parameter No. 5203) is 1.

2 When rigid tapping is performed on the second and third spindles,
- The settings of parameter Nos. 5365 and 5366 are applied to the second and third spindles as well as the first spindle if the SPR parameter (bit 1 of parameter No. 5204) is 0.
- The settings of parameter Nos. 5369 and 5370 and the settings of parameter Nos. 5373 and 5374 are applied, respectively, to the second and third spindles if the SPR parameter (bit 1 of parameter No. 5204) is 1.

Caution

(1) The linear acceleration/deceleration time constant parameter for rigid tapping specifies the time required for the spindle to reach its maximum permissible rotation speed. The actual time constant is obtained by calculating the ratio of the maximum permissible spindle rotation speed to the S command. However, the bell–shaped acceleration/deceleration time constant does not depend on that ratio. Instead, it is specified directly in a parameter.

(2) If the time constant for a pull–out operation is valid (the TDR parameter (bit 2 of parameter No. 5201) is 1), the bell–shaped acceleration/deceleration time constants specified in parameter Nos. 5365 to 5374 are used.

(3) If the bell–shaped acceleration/deceleration time constants (parameter Nos. 5365 to 5370) are 0, linear acceleration/deceleration is applied.

(4) If exponential acceleration/deceleration is selected for rigid tapping, bell–shaped acceleration/deceleration is disabled.

9.10.12 Reference Item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.13.8	RIGID TAPPING
CONNECTION MANUAL (This manual)		9.3	SPINDLE SPEED CON-TROL
		9.9	MULTI–SPINDLE
FANUC SERVO AMPLIFIER αi series DESCRIPTIONS (B–65282E)		11.4	RIGID TAPPING
FANUC AC SPINDLE MOTOR αi/βi series PARAMETER MANUAL (B–65280E)		2.3	RIGID TAPPING

9.11 SPINDLE SYNCHRONOUS CONTROL

General

This function enables the synchronous control of two spindles. It also enables the control of the rotation phase of a spindle, allowing non–standard workpieces as well as rods to be held by either of the two spindles.

Synchronous–spindle configuration

In spindle synchronous control, the spindle to which an S command is issued is called the master spindle. A spindle which ignores any S command that is issued for it, instead rotating synchronously with the master spindle, is called the slave spindle.

The table below shows the synchronous spindle configuration.

	Master spindle	Slave spindle
T series/M series	First serial spindle	Second serial spindle

Supplementary description

For details of synchronous–spindle connection, see the description of serial spindles.

The following description relates to this CNC.

- Synchronous control of spindle phase is executed when the signal for controlling the spindle phases in synchronization is entered in spindle synchronization control mode (after output of the signal indicating that the synchronous control of spindle speed has been completed). The signal indicating that the synchronous control of spindle phase is completed is output when the difference between the error pulses of the two spindles does not exceed the number of pulses specified in parameter No. 4810 of the NC function.
  The positions of spindle phase synchronization can be specified in spindle parameter No. 4034 on each of tool post 1 and tool post2.

  When the two spindles are subject to spindle–phase synchronous–control (until the spindle–phase synchronous–control completion signal, FSPPH <F044#3>, turns to "1"), they are not synchronized with each other.

  Do not specify spindle–phase synchronous control while the two spindles are holding a workpiece. Specifying this item causes phase synchronous control to start automatically.
- PMC signal, SYCAL <F044#4> is provided to monitor a synchronization errors between spindles for which spindle synchronization control or synchronous control of spindle phase is in effect. The synchronization error between the two spindles is always monitored. The SYCAL signal is set to 1 when the error (the absolute value of the error pulse) specified in parameter No. 4811 of tool post 1 is exceeded, and set to 0 when not exceeded.

- Constant surface speed control can be executed in synchronization control even while a workpiece is being held with the two spindles. However, if the speed is to change in excess of the specified time constant, the speed changes within the extent specified by time constant.
- The maximum speed in synchronization control is determined by the maximum speed of the spindle motor of master spindle (parameter No. 4020).

(Example) Maximum speed of the spindle motor of tool post 1: 6000 min^{-1}
Maximum speed of the spindle motor of tool post 2: 4500 min^{-1}

In the example above, a maximum spindle speed of 6,000 min^{-1} is specified for a spindle of tool post 1, although a spindle–speed command can specify up to 12 bits, 4096. If 6,000 min^{-1} is specified while synchronous control is specified, an overspeed alarm is issued for a spindle of tool post 2. Therefore, do not specify a value of more than 4,500 min^{-1} in this case.

- Like the conventional spindle speed (S) command for which 4 or 5 digits are issued for the first spindle, the signal for specifying spindle speed can be generated when spindle synchronization control or synchronous control of spindle phase are in the process of being put into effect. The SIND, SSIN SSGN, R01l to R12l, *SSTP, and SOR signals are effective as usual.

  However, in the usual mode of spindle rotation control, spindle speed can be controlled by the PMC function when the following conditions are satisfied: The SIND signal is set to 1 and the SSIN, SSGN, and R01l to R12l signals are provided. When spindle synchronization control is in the process of being put into effect, something other than the R01l to R12l signals is required to control the spindle speed in synchronization. The maximum spindle gear speed must be properly set in parameters No. 3741, 3742, 3743 and 3744. When the value set in the parameter corresponding to the selected gear is 0, the rotations of the spindles are not synchronized even if a command is entered in the R01I to R12I signals.
- The S command for the master spindle and the PMC control signal for spindle control become effective when issued before spindle synchronization control or synchronous control of spindle phase are put into effect. The S command issued in synchronization control becomes effective for the first spindle immediately after synchronization control is canceled.
- The load may change due to cutting (or threading). When the load changes in spindle synchronization control, the spindle speed may change and the signal indicating that the synchronous control of spindle speed is completed may go off temporarily.
- Parameters No. 4800 #0 (for the master spindle) and #1 (for the slave spindle) are used to set the direction of rotation of the first spindle and second spindle, respectively.
- The gear ratio of the spindle to the position coder must be set to one–to–one.

- In spindle synchronization control , the compensation value for spindle speed offset (parameter No. 3731) is disabled.
- A spindle–phase synchronous control command is effective only in synchronous spindle control mode. The specified phase can be repeatedly changed under synchronous control.

Signal

See the manual of serial spindles.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4800							ND2	ND1

[Data type] Bit type

ND1 In controlling the spindle synchronization, the direction of the first spindle (master spindle) motor rotation is:

0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

ND2 In controlling the spindle synchronization, the direction of the 2nd spindle (slave spindle) motor rotation is:

0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

4810	Error pulse between two spindles when synchronizing phases in the serial spindle synchronization control mode

[Data type] Byte type

[Unit of data] Pulse

[Valid data range] 0 to 255

Set the difference in error pulses between two spindles when synchronizing phases in the serial spindle synchronization control mode.

When the difference in error pulse between two spindles is within the value set in this parameter, the spindle phase synchronization completion signal FSPPH <F044#3> becomes “1”.

This parameter is used to check the difference in phase in synchronization control and to confirm the completion of synchronization in the serial spindle synchronization control mode.

4811	Allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode

[Data type] Word type

[Unit of data] Pulse

[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode.

This parameter is used to output the inter–spindle phase error detection signal SYCAL in the serial spindle synchronization control mode. The SYCAL <F044#4> signal becomes “1” when a phase error exceeding the value set in this parameter is found.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.

Note

NOTE

Signal SYCAL <F044#4> is used for monitoring a phase shift in synchronous control. The processing performed when a phase shift is detected depends on the specifications determined by the machine tool builder.

Reference item

FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B–65280EN)	2.5	Spindle synchronization control

9.12 SPINDLE ORIENTATION

General

This function stops the spindle at a specified position. The spindle can be stopped in either of the following two ways.

- The spindle is stopped using mechanical stop.
- The spindle is stopped by applying a function of the spindle control unit.

Mechanical stop

To mechanically stop the spindle by using, for example, a shot pin, rotate the spindle at a constant low speed and drive the pin into the spindle. The spindle can be rotated at a low constant speed by applying either of the following methods.

- Spindle orientation signal (See 9.3, "Spindle Control.")
- Spindle output control by the PMC (See 15.4.)

Using the spindle control unit

Some spindle control units can position the spindle motor by using sensors and position coders. The CNC itself does not control positioning by using these units.

Serial spindle orientation by a position coder

In serial spindle orientation by a position coder, the stop position is specified either by a parameter or by the PMC (spindle orientation function with the stop position externally set).

Signal

Spindle orientation signals with the stop position externally set

SHA00 to SHA11 for the first spindle <G078, G079>

SHB00 to SHB11 for the second spindle <G080, G081>

[Classification] Input signal

[Function] This command is used for specifying a stop position with an absolute position within one rotation in the following equation:

$$= \frac{360}{4096} \times \sum_{i=0}^{n} (2^i \times Pi)$$

where
Pi = 0 when SHAi = 0
Pi = 1 when SHAi = 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702					OR2	OR1		

[Data type] Bit

OR1 Whether the stop–position external–setting type orientation function is used by the first spindle motor

0 : Not used
1 : Used

OR2 Whether the stop–position external–setting type orientation function is used by the second spindle motor

0 : Not used
1 : Used

Caution

CAUTION

1 To perform spindle orientation by using the spindle control unit, the signals of the spindle control unit must be used. To perform serial spindle orientation by using a position coder (to perform serial spindle orientation with the stop position set externally), the serial spindle control unit signals must be used.
2 When the spindle orientation function of stop position external setting type is used, the stop position parameters in spindle orientation with a position coder (No. 4031 and 4204) are invalid.

Note

NOTE

Spindle orientation with the spindle positioning function differs from that described in this section. For details, see Section 9.7, “Spindle Positioning.”

Reference item

FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B–65280EN)		Position coder method spindle orientation

9.13 SPINDLE OUTPUT SWITCHING

General

Spindle output switching switches between the two motor windings, one for low speed and the other for high speed, incorporated into the special spindle motors. This ensures that the spindle motor demonstrates stable output characteristics over a wide range.

Since spindle output switching is a function of the spindle control unit, also refer to the manual for the spindle control unit being used.

This section describes the relationship between spindle output switching and the spindle control function in the CNC.

Operation of output–switchable spindle motor

To switch the spindle output characteristics, the windings are usually switched using a relay. Prior to the completion of winding switching, the spindle rotates free from drive power.

Output switching changes the relationship between a speed command, issued from the CNC to the spindle, and the output characteristics of the spindle motor. However, the relationship between the speed command and spindle motor speed is not changed.

Output switching timing

During actual machining, the spindle is usually controlled in the following way.

(1) Constant spindle speed during cutting, such as milling

(2) Continuously changing spindle speed during cutting, such as in constant surface speed control

(3) Controlling the position loop including the spindle motor during rigid tapping, spindle positioning, Cs contour control, etc.

For applications such as those in (1), we recommend switching the output characteristics for low speed and high speed by using the spindle motor speed detection signal of the spindle control unit.

For applications such as those described in (2) and (3), the spindle must not rotate with no drive power applied during cutting or positioning. It is necessary for the output characteristics to be switched appropriately before machining or for output switching to be masked by using a PMC ladder sequence.

Output switching and gear switching

Spindle output switching ensures that the spindle motor demonstrates stable characteristics over a wide range, and eliminates the requirement of a mechanical spindle gear switching mechanism.

In creating a PMC ladder sequence for output switching, however, using the gear switching of the CNC's spindle control function (see 9.3) may facilitate programming.

Note the following points when using gear switching for CNC spindle control for output switching with a machine tool having no mechanical gear switching mechanism.

- When gear selection output signals, GR2O and GR1O <F034 #0, #1>, are used (for machining centers in which constant surface speed control is not provided and GTT, bit 4 of parameter No. 3706, is set to 0)

  Set two gears, which are almost the same.

  (Example: Value of parameter No. 3741 = value of parameter No. 3742 – 1, value of No. 3742 = Maximum spindle speed)

  When parameter No. 3741 is equal to parameter No. 3742, the CNC judges that one gear is used, and does not output the GR2O signal.
- The parameters related to gear switching points, SGT, bit 3 of parameter No. 3705, and SGB, bit 2 of parameter No. 3761, parameter Nos. 3761 and 3751 can be used.
- In usual spindle control, depending on the speed at switching points, the speed specified by the spindle speed command may differ slightly from the actual speed in the area where the maximum spindle speed is set to the maximum speed ±1. (This is because the spindle motor speed, specified by the speed command, is calculated based on the settings of parameter Nos. 3741 to 3744.)

  This does not apply to rigid tapping. (Because the machine tool is controlled using the feedback signal from the detector in the position loop.)
- When gear selection input signals, GR1 and GR2 <G028 #1, #2>, are used (for lathes or machining centers in which constant surface speed control is provided or GTT, bit 4 of parameter No. 3706, is set to 1)

  Parameter settings are read according to the input signal information. Unlike the GR2O and GR1O signals, these signals do not require special parameter settings.

Example) When parameter Nos. 3741 and 3742 are set to the maximum spindle speed.

Create a PMC sequence that specifies the following.
For gear 1, set GR1 and GR2 to 0.
For gear 2, set GR2 to 0 and set GR1 to 1.

The PMC must determine the switching timing on the basis of some information.

Reference item

CONNECTION MANUAL (This manual)	9.3 9.10	Spindle control Rigid tapping
FANUC AC SPINDLE MOTOR αi series PARAMETER MANUAL (B–65280EN)	5.1	Output switching control

10 TOOL FUNCTIONS

10.1 TOOL FUNCTION

General

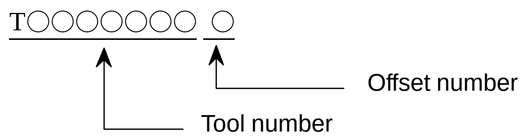
- **M series**

Selection of tools can be done by commanding tool numbers with up to an 8-digit numeral after address T.

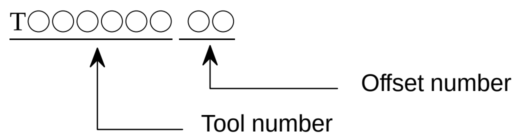
- **T series**

Selection of tools and offset amounts can be done by commanding tool numbers and offset numbers with up to an 8-digit numeral after address T. The offset number is specified with the last one or two digits of the T code. The tool number is specified with the remaining digits after excluding the one or two digits used to specify the offset number.

When the last one digit is used to specify the offset number:
(Parameter LD1 (No. 5002#0)=1)



When the last two digits are used to specify the offset number:
(Parameter LD1 (No. 5002#0)=0)



When a T code is specified, the code signal and strobe signal corresponding to the specified tool number are issued. The machine selects a tool according to the issued signals. The code signal is held until another T code is specified.

In a block, no more than one T code can be specified. The maximum number of digits that can follow T can be specified in parameter 3032. If this number is exceeded, an alarm occurs.

Signal

See Section 8.1.

Parameter

3032	Allowable number of digits for the T code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the T code.

	#7	#6	#5	#4	#3	#2	#1	#0
5002							LGN	LD1

[Data type] Bit

LD1 Wear offset number of tool offset

0 : Specified using the lower two digits of a T code
1 : Specified using the lower one digit of a T code

LGN Geometry offset number of tool offset

0 : Is the same as wear offset number
1 : Specifies the geometry offset number by the tool selection number

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	

[Data type] Bit

TGC When a T code is specified in a block containing G50, G04, or G10:

0 : No alarm occurs.
1 : P/S alarm No. 254 occurs.

Alarm and message

Number	Message	Description
030	ILLEGAL OFFSET NUMBER (T series)	The offset number in T function specified for tool offset is too large. Modify the program.
043	ILLEGAL T–CODE COMMAND (M series)	In a system using the DRILL–MATE with an ATC, a T code was not specified together with the M06 code in a block. Alternatively, the T code was out of range.
245	T–CODE NOT ALOWED IN THIS BLOCK (T series)	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.

Note

NOTE
When a move command and a tool function are specified in the same block, the commands are executed in one of the following two ways:
(i) Simultaneous execution of the move command and tool function commands.
(ii) Executing tool function commands upon completion of move command execution.
The selection of either (i) or (ii) depends on the sequence program of PMC.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.10.1	TOOL SELECTION FUNCTION
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.10.1	TOOL SELECTION FUNCTION
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.10.1	TOOL SELECTION FUNCTION
	OPERATOR’S MANUAL (T series) (B–64134EN)	II.10.1	TOOL SELECTION FUNCTION
CONNECTION MANUAL (This manual)		8	AUXILIARY FUNCTION

10.2 TOOL COMPENSATION VALUE/ TOOL COMPENSATION NUMBER/ TOOL COMPENSATION MEMORY

General (M series)

Tool compensation values include tool geometry compensation values and tool wear compensation values (Fig. 10.2 (a)).
The geometry compensation and wear compensation can be combined into to the tool compensation.

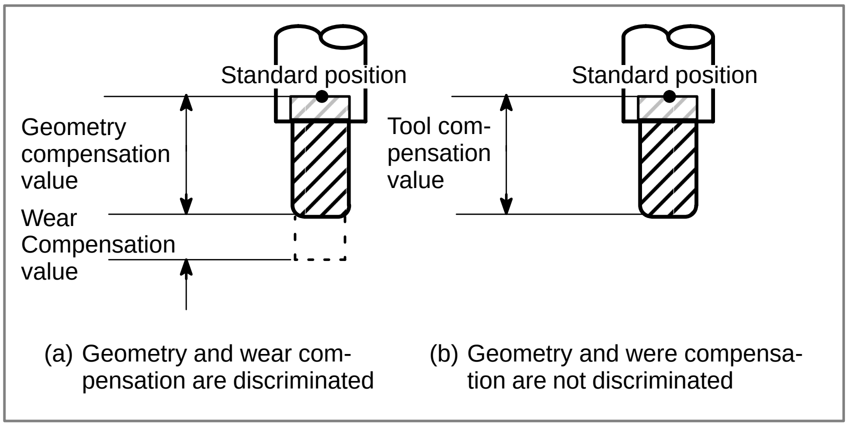


Fig. 10.2 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.

- **Range of tool compensation value**

Tool offset amount range which can be set is as follows:

Increment system	Tool compensation (Geometry compensation)		Tool wear compensation	
	Metric input	Inch input	Metric input	Inch input
IS-B	±999.999mm	±99.9999inch	±99.999mm	±9.9999 inch
IS-C	±999.9999mm	±99.99999inch	±99.9999mm	±9.99999inch

- **Tool compensation number**

The memory can hold 400 sets of tool compensation values.

- **Tool compensation memory**

(1) Tool compensation memory C

Memory for geometry compensation and wear compensation is separate in tool compensation memory C. Geometry compensation and wear compensation can thus be set separately. Separate memories are prepared for cutter compensation (for D code) and for tool length compensation (for H code).

The above description is summarized as follows:

Tool compensation memory	Compensation amount
C	Geometry compensation value for H code
	Geometry compensation value for D code
	Wear compensation value for H code
	Wear compensation value for D code

General (T series)

Tool compensation values include tool geometry compensation values and tool wear compensation values (Fig. 10.2 (b)).
Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear (Fig. 10.2 (c)).

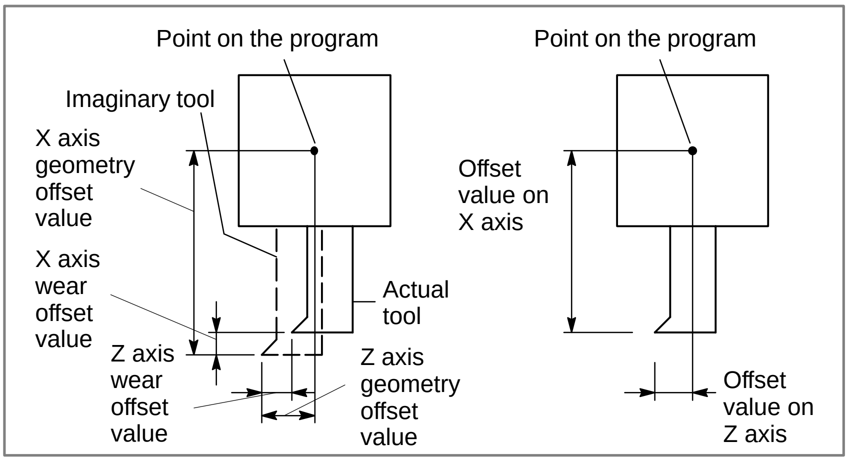


Fig. 10.2 (b) Difference the tool geometry offset from tool wear offset

Fig.10.2 (c) Not difference the tool geometry offset from tool wear offset

Tool compensation values can be entered into CNC memory from the MDI unit or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address T in a program.
The value is used for tool offset or tool nose radius compensation.

- **Range of tool compensation value**

Tool offset amount range which can be set is as follows:

Increment system	Tool compensation (geometry compensation, wear compensation)	
	Metric input	Inch input
IS-B	±999.999 mm	±99.9999 inch
IS-C	±999.9999 mm	±99.99999 inch

- **Tool compensation number**

The memory can hold 64 sets of tool compensation values.

- **Tool compensation memory**

Memory for geometry compensation and wear compensation is prepared separately. Geometry compensation and wear compensation can thus be set separately.

The above description is summarized as follows:

Compensation amount
Geometry compensation
Wear compensation

- **Disabling soft key [ERASE] on the offset screen**

A parameter can be set to turn off the display of soft key [ERASE] on the offset screen and disable the soft key's function to erase an offset.

Signal

Tool offset direction signal G2RVY <G090#2>, G2RVZ <G090#1>, G2RVX <G090#0>

[Classification] Input signal

[Function] Changes the direction of a compensation amount when the tool offset is used.

[Operation] When an axis–specific tool offset value (wear tool offset value + figure tool offset value) is applied, that value is used without being modified if the signal is 0. If the signal is 1, the tool offset is applied in a reverse direction.

NOTE
G2RVY is valid when using the Y–axis offset option.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G090						G2RVY	G2RVZ	G2RVX

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3109							DWT	

[Data type] Bit

DWT Characters G and W in the display of tool wear/geometry compensation amount

0 : The characters are displayed at the left of each number.
1 : The characters are not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3205				OSC				

[Data type] Bit

OSC On the offset screen, offset value erasure by a soft key is:
0 : Enabled.
1 : Disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Data type] Bit

WOF Setting the tool wear compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

GOF Setting the tool geometry compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Data type] Word

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No. 3290 and bit 1 (GOF) of parameter No. 3290, parameter Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values.

Example:

The following setting disables the modification of both the tool geometry compensation values and tool wear compensation values corresponding to offset numbers 100 to 110:

Bit 1 (GOF) of parameter No. 3290 = 1 (Disables tool geometry compensation value modification.)

Bit 0 (WOF) of parameter No. 3290 = 1 (Disables tool wear compensation value modification.)

Parameter No. 3294 = 100

Parameter No. 3295 = 11

If bit 0 (WOF) of parameter No. 3290 is set to 0, the modification of the tool geometry compensation values alone is disabled. The tool wear compensation values may be modified.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP						LGN	

[Data type] Bit

LGN Geometry offset number of tool offset

0 : Is the same as wear offset number
1 : Specifies the geometry offset number by the tool selection number

WNP Imaginary tool tip direction used for tool nose radius compensation is the direction specified by:

0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

[Data type] Bit

ORC Tool offset value

0 : Set by the diameter specification (Can be set in only the axis under diameter programming)
1 : Set by the radius specification

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:

0 : Not performed
1 : Performed

5013	Maximum value of tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation. The following alarm or warning will be informed when the tool wear compensation (absolute value) exceeding this setting value is set.

Input from MDI	Too many digits
Input by G10	P/S 32 offset value is out of range by G10

5014	Maximum value of incremental input for tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation at an incremental input. If the incremental value exceeds the set value, the following alarm or warning message is indicated:

Input from MDI	Data is out of range
Input by G10	P/S 32 offset value is out of range by G10

Alarm and message

Number	Message	Description
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
5300	SET ALL OFFSET DATAS AGAIN	After the inch/metric automatic conversion function (OIM: Bit 0 of parameter No. 5006) for tool offset data is enabled or disabled, all the tool offset data must be reset. This message reminds the operator to reset the data. If this alarm is issued, reset all the tool offset data. Operating the machine without resetting the data will result in a malfunction.

Warning message	Content
DATA IS OUT OF RANGE	The value searched exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.

10.3 TOOL LIFE MANAGEMENT

10.3.1 Tool life management

General

When tools are classified into several groups, average tool life (No. of uses or time) is designated for each group. Whenever a tool is used, the usage time is subtracted from the tool life; when the tool life expires, the next tool in the group is selected. The tool sequence within a group is arranged in advance.

Signal

The end of a tool's life is reported by tool change signal TLCH or individual tool change signal TLCHI. Tool change signal TLCH is set to 1 at the end of the life of the last tool of a group. Individual tool change signal TLCHI is set to 1 at the end of the life of the current tool.

Tool change signal TLCH <F064#0>

[Classification] Output signal

[Function] Reports the end of the life of the last tool of a group.

[Output condition] The signal is set to 1 when:

· The life of the last tool of a group ends, after tool change has been performed each time the end of the life of each tool in a group is detected.

The signal is set to 0 when:

· Tool–change reset is completed for all groups in which no available tools remain.

NOTE
The TLCH signal turns to “1” when the CNC is reset by M02 or M30, for instance after the tool life, based on the frequency of times used, is reached. When tool life is specified by usage time, TLCH turns to “1” when the tool life limit is reached. The signal will change during machine operation, but machining will continue until the end of the program.

Tool change reset signal TLRST <G048#7>

[Classification] Input signal

[Function] Clears all executable data, including the life count of the group, *, and @.

To clear the data, specify a group number by tool group number selection signal after replacing the worn–out tools that are displayed on the CRT. The data can also be cleared from the MDI.

[Operation] When the signal is set to 1, the control unit operates as follows:

· Clears all executable data, including the life count of the group.

If the same group is specified after machining is resumed, the first tool in the group is selected.

> **NOTE**
> Tool change reset signal TLRST is valid only when the automatic operating signal OP is “0”.

Individual tool change signal TLCHI <F064#2> (M series)

[Classification] Output signal

[Function] Reports the end of the life of the current tool. The following processing can be programmed: A running program is interrupted by a tool-change program when the signal turns to “1”. Execution of the interrupted program is resumed when the tool is changed.

[Output condition] The signal is set to “1 ”when:

· The end of the life of the current tool is detected.

The signal is set to “0” when:

· Individual tool-change reset is executed.

Individual tool change reset signal TLRSTI <G048#6> (M series)

[Classification] Input signal

[Function] Sets the individual tool change signal TLCHI to “0”.

[Operation] When the signal is set to “1”, the control unit operates as follows:

· Sets the individual tool change signal to “0”.

> **NOTE**
> 1 These signals are valid only when tool life management is performed on the basis of the tool life calculated in terms of time or cutting length.
> 2 Individual tool change signal TLCHI is not cleared by reset.

Tool skip signal TLSKP <G048#5>

[Classification] Input signal

[Function] A tool which has not reached its lifespan may be changed by one of two methods:

(i) Designate the group number for the tool by tool group number selection signal then turn the tool skip signal TLSKP to "1". The next T-code command will pass over the current tool in the group for which the skip was designated, and select the next tool.

(ii) Turn the TLSKP signal to "1" without designating a group number, and the machine will skip to the next tool in the group currently in use.

Either of these methods is set using parameter SIG (No. 6800#3). Tool life is counted from zero. When the TLSKP signal is "1" and the last tool in the group is being used, the TLCH signal turns to "1".

[Operation] When the signal is set to "1", the control unit operates as follows:

· Selects the next tool in the group for which a skip is specified with the next T code.

· Assumes the number of the group to which the current tool belongs.

CAUTION
The cycle start lamp signal (STL) and feed hold lamp signal (SPL) must both be "0" before inputting the TLSKP signal.

New tool select signal TLNW <F064#1>

[Classification] Output signal

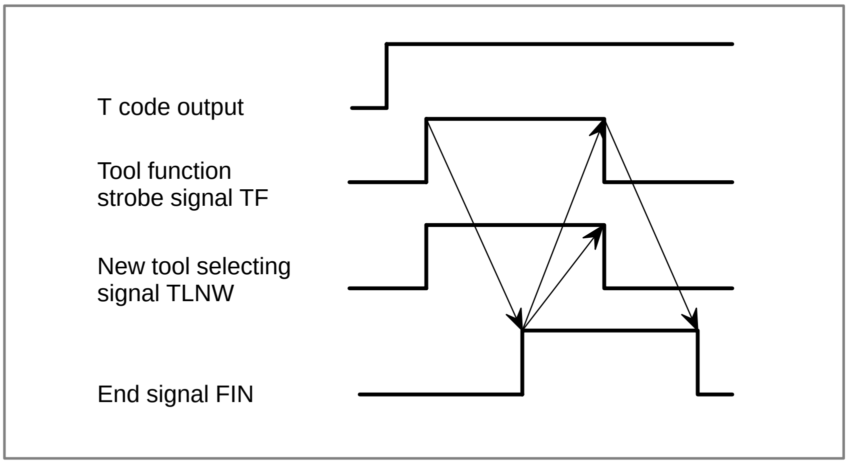
[Function] Reports that a new tool of a certain group is selected.
This signal can be used when, for example, a compensation value is to be measured automatically when a new tool is selected.
The new tool select signal is issued at the same timing as TF (tool function strobe signal).

[Output condition] The signal is set to "1" when:

· A new tool of a certain group is selected.

The signal is set to "0" when:

· The completion signal is set to "1".



Tool group number select signal TL01 to TL256 (M series) <G047#0 to G048#0> TL01 to TL64 <G47#0 to #6> (T series)

[Classification] Input signal

[Function] When the TLRST or TLSKP signals are input, the tool group number must be given in advance, using the tool group number selection signals TL01 to TL64 (T series) or TL01 to TL128 (M series).
Command the following value in binary form:
Tool group number to be specified –1

[Operation] A specified tool group is selected.

Tool life count override signal *TLV0 to *TLV9 <G049#0 to G050#1> (M series)

[Classification] Input signal

[Function] Overrides the life count (time) if parameter LFV (No. 6801#2) is specified.

Each of the ten binary code signals has a unique override value that becomes valid when the signal is set to “0”. The life count is overridden by the sum of the valid override values. The override value can be specified in steps of 0.1, within the range of 0 to 99.9.

$$\text{Override value} = \sum_{i=0}^{9} \{2^i \times Vi\}$$

*TLV0	× 0.1
*TLV1	× 0.2
*TLV2	× 0.4
*TLV3	× 0.8
*TLV4	× 1.6
*TLV5	× 3.2
*TLV6	× 6.4
*TLV7	×12.8
*TLV8	×25.6
*TLV9	×51.2

(Example) When *TLV7, *TLV6, and *TLV3 are set to “0”, the override value is calculated as follows:

12.8 + 6.4 + 0.8 = 20.0

The life count is multiplied by 20.0.

[Operation] The actual cutting time is counted and multiplied by the override value obtained by the signals. The calculated time is used as the basis for tool-life management.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0
G050							*TLV9	*TLV8

	#7	#6	#5	#4	#3	#2	#1	#0
F064						TLCHI	TLNW	TLCH

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6800			SNG	GRS	SIG	LTM	GS2	GS1
	M6T	IGI	SNG	GRS	SIG	LTM	GS2	GS1

[Data type] Bit

GS1, GS2 This parameter sets the combination of the number of tool life groups which can be entered, and the number of tools which can be entered per group as shown in the table below.

GS2	GS1	M series		T series	
		Group count	Tool count	Group count	Tool count
0	0	1– 16	1– 16	1– 16	1– 16
0	1	1– 32	1– 8	1– 32	1– 8
1	0	1– 64	1– 4	1– 64	1– 4
1	1	1–128	1– 2	1– 16	1– 16

LTM Tool life

0 : Specified by the number of times
1 : Specified by time

SIG Group number is

0 : Not input using the tool group number selection signal during tool skip (The current group is specified.)
1 : Input using the tool group signal during tool skip

GRS Tool change reset signal

0 : Clears only the execution data of a specified group
1 : Clears the execution data of all entered groups

SNG Input of the tool skip signal when a tool that is not considered tool life management is selected.

0 : Skips the tool of the group used last or of the specified group (using SIG, #3 of parameter No. 6800).
1 : Ignores a tool skip signal

IGI Tool back number

0 : Not ignored
1 : Ignored

M6T T code in the same block as M06

0 : Judged as a back number
1 : Judged as a next tool group command

	#7	#6	#5	#4	#3	#2	#1	#0
6801		EXG					TSM	
	M6E	EXT			EMD	LFV		

[Data type] Bit

TSM When a tool takes several tool numbers, life is counted in tool life management:

0 : For each of the same tool numbers.
1 : For each tool.

LFV Specifies whether life count override is enabled or disabled when the extended tool life management function is used.

0 : Disabled
1 : Enabled

EMD An asterisk (*) indicating that a tool has been expired is displayed,

0 : When the next tool is selected
1 : When the tool life is expired

EXG Tool life management data registration by G10 (T series) is:

0 : Performed after the data for all tool groups has been cleared.
1 : Performed by adding/changing or deleting the data for a specified group.

EXT Specifies whether the extended tool life management function is used.

0 : Not used
1 : Used

M6E When a T code is specified in the same block as M06

0 : The T code is processed as a return number or as a group number selected next. Either is set by parameter M6T No. 6800#7.
1 : The tool group life is counted immediately.

	#7	#6	#5	#4	#3	#2	#1	#0
6803							LFE	LGR
							LFE	

NOTE
After this parameter has been set, the power must be turned off then on again for the setting to become effective.

[Data type] Bit

LGR When the tool life management function is used, a tool life type is:

0 : Chosen based on the LTM parameter (bit 2 of parameter No.6800) for all groups.
1 : Set to either count or duration on a group–by–group basis.

When LGR is set to 1, the specification of address Q is added to the G10 (tool life management data setting) command format. As shown in the example below, specify the tool life of each group as either a count or a duration. If address Q is omitted for a group, the specification of the LTM parameter (bit 2 of parameter No.6800) applies to the group.

Example: When the LTM parameter (bit 2 of parameter No.6800) is set to 0

```
G10 L3 ;
P1 L10 Q1 ; (Q1: The life of group 1 is specified as a count.)
    :
P2 L20 Q2 ; (Q2: The life of group 2 is specified as a duration.)
    :
P3 L20 ;
    (Omission of Q: The life of group 3 is specified as a count.)
    :
G11 ;
M30 ;
%
```

LFE When a tool life is specified by count:
0 : A count value from 0 to 9999 can be specified.
1 : A count value from 0 to 65535 can be specified.

6810	
	Tool life management ignored number

[Data type] Word

[Valid data range] 0 to 9999

This parameter sets the tool life management ignored number.

When the set value is subtracted from a T code, a remainder is used as the tool group number of tool life management when a value exceeding the set value is specified in the T code.

6811	Tool life count restart M code

[Data type] Byte

[Valid data range] 0 to 255 (not including 01, 02, 30, 98, and 99)
When zero is specified, it is ignored.

When the life is specified by the number of times, the tool exchange signal is output when a tool life count restart M code is specified if tool life of at least one tool group is expired. A tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified. A tool life counter is then incremented by one.

When the life is specified by time, a tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified.

Alarm and message

Number	Message	Description
149	FORMAT ERROR IN G10L3	A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T–CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP (M series)	When the group is not commanded, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T–CODE IN M06 (M series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
	ILLEGAL T–CODE IN M06 (T series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.10.2	Tool Life Management Function
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.10.2	Tool Life Management Function
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.10.2	Tool Life Management Function
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.10.2	Tool Life Management Function

10.3.2 Tool Life Arrival Notice Signal (M Series)

General

Once the remaining life of tools (to be kept used until new tools are selected) is set up for each group, it becomes possible to check the remaining tool life for the groups. When the actual remaining life of tools becomes shorter than the remaining tool life setting, a signal is output to the PMC.

Signal description

The tool life arrival notice signal TLCHB is used to notify that the actual remaining tool life is shorter than the corresponding remaining tool life setting. This signal is output when ”tool life setting (LIFE) – tool life count (COUNT)” reaches the remaining life setting for the group.

Tool life arrival notice signal TLCHB <F064#3>

[Classification] Output signal

[Function] Notifies that the life of the last tool in the group has expired.

[Output condition] This signal is 1 in the following cases:

- The actual remaining life ”LIFE – COUNT” is equal to the value set in a parameter → ”equal” type
- The actual remaining life ”LIFE – COUNT” is less than the value set in a parameter → ”less–than” type

This signal is 0 in the following cases:

- The actual remaining life ”LIFE – COUNT” is greater than the value set in a parameter → ”equal” type
- The actual remaining life ”LIFE – COUNT” is equal to the value set in a parameter → ”less–than” type

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F064						TLCHB		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6802	RMT							

[Data type] Bit

RMT Specifies when to turn off the tool life arrival signal TLCHB, as follows:

0 : The actual remaining life is longer than that specified in a parameter (”less than” type).

1 : The actual remaining life is not equal to that specified in a parameter (”equal” type).

10.4 CUTTER COMPENSATION

10.4.1 Cutter Compensation C (M Series)

General

When the tool is moved, the tool path can be shifted by the radius of the tool.

To make an offset as large as the radius of the tool, first create an offset vector with a length equal to the radius of the tool (start–up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head points to the center of the tool.

If a linear interpolation, corner offset, or circular interpolation command is specified after start–up, the tool path can be shifted by the length of the offset vector during machining.

To return the tool to the start point at the end of machining, cancel the cutter compensation mode.

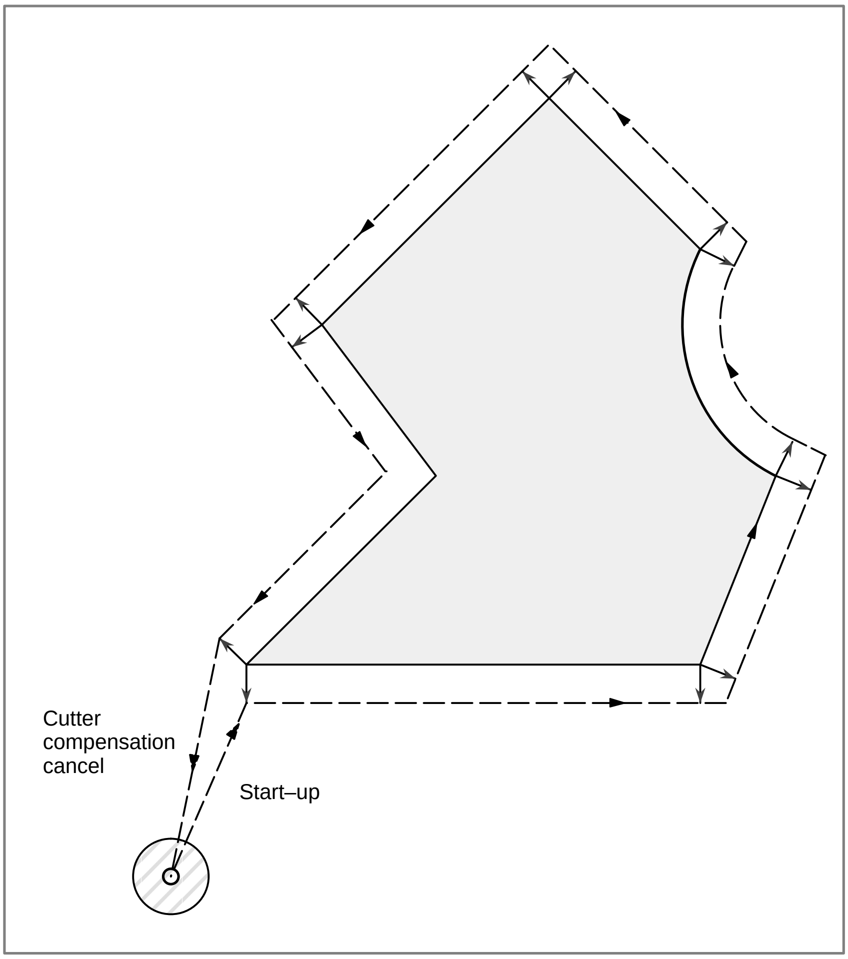


Fig. 10.4.1 Outline of cutter compensation C

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5001						OFH		

[Data type] Bit

OFH Offset number of tool length compensation, cutter compensation and tool offset

0 : Specifies the tool length compensation using an H code, and cutter compensation C using a D code
Tool offset conforms to TPH in parameter TPH (No. 5001#5).

1 : Specifies the tool length compensation, cutter compensation and tool offset using H codes

	#7	#6	#5	#4	#3	#2	#1	#0
5003						CCN	SUV	SUP

[Data type] Bit

SUP Start up or cancel in cutter compensation C

0 : Type A
1 : Type B

SUV When G40, G41, and G42 are specified independently,

0 : The start up and cancel operation conforms to the standard specification.
1 : Moves by a distance corresponding to the offset vector which is vertical to the next block movement.

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.
1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

	#7	#6	#5	#4	#3	#2	#1	#0
5008		GCS	QCR	MCR	CNV	G39	CNC	CNI

[Data type] Bit

CNI Interference check for cutter compensation C (M series) or tool–tip radius compensation (T series) is:

0 : Performed
1 : Not performed

CNC During interference check for cutter compensation C (M series) or tool–tip radius compensation (T series), when the direction of movement after application of the offset differs from the programmed direction by between 90° and 270°:

0 : An alarm is issued.
1 : No alarm is issued.

G39 The corner rounding function (G39) in cutter compensation C mode is:
0 : Disabled.
1 : Enabled.

CNV The interference check and vector erasure of cutter compensation C (M series) or tool–nose radius compensation (T series) are:
0 : Performed.
1 : Not performed.

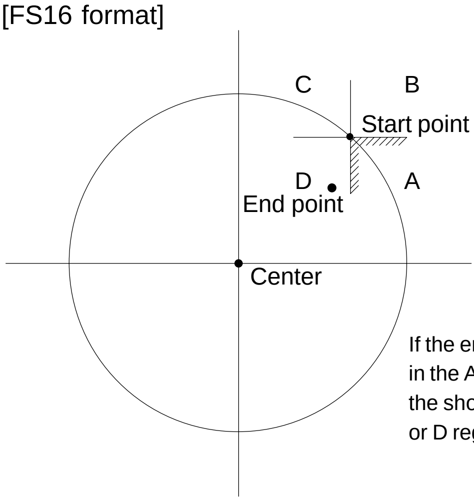
MCR If G41/G42 (cutter compensation C (M series) or tool–nose radius compensation (T series)) is specified in the MDI mode, an alarm is:
0 : Not raised.
1 : Raised. (P/S5257)

NOTE
In the MDI mode, cutter compensation C (M series) or tool–nose radius compensation (T series) is not performed, irrespective of the setting of this parameter.

QCR The travel distance of circular interpolation in cutter compensation C (M series) or tool–nose radius compensation (T series) is judged:
0 : In the FS16 format.
1 : In the FS15 format.

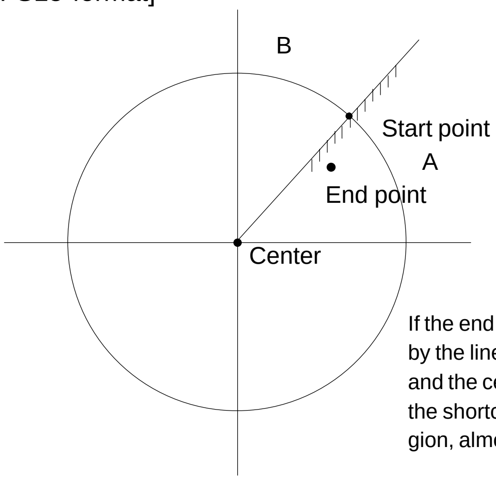
FS16 and FS15 determine the travel distance in different ways if the radius of arc at the start point of circular interpolation is different from that at the end point (if the end point is not on the arc). By this parameter, the method of determining the travel distance of circular interpolation can be selected.

[FS16 format]



If the end point viewed from the start point is in the A region, the movement is made along the shortcut. If the end point is in the B, C, or D region, almost a single turn is made.

[FS15 format]



If the end point is in the A region separated by the line L drawn between the start point and the center, the movement is made along the shortcut. If the end point is in the B region, almost a single turn is made.

NOTE
The setting of this parameter determines the travel distance determination method for circular interpolation not during cutter compensation C (M series) or tool–nose radius compensation (T series) as well. Accordingly, if this parameter is set, the setting of bit 3 (CQD) of parameter No. 3450 is invalid.

GCS If G49 (G code for canceling tool length compensation) and G40 (G code for canceling cutter compensation) are specified in a single block, the tool length compensation is cancelled:
0 : In the next block.
1 : In the specified block.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during cutter compensation C

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of the corner during cutter compensation C (M series).

Alarm and message

Number	Message	Description
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for cutter compensation C. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.
035	CAN NOT COMMANDED G39	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
036	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
041	INTERFERENCE IN CRC	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
5257	G41/G42 NOT ALLOWED IN MDI MODE	G41/G42 (cutter compensation C: M series, tool–nose radius compensation: T series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.14.4, 14.5	Cutter compensation C
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.14.4, 14.5	Cutter compensation C

10.4.2 Tool Nose Radius Compensation (T Series)

General

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

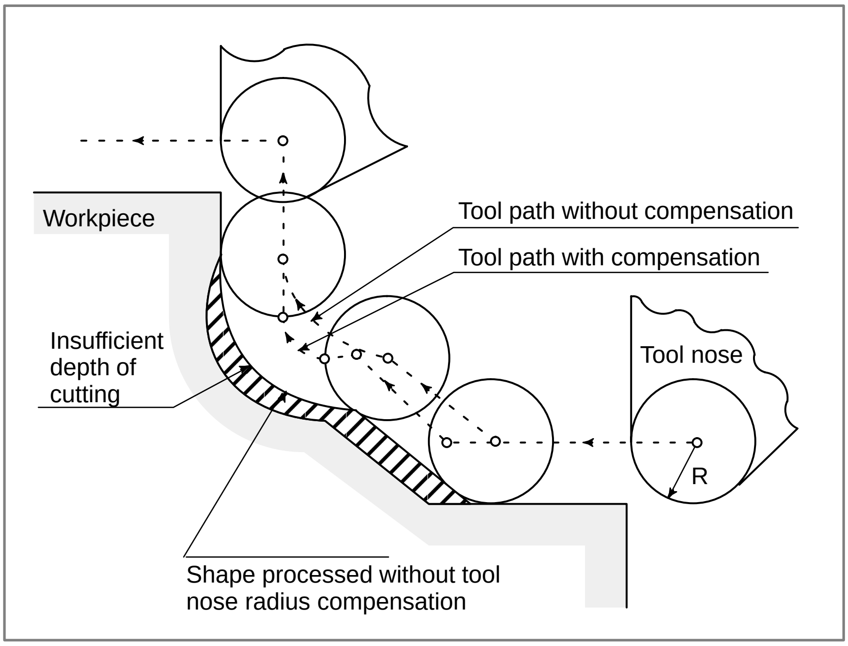


Fig. 10.4.2 Tool path of tool nose radius compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP							

[Data type] Bit

WNP Imaginary tool tip direction used for tool nose radius compensation is the direction specified by:

0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5003						CCN		

[Data type] Bit

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.

1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

	#7	#6	#5	#4	#3	#2	#1	#0
5008			QCR	MCR	CNV		CNC	CNI

[Data type] Bit

CNI Interference check for cutter compensation C (M series) or tool–tip radius compensation (T series) is:

0 : Performed

1 : Not performed

CNC During interference check for cutter compensation C (M series) or tool–tip radius compensation (T series), when the direction of movement after application of the offset differs from the programmed direction by between 90° and 270°:

0 : An alarm is issued.

1 : No alarm is issued.

CNV The interference check and vector erasure of cutter compensation C (M series) or tool–nose radius compensation (T series) are:

0 : Performed.

1 : Not performed.

MCR If G41/G42 (cutter compensation C (M series) or tool–nose radius compensation (T series)) is specified in the MDI mode, an alarm is:

0 : Not raised.

1 : Raised. (P/S5257)

NOTE

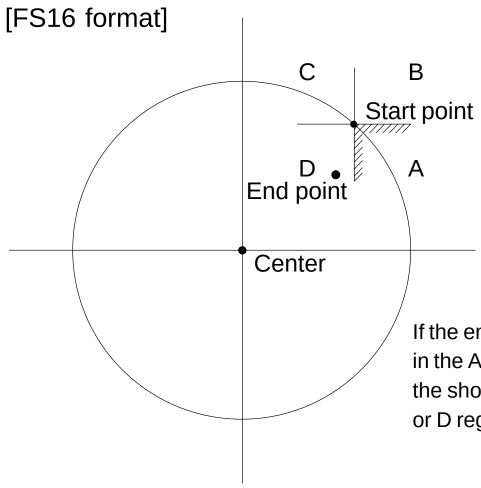
In the MDI mode, cutter compensation C (M series) or tool–nose radius compensation (T series) is not performed, irrespective of the setting of this parameter.

QCR The travel distance of circular interpolation in cutter compensation C (M series) or tool–nose radius compensation (T series) is judged:

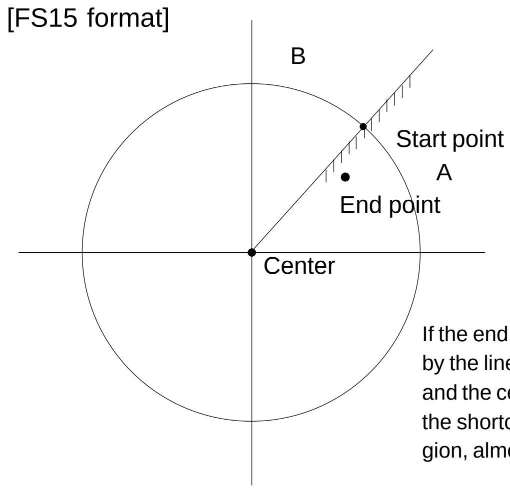
0 : In the FS16 format.

1 : In the FS15 format.

FS16 and FS15 determine the travel distance in different ways if the radius of arc at the start point of circular interpolation is different from that at the end point (if the end point is not on the arc). By this parameter, the method of determining the travel distance of circular interpolation can be selected.



If the end point viewed from the start point is in the A region, the movement is made along the shortcut. If the end point is in the B, C, or D region, almost a single turn is made.



If the end point is in the A region separated by the line L drawn between the start point and the center, the movement is made along the shortcut. If the end point is in the B region, almost a single turn is made.

NOTE

The setting of this parameter determines the travel distance determination method for circular interpolation not during cutter compensation C (M series) or tool–nose radius compensation (T series) as well. Accordingly, if this parameter is set, the setting of bit 3 (CQD) of parameter No. 3450 is invalid.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during tool nose radius compensation

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of a corner during tool nose radius compensation.

Alarm and message

Number	Message	Description
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.
5257	G41/G42 NOT ALLOWED IN MDI MODE	G41/G42 (cutter compensation C: M series, tool–nose radius compensation: T series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.14.2, 14.3	Tool Nose Radius Compensation
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.14.2, 14.3	Tool Nose Radius Compensation

11 PROGRAM COMMAND

11.1 DECIMAL POINT PROGRAMMING/ POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING

General

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F . . . . M series
(for a type common to all axes)
X, Y, Z, U, V, W, A, B, C, I, J, K, R, F. . . . . T series

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal point notation is used, a value without decimal point is considered to be specified in millimeters, inches or degree. When standard decimal point notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal point notation by using the DPI bit (bit 0 of parameter 3401). Setting the AXDx parameter (bit 0 of parameter No. 3455) to 1 (rather than using the M series) enables the calculator–type decimal input to be set up for individual axes separately. Values can be specified both with and without decimal point in a single program.

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point

0 : The least input increment is assumed.
1 : The unit of mm, inches, degree, or s is assumed. (Pocket calculator type decimal point programming)

	#7	#6	#5	#4	#3	#2	#1	#0
3455								
								AXD

[Data type] Bit axis

AXD If a decimal point is omitted for an address with which a decimal point can be used, the value is determined:

0 : In accordance with the least input increment.

1 : In millimeters, inches, or seconds. (calculator–type decimal point input)

NOTE

1 This parameter is valid if bit 0 (DPI) of parameter No. 3401 is set to 0.

2 Because some addresses (such as R and K) are not related to an axis, setting this parameter for all axes is not equivalent to setting bit 0 (DPI) of parameter No. 3401 to 1.

3 This parameter cannot be used together with:
  1) Macro executor
  2) Macro call argument
  3) Manual Guide 0*i*

Alarm and message

Number	Message	Description
007	ILLEGAL USE OF DECIMAL POINT	Decimal point “ · ” input error (A decimal point was input after an address with which it can not be used. Or multiple decimal points were input.) Modify the program.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.8.4	Decimal point programming
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.8.3	Decimal point programming
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.8.4	Decimal point programming
	OPERATOR’S MANUAL (T series) (B–64134EN)	II.8.3	Decimal point programming

11.2 G CODE SYSTEM (T SERIES)

General

There are three G code systems : A,B, and C (Table 11.2). Select a G code system using parameter GSC (No. 3401#7) and parameter GSB (No. 3401#6).

Table 11.2 G code list for T series (1/3)

G code A	G code B	G code C	Group	Function
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04	00	Dwell
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting

Table 11.2 G code list for T series (2/3)

G code A	G code B	G code C	Group	Function
G36	G36	G36	00	Automatic tool compensation X (When the bit 3 (G36) of parameter No. 3405 is set to 0)
G37	G37	G37		Automatic tool compensation Z
G37.1	G37.1	G37.1		Automatic tool compensation X
G37.2	G37.2	G37.2		Automatic tool compensation Z
G39	G39	G39		Corner circular interpolation
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G60	G60	G60	00	Single direction positioning
G65	G65	G65		Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON
G69	G69	G69		Mirror image for double turrets OFF
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle

Table 11.2 G code list for T series (3/3)

G code A	G code B	G code C	Group	Function
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscilation grinding cycle (for grinding machine)
G74	G74	G75		Oscilation direct constant–dimension grinding cycle (for grinding machine)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per rotation feed
–	G90	G90	03	Absolute programming
–	G91	G91		Incremental programming
–	G98	G98	11	Return to initial level
–	G99	G99		Return to R point level

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						

[Data type] Bit

GSB, GSC The G code system is set.

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR			G91			G01

[Data type] Bit

G01 Mode entered when the power is turned on or when the control is cleared

0 : G00 mode (positioning)
1 : G01 mode (linear interpolation)

G91 When the power is turned on or when the control is cleared

0 : G90 mode (absolute command)
1 : G91 mode (incremental command)

CLR Reset button on the CRT/MDI panel, external reset signal, reset and rewind signal and emergency stop signal

0 : Cause reset state.
1 : Cause clear state.

Alarm and message

Number	Message	Description
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.

Note

NOTE

1 If the CNC enters the clear state (see bit 6 (CLR) of parameter 3402) when the power is turned on or the CNC is reset, the modal G codes change as follows.
   (1) G codes marked with ◤ in Table 11.2 are enabled.
   (2) When the system is cleared due to power-on or reset, whichever specified, either G20 or G21, remains effective.
   (3) Bit 7 (G23) of parameter No. 3402 is used to specify whether G22 or G23 is to be selected upon power–on. The selection of G22 or G23 is not, however, changed when the CNC is cleared upon a reset.
   When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.
   (4) Setting bit 0 (G01) of parameter 3402 determines which code, either G00 or G01, is effective.
   (5) Setting bit 3 (G91) of parameter 3402 determines which code, either G90 or G91, is effective.

2 G codes of group 00 except G10 and G11 are single-shot G codes.

3 Alarm 010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.

4 G codes of different groups can be specified in the same block.
   If G codes of the same group are specified in the same block, the G code specified last is valid.

5 If a G code of group 01 is specified in a canned cycle, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.

6 When G code system A is used for a canned cycle, only the initial level is provided at the return point.

7 G codes are displayed for each group number.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.3	PREPARATORY FUNCTION (G FUNCTION)
		APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.3	PREPARATORY FUNCTION (G FUNCTION)
		APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

11.3 PROGRAM CONFIGURATION

General

A program consists of the following components:

Table 11.3 Program components

Components	Descriptions
Tape start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Tape end	Symbol indicating the end of a program file

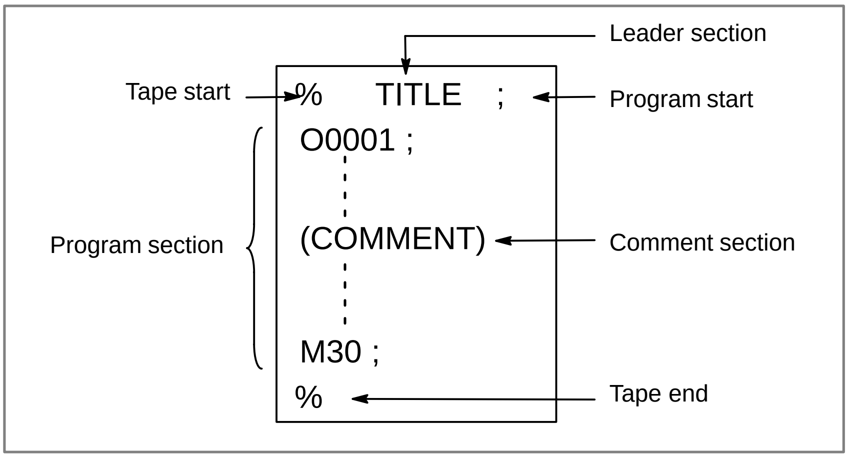


Fig. 11.3 Program configuration

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100							CTV	

This parameter can be set at “Setting Screen.”

[Data type] Bit

CTV Character counting for TV check in the comment section of a program.

0 : Not performed
1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99					

[Data type] Bit

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:

0 : Completed
1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:

0 : Completed
1 : Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3404		EOR				SBP		

[Data type] Bit

SBP Address P of the block including M198 in the subprogram call function

0 : Indicating a file number
1 : Indicating a program number

EOR When the end-of-record mark (%) is read during program execution:

0 : P/.S alarm No. 5010 occurs.
(Automatic operation is stopped, and the system enters the alarm state.)
1 : No alarm occurs.
(Automatic operation is stopped, and the system is reset.)

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

When the subprogram call function is used, this parameter sets the M code for calling a program in a file stored on the external input/output device.

NOTE
The M code is judged to be M198 when zero is specified as the setting value.

Alarm and message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input).
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective (when TVC, bit 0 of setting parameter 0000 is set to 1).
5010	END OF RECORD	The end of record (%) was specified.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.12	Program Configuration
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.12	Program Configuration
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.12	Program Configuration
	OPERATOR’S MANUAL (T series) (B–64134EN)	II.12	Program Configuration

11.4 INCH/METRIC CONVERSION

General

Either inch or metric input can be selected by G code.

Signal

Inch input signal INCH<F002#0>

[Classification] Output signal

[Function] This signal indicates that inch input mode is active.

[Output condition] "1"indicates that the inch input mode (G20) is in progress, and "0" indicates that metric input mode (G21) is in progress.

This signal changes to the corresponding state when modes are switched using the setting data display on the MDI panel.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002								INCH

Parameter

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

Setting entry is acceptable.

[Data type] Bit

INI Unit of input

0: In mm
1: In inches

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

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

[Data type] Bit

INM Least command increment on the linear axis

0 : In mm (metric system machine)
1 : In inches (inch system machine)

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

NOTE
When this parameter is changed, turn off the power before continuing operation.

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of the rotation type. (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is of linear axis type (i.e. not rounded in 0 to 360°). · Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axes roll over function and the index table indexing function (M series).

	#7	#6	#5	#4	#3	#2	#1	#0
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

NOTE
M series are not equipped with this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

	#7	#6	#5	#4	#3	#2	#1	#0
3104								MCN

[Data type] Bit

MCN Machine position is:

0 : Not displayed according to the unit of input.
(Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.)
1 : Displayed according to the unit of input.
(When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point

0 : Assumed to be 0.001
1 : Depending on the input increment.
(For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:

0 : Not performed
1 : Performed

NOTE
If you change this parameter, re–set the tool compensation data.

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

[Data type] Bit

PIM When only the axes controlled by the PMC are used, the linear axis is:
0 : Influenced by inch/millimeter input.
1 : Not influenced by inch/millimeter input.

Warning

WARNING
When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re–set according to the least input increment.
However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re–set.

Note

NOTE
1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
2 Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.
3 The inch and metric input can also be switched using settings.

Reference item

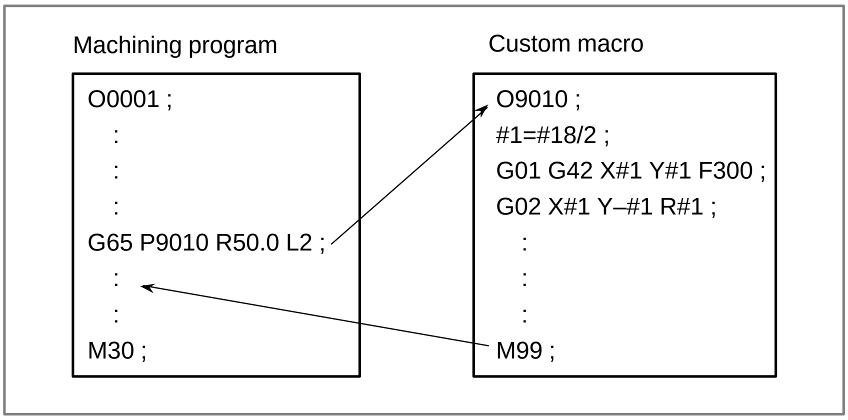
Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.8.3	Inch/metric conversion (G20, G21)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.8.2	Inch/metric conversion (G20, G21)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.8.3	Inch/metric conversion (G20, G21)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.8.2	Inch/metric conversion (G20, G21)

11.5 CUSTOM MACRO

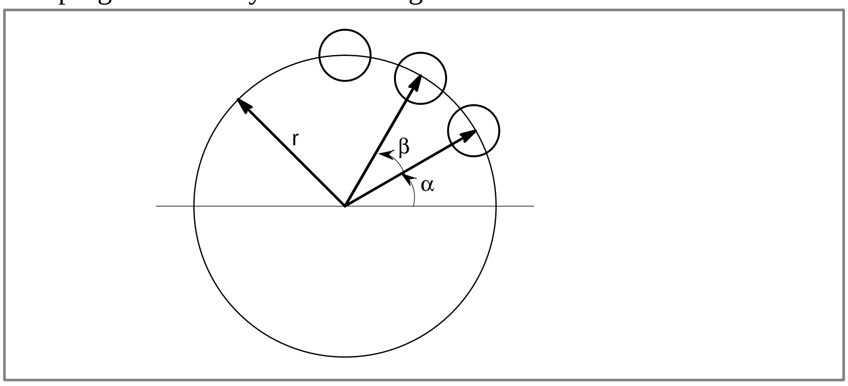
11.5.1 Custom Macro

General

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user–defined canned cycles. A machining program can call a custom macro with a simple command, just like a subprogram.



This means that a function of general use can be formed when programming a certain function as a custom macro. That is, programs can be written using variables for data that might change or be unknown. This can be further applied to group technology.
Similar workpieces can be collected as a group and a universal custom macro body can be programmed using variables applicable to each group. In this way, programming is not required for the workpieces in the group. The programmer only need to assign actual values to the variables.



Bolt hole circles as shown in the above figure can be made easily.
Once a custom macro body for the bolt hole circle is programmed and registered, the CNC can operate as if it has the bolt hole circle cutting function.
Programmers can use the bolt hole circle function by using the following command only:
(Example of calling bolt hole circle)

G65 P$\underline{p}$ R$\underline{r}$ A$\underline{\alpha}$ B$\underline{\beta}$ K$\underline{k}$;

P : Macro number of bolt hole circle
r : Radius
α : Start angle
β : Angle between circles
k : Number of circles

Signal

Custom Macro Input Signal UI000 to UI015 <G054, G055>

[Classification] Input signal

[Function] No function is provided for the control unit. These signals can be read by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UI000 UI001 UI002 UI003 : UI014 UI015	1 1 1 1 : 1 1	#1000 #1001 #1002 #1003 : #1014 #1015	"0" at "0" and "1" at "1"
UI000 to UI015	16	#1032	16–bit binary code *1

*1 Variable value $\#1032 = \sum_{i=0}^{15} \{\#[1000 + i] \times 2^{i}\}$

Custom Macro Output Signal UI000 to UI015 <F054, F055> UO100 to UO131 <F056 to F059>

[Classification] Output signal

[Function] No function is provided for the control unit. These signals can be read or written by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UO000	1	#1100	"0" at "0" and "1" at "1"
UO001	1	#1101	
UO002	1	#1102	
UO003	1	#1103	
:	:	:	
UO014	1	#1114	
UO015	1	#1115	
UO000AUO015	16	#1132	16–bit binary code *1
UO100AUO115	32	#1133	32–bit binary code *2

*1 Variable value $\#1132 = \sum_{i=0}^{15}\{\#[1100 + i] \times 2^{i}\}$

*2 Variable value $\#1133 = \sum_{i=0}^{30}\{2^{i} \times V_{i}\} - 2^{31} \times V_{31}$

Where Vi=0 when UO1i is H0Iand Vi=1 when UO1i is H1I

These system variables can be used on the left side of an assignment statement as well as on the right side.
The value assigned to the system variable used on the left side last is used for the value of the system variable to be assigned on the right side.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008

	#7	#6	#5	#4	#3	#2	#1	#0
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Parameter

- **Settings for executing single block stop and custom macros**

	#7	#6	#5	#4	#3	#2	#1	#0
6000	SBV		SBM	HGO		HMC	MGO	

[Data type] Bit

MGO When a GOTO statement for specifying custom macro control is executed, a high–speed branch to 20 sequence numbers executed from the start of the program is:
0 : A high–speed branch is not caused to n sequence numbers from the start of the executed program.
1 : A high–speed branch is caused to n sequence numbers from the start of the program.

HMC A custom macro is executed:
0 : At a normal speed.
1 : At a high speed.

> **NOTE**
> When this parameter is set, the CNC executes a custom macro first. For this reason, when this parameter is set, performance of the following functions may be degraded:
> - Screen display of CNC
> - Macro executor (excluding execution macros)

HGO When a GOTO statement for specifying custom macro control is executed:
0 : A high–speed branch is not caused to 30 sequence numbers, immediately following the point of execution.
1 : A high–speed branch is caused to 30 sequence numbers, immediately before the point of execution.

SBM Custom macro statement
0: Not stop the single block
1: Stops the single block

If you want to disable the single blocks in custom macro statements using system variable #3003, set this parameter to 0. If this parameter is set to 1, the single blocks in custom macro statements cannot be disabled using system variable #3003. To control single blocks in custom macro statements using system variable #3003, use bit 7 (SBV) of parameter No. 6000.

> **NOTE**
> This bit is invalid when bit 0 (NOP) of parameter No. 6000 is set to 1. (M series)

SBV Custom macro statement
0 : Not stop the single block
1 : Stops the single block

To control single blocks in custom macro statements using system variable #3003, use this parameter to enable or disable single blocks in custom macro statements.

This bit is valid when bit 5 (SBM) of parameter No. 6000 is set to 0.

- **Other settings**

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV	TCS	CRO	PV5		PRT	

[Data type] Bit

PRT Reading zero when data is output using a DPRNT command
0 : Outputs a space
1 : Outputs no data

PV5 Custom macro common variables
0 : Outputs custom macro common variables #500 through #599.
1 : Outputs custom macro common variables #100 through #199 and #500 through #599.

CRO ISO code output using a BPRNT command or a DPRNT command
0 : Outputs only LF after data is output
1 : Outputs LF and CR after data is output

TCS Custom macro (subprogram)
0 : Not called using a T code
1 : Called using a T code

CCV Custom macro’s common variables Nos. 100 through 149
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

CLV Custom macro’s local variables #1 through #33
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

- **Setting when macro statement is input/output with EIA code**

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0
6011	=7	=6	=5	=4	=3	=2	=1	=0
6012	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
6014	]7	]6	]5	]4	]3	]2	]1	]0

[Data type] Bit
These parameters are used to input/output macro statements with EIA code.

The numeral of a suffix indicates the bit position in a code.

***0 to *7** Set the hole pattern of an EIA code indicating *.

=0 to =7 Set the hole pattern of an EIA code indicating =.

#0 to#7 Set the hole pattern of an EIA code indicating #.

[0 to [7 Set the hole pattern of an EIA code indicating [.

]0 to]7 Set the hole pattern of an EIA code indicating].

0 :Corresponding bit is 0

1 :Corresponding bit is 1.

- **Setting an M code that calls a program entered in a file**

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

When the subprogram call function is used, this parameter sets the M code for calling a program in a file stored on the external input/output device.

> **NOTE**
> The M code is judged to be M198 when zero is specified as the setting value.

- **Setting G codes that call custom macros of program Nos.9010 to 9019**

6050	G code that calls the custom macro of program number 9010
6051	G code that calls the custom macro of program number 9011
6052	G code that calls the custom macro of program number 9012
6053	G code that calls the custom macro of program number 9013
6054	G code that calls the custom macro of program number 9014
6055	G code that calls the custom macro of program number 9015
6056	G code that calls the custom macro of program number 9016
6057	G code that calls the custom macro of program number 9017
6058	G code that calls the custom macro of program number 9018
6059	G code that calls the custom macro of program number 9019

[Data type] Word

[Valid data range] 1 to 999

These parameters set the G codes that call the custom macros of program numbers 9010 through 9019.

> **NOTE**
> Setting value 0 is invalid. No custom macro can be called by G00.

- **Setting M codes that call subprograms of program Nos.9001 to 9009**

6071	M code that calls the subprogram of program number 9001
6072	M code that calls the subprogram of program number 9002
6073	M code that calls the subprogram of program number 9003
6074	M code that calls the subprogram of program number 9004
6075	M code that calls the subprogram of program number 9005
6076	M code that calls the subprogram of program number 9006
6077	M code that calls the subprogram of program number 9007
6078	M code that calls the subprogram of program number 9008
6079	M code that calls the subprogram of program number 9009

[Data type] Two–word

[Valid data range] 1 to 99999999

These parameters set the M codes that call the subprograms of program numbers 9001 through 9009.

> **NOTE**
> Setting value 0 is invalid. No custom macro can be called by M00.

- **Setting M codes that call custom macros of no.9020 to 9029**

6080	M code that calls the custom macro of program number 9020
6081	M code that calls the custom macro of program number 9021
6082	M code that calls the custom macro of program number 9022
6083	M code that calls the custom macro of program number 9023
6084	M code that calls the custom macro of program number 9024
6085	M code that calls the custom macro of program number 9025
6086	M code that calls the custom macro of program number 9026
6087	M code that calls the custom macro of program number 9027
6088	M code that calls the custom macro of program number 9028
6089	M code that calls the custom macro of program number 9029

[Data type] Two–word

[Valid data range] 1 to 99999

These parameters set the M codes that call the custom macros of program numbers 9020 through 9029.

> **NOTE**
> Setting value 0 is invalid. No custom macro can be called by M00.

- **ASCII codes that call subprogram of program No. 9004**

6090	ASCII code that calls the subprogram of program number 9004
6091	ASCII code that calls the subprogram of program number 9005

[Data type] Byte

[Valid data range] 65 (A:41H) to 90 (Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal. Addresses that can be used are as follows:

T series : A, B, F, H, I, K, M, P, Q, R, S, T
M series: A, B, D, F, H, I, J, K, L, M, P, Q, R, S, T, X, Y, Z

> **NOTE**
> Set 0 when no subprogram is called

Alarm and message

Number	Message	Description
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro, or the header contents are improper in a high–speed cycle cutting. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.

Number	Message	Description
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦ 3 is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
199	MACRO WORD UNDE FINED	Undefined macro word was used. Modify the custom macro.

Caution

CAUTION

Machine tool builders: You are requested to attach your custom macro program tape or program list to the CNC unit without fail.

If it is necessary to replace part program storage memory due to a failure, FANUC servicemen or end user in charge of maintenance should know the contents of your custom macro for the purpose of repairing the trouble immediately.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.15	Custom macro
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.15	Custom macro
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.15	Custom macro
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.15	Custom macro

11.5.2 Interruption Type Custom Macro

General

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine. This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

M96 P○○○○ ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.

(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.

(3) At regular intervals, information on current machining is read.

Listed above are examples like adaptive control applications of the interruption type custom macro function.

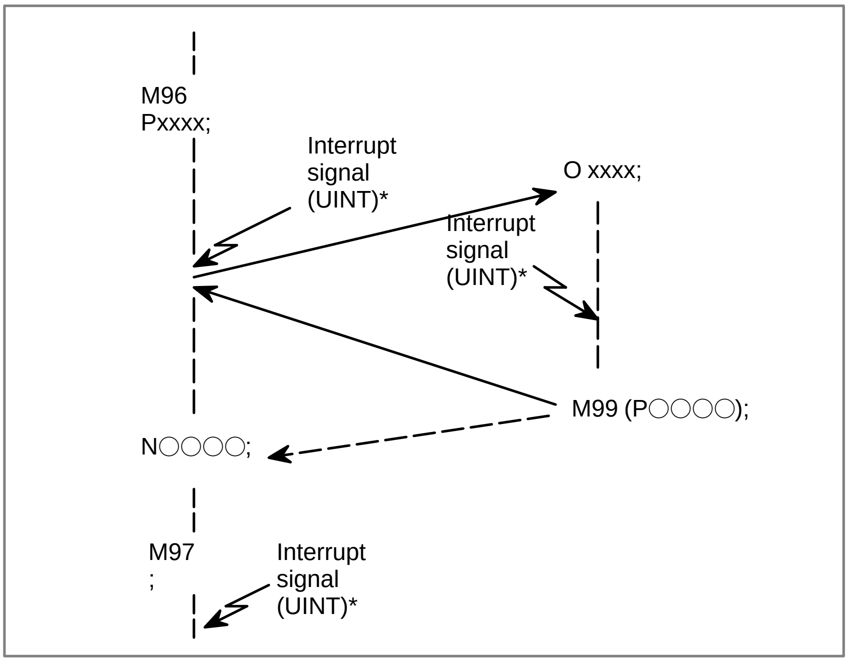


Fig 11.5.2 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.
Any interrupt signal (UNIT, asterisked in Fig. 11.5.2) issued after M97 is ignored.
Do not enter an interrupt signal during execution of an interrupt macro.

Signal

Interrupt Signal for Custom Macro UINT<G053#3>

[Classification] Input signal

[Function] This signal calls and executes a program in memory. During execution, a program in automatic operation is suspended.
To enable this signal to be accepted, a particular miscellaneous function must be specified in a command program for automatic operation. In addition, automatic operation must already be started to accept this signal.
The particular miscellaneous function code is set by parameter 6003, 6033 and 6034.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053					UINT			

Parameter

- **Various Setting for Custom Macro**

	#7	#6	#5	#4	#3	#2	#1	#0
6003	MUS	MCY	MSB	MPR	TSE	MIN	MSK	

[Data type] Bit

MSK Absolute coordinates at that time during custom macro interrupt

0 : Not set to the skip coordinates (system variables #5061 and later)
1 : Set to the skip coordinates (system variables #5601 and later)

MIN Custom macro interrupt

0 : Performed by interrupting an in–execution block (Custom macro interrupt type I)
1 : Performed after an in–execution block is completed (Custom macro interrupt type II)

TSE Custom macro interrupt signal UINT

0 : Edge trigger method (Rising edge)
1 : Status trigger method

MPR Custom macro interrupt valid/invalid M code

0 : M96/M97
1 : M code set using parameters (Nos. 6033 and 6034)

MSB Interrupt program

0 : Uses a dedicated local variable (Macro–type interrupt)
1 : Uses the same local variable as in the main program (Subprogram–type interrupt)

MCY Custom macro interrupt

0 : Not performed during cycle operation
1 : Performed during cycle operation

MUS Interrupt–type custom macro

0 : Not used
1 : Used

- **Setting M code that makes interruption effective and ineffective**

6033	M code that validates a custom macro interrupt
6034	M code that invalidates a custom macro interrupt

[Data type] Byte type

[Valid data range] 0 to 255

These parameters set the custom macro interrupt valid/invalid M codes.

> **NOTE**
> These parameters can be used when MPR, #4 of parameter No. 6003, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

Note

> **NOTE**
> 1 No interrupt–type custom macro can be used during DNC operation.
> 2 No interrupt–type custom macro can be used during multiple repetitive canned cycle execution.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.15.11	Interruption type custom macro
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.15.11	Interruption type custom macro
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.15.11	Interruption type custom macro
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.15.11	Interruption type custom macro

11.6 CANNED CYCLE (M SERIES)/CANNED CYCLE FOR DRILLING (T SERIES)

General

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

Explanations

A canned cycle consists of a sequence of six operations.

Operation 1 Positioning a hole position
Operation 2 Rapid traverse up to point R level
Operation 3 Hole machining
Operation 4 Operation at the bottom of a hole
Operation 5 Retraction to point R level
Operation 6 Rapid traverse up to the initial point

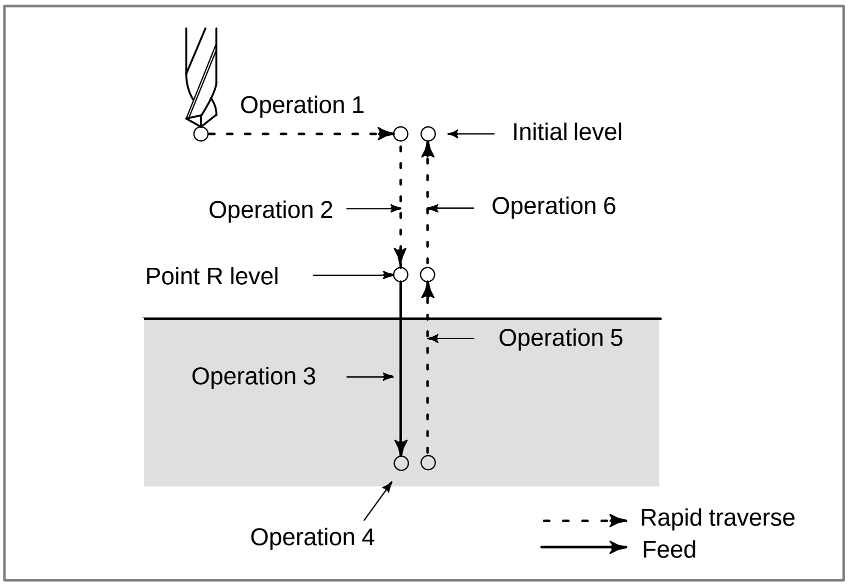


Fig. 11.6 (a) Canned cycle operation sequence

SPINDLE CONTROL

In some canned cycles, a spindle command to rotate the spindle in reverse direction may be output.

The following canned cycles require spindle control:

M series	T series
Reverse tapping cycle G74	Face tapping cycle (G84)
Fine boring cycle G76	Side tapping cycle (G88)
Tapping cycle G84	
Boring cycle G86	
Back boring cycle G87	
Boring cycle G88	

For spindle control, the following normal miscellaneous functions are used:

See the description of the miscellaneous functions.

M03: CW spindle rotation
M04: CCW spindle rotation
M05: Spindle stop
M19: Spindle orientation (M series)

When the rotation direction of the spindle is to be switched from one direction to the other (for example, when M04 is output during M03 operation), a parameter can specify whether to send M05 at the time switching.

Timing charts are described in the following page:

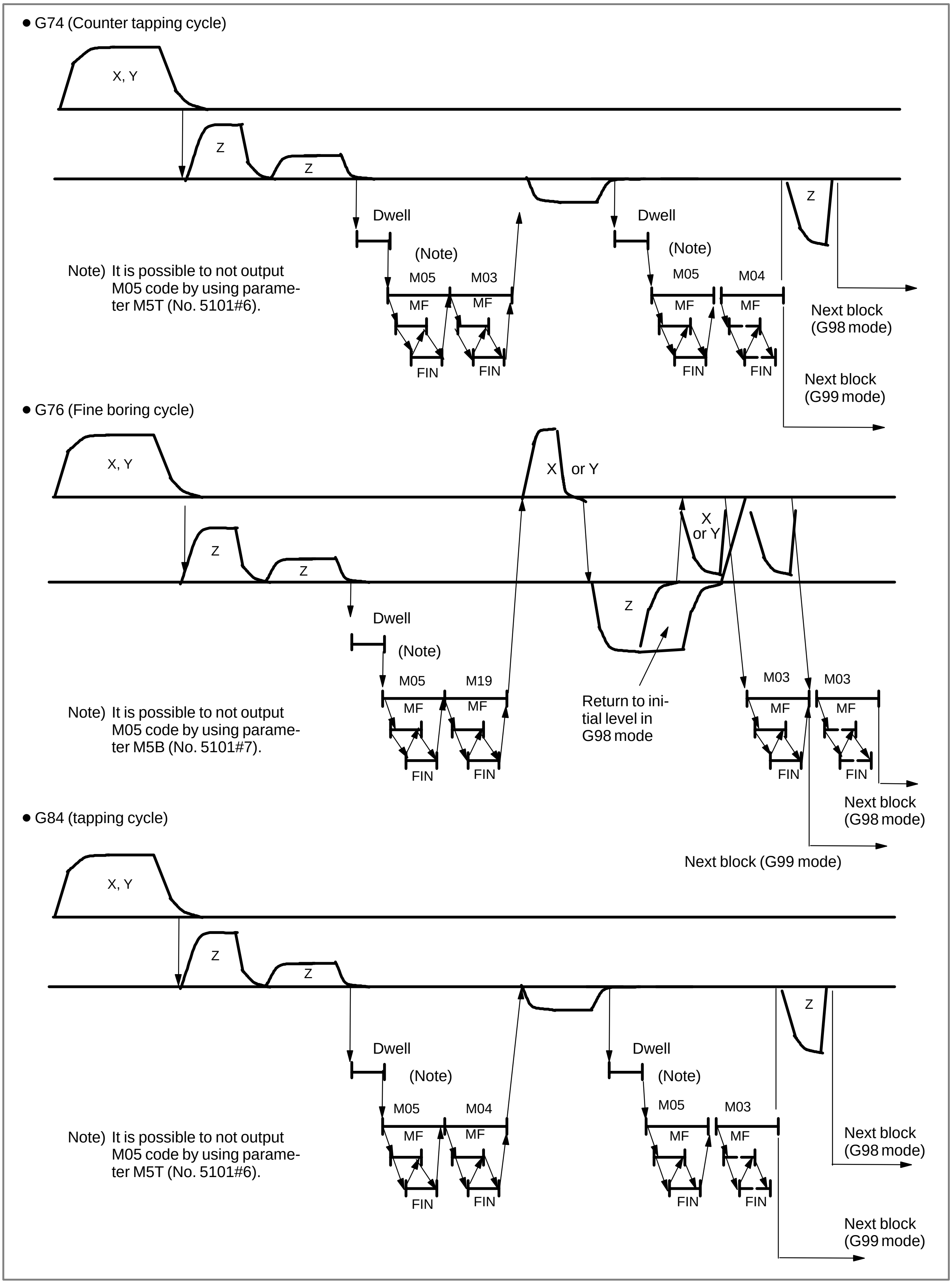


Fig. 11.6 (b) Canned cycle for M series (1/2)

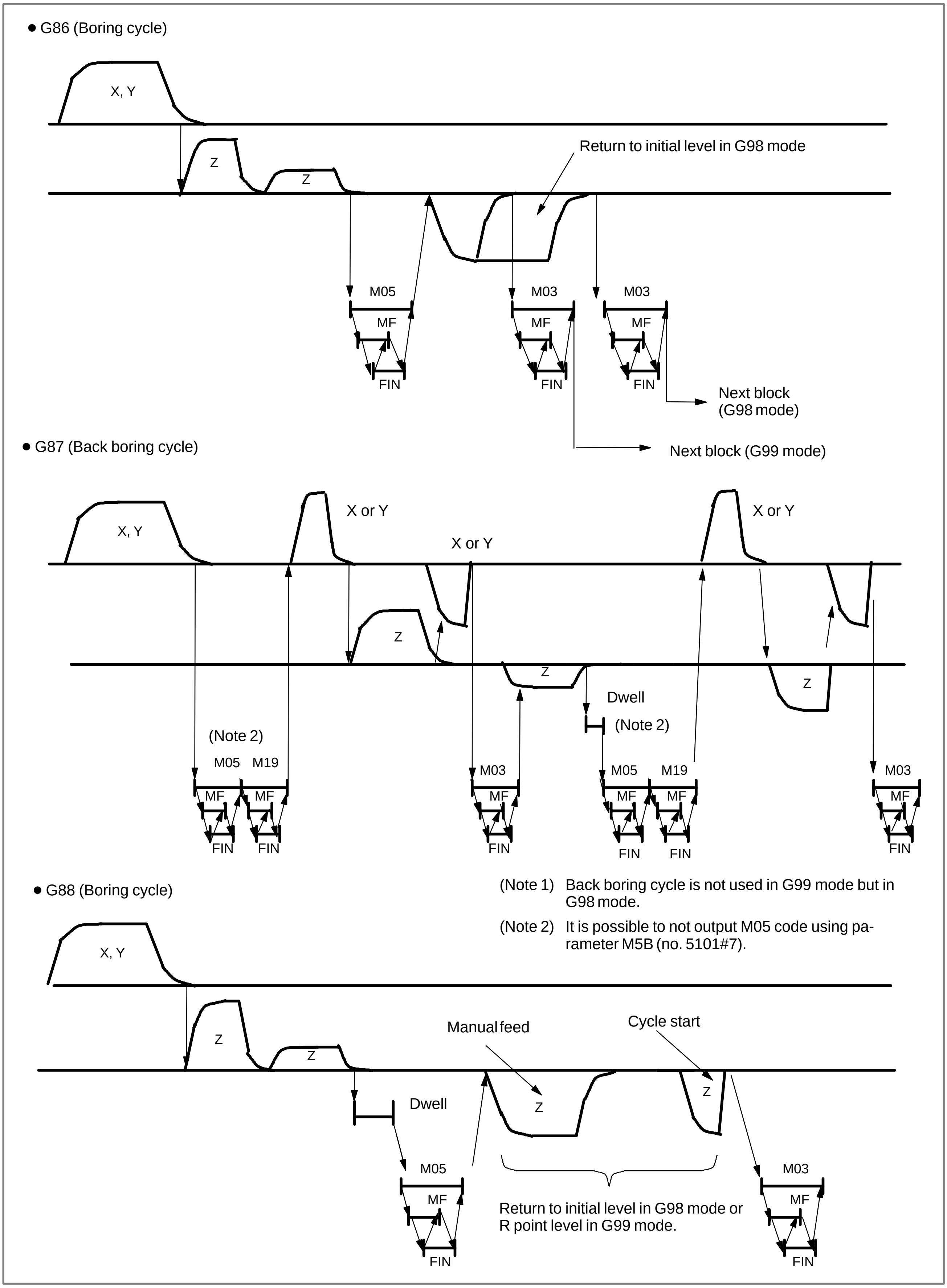


Fig. 11.6 (c) Canned cycle for M series (2/2)

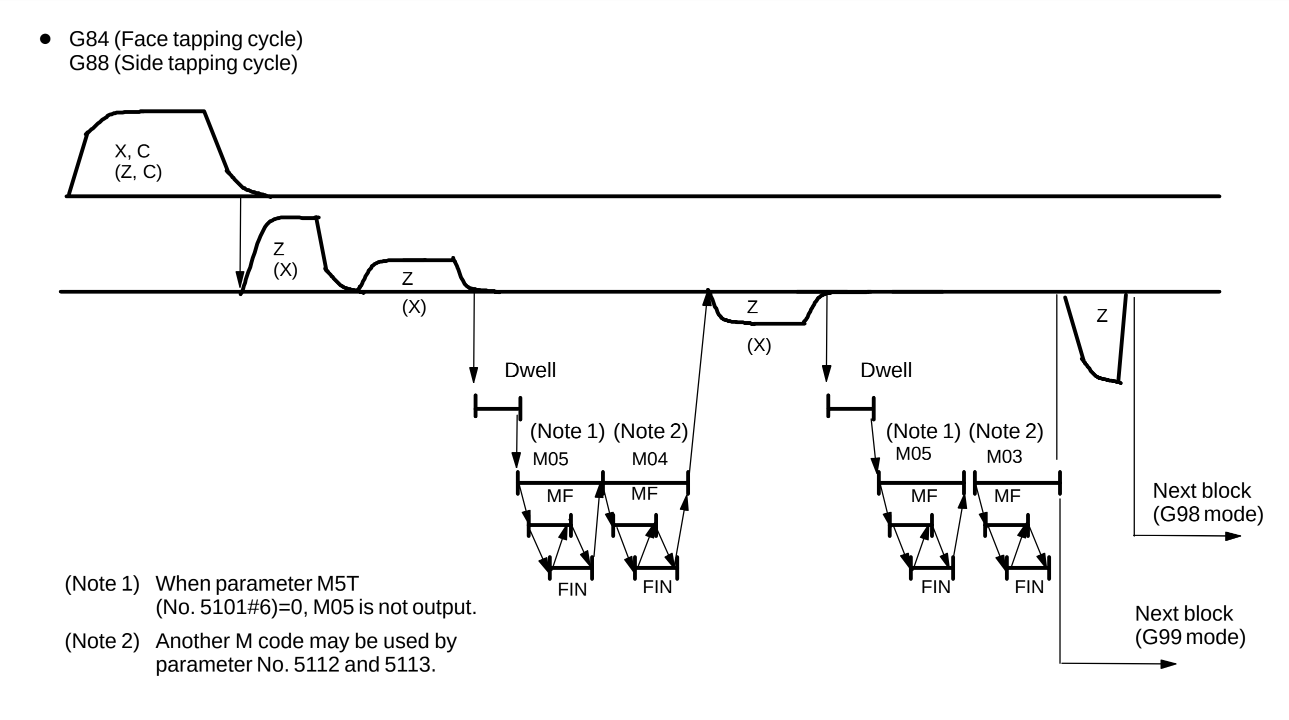


Fig 11.6 (d) Canned cycle for T series

- **M code used for C–axis clamp/unclamp (T series)**

When an M code specified in parameter No.5110 for C–axis clamp/unclamp is programmed, the CNC issues the M code for C–axis clamp after the tool is positioned and before the tool is fed in rapid traverse to the point–R level. The CNC also issues the M code (M code C–axis clamp +1) for C–axis unclamp after the tool retracts to the point–R level. The tool dwells for the time specified in parameter No. 5111.

Tapping signal

During a tapping cycle, the tapping signal is output. The tapping signal is also output while the G code of the tapping cycle is valid.

Override

During tapping, cutting feedrate override is always set to 100%.

Feed hold

When the feed hold key is pressed during tapping, the movement is not stopped immediately but the movement is stopped when the tool is returned to level R.

Dry run

The TDR bit (bit 5 of parameter No. 1401) specifies whether dry run is valid during tapping.

Signal

Tapping signal
TAP <F001#5>

[Classification] Output signal

[Function] Reports that the system is in tapping mode.

[Output condition] The signal is set to 1 when:
– The system is in tapping cycle mode.
G74, G84: M series
G84, G88: T series
– The system is in tapping mode.
G63: M series
The signal is set to 0 when:
– The system is in neither tapping cycle mode nor tapping mode.
– A reset or emergency stop is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F001			TAP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101		M5T			ILV	RTR		FXY
	M5B	M5T	RD2	RD1			EXC	FXY

[Data type] Bit

FXY The drilling axis in the drilling canned cycle is:
0 : Always the Z–axis
1 : The axis selected by the program

> **NOTE**
> In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 10/11 format.

EXC G81
0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

RTR G83 and G87
0 : Specify a high–speed peck drilling cycle
1 : Specify a peck drilling cycle

ILV Initial point position in drilling canned cycle
0 : Not updated by reset
1 : Updated by reset

RD2, RD1 Set the axis and direction in which the tool in drilling canned cycle G76 or G87 is got free. RD2 and RD1 are set as shown below by plane selection.

RD2	RD1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	–X	–Z	–Y
1	0	+Y	+X	+Z
1	1	–Y	–X	–Z

M5T When a spindle rotates from the forward to the reverse direction and vice versa in tapping cycles G84 and G74 for M series (G84 and G88 for T series), befor M04 or M03 is output:
For T series

0 : Not output M05
1 : Outputs M05

For M series

0 : Outputs M05
1 : Not output M05

M5B In drilling canned cycles G76 and G87:

0 : Outputs M05 before an oriented spindle stops
1 : Not output M05 before an oriented spindle stops

	#7	#6	#5	#4	#3	#2	#1	#0
5102	RDI	RAB						

[Data type] Bit

RAB The R command for the drilling canned cycle in the Series 10/11 tape format is:

0 : Regarded as an incremental command
1 : Regarded as:
An absolute command in the case of G code system A
An absolute command in the case of G code system B or C when the G90 mode is specified.
An incremental command in the case of G code system B or C when the G91 mode is specified.

RDI The R command for the drilling canned cycle in the Series 10/11 tape format:

0 : Is regarded as the specification of a radius
1 : Follows the specification of a diameter/radius for the drilling axis

	#7	#6	#5	#4	#3	#2	#1	#0
5103								
								SIJ

[Data type] Bit

SIJ A tool shift value for the drilling canned cycle G76 or G87 in the Series 10/11 tape format is specified by:

0 : Address Q
1 : Address I, J, or K

5110	C–axis clamp M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 99

This parameter sets the C–axis clamp M code in a drilling canned cycle.

5111	Dwell time when C–axis unclamping is specified in drilling canned cycle

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 32767

This parameter sets the dwell time when C–axis unclamping is specified in a drilling canned cycle.

5112	Spindle forward–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle forward–rotation M code in a drilling canned cycle.

> **NOTE**
> M03 is output when “0” is set.

5113	Spindle reverse–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle reverse–rotation M code in a drilling canned cycle.

> **NOTE**
> M04 is output when “0” is set.

5114	Return or clearance value of drilling canned cycle G83
	Return value of high–speed peck drilling cycle G73

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 32767

For M series, this parameter sets the return value in high–speed peck drilling cycle G73 (G83 for T series).

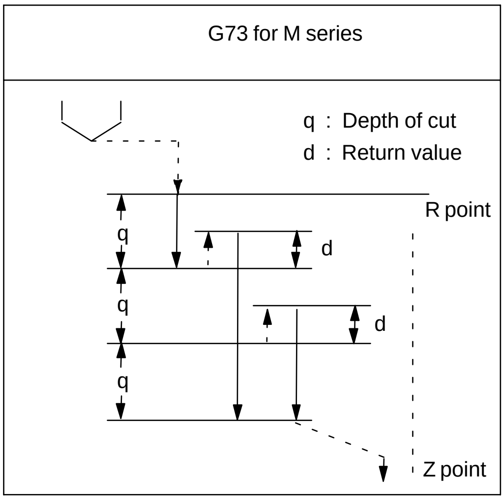


Fig. 11.6 (e) High–speed peck drilling cycle (G73) for M series

For T series, this parameter sets the return or clearance value in drilling canned cycle G83.

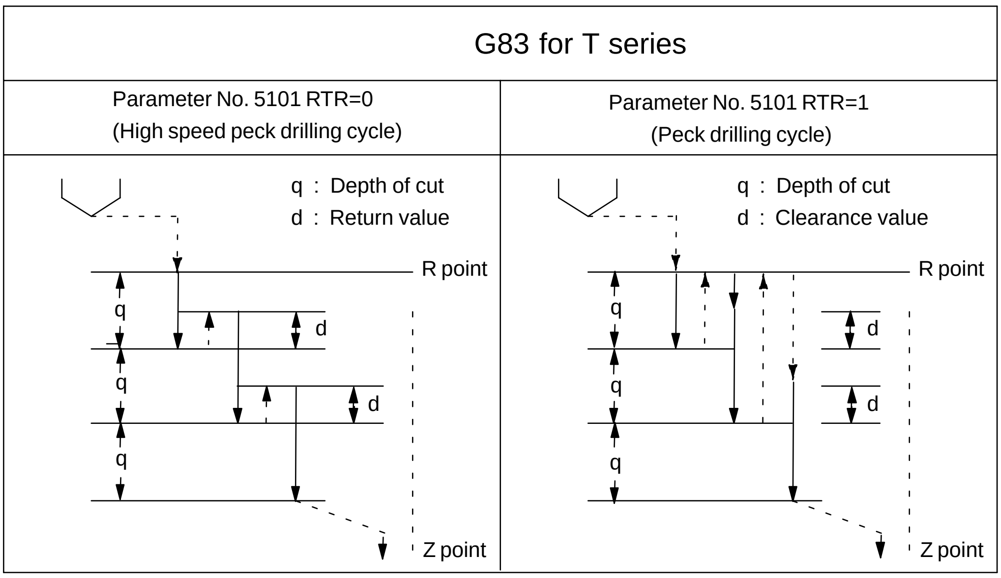


Fig. 11.6 (f) Drilling canned cycle (G83) for T series

5115	
	Clearance canned cycle G83

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 32767

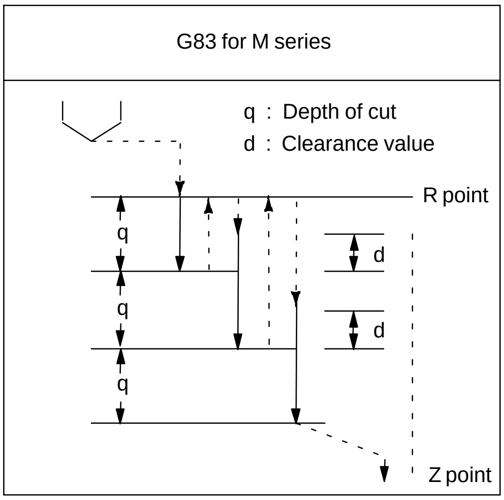


Fig. 11.6 (g) Peck drilling cycle (G83) for M series

Alarm and message

Number	Message	Description
044	G27–G30 NOT ALLOWED IN FIXED CYCLE (M series)	One of G27 to G30 is commanded in a canned cycle mode. Modify the program.

Note

NOTE

A parameter FXY (No. 5101#0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.13.1	Canned cycle
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.13.3	Canned cycle for hole machining
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.13.1	Canned cycle
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.13.3	Canned cycle for hole machining

11.7 EXTERNAL MOTION FUNCTION (M SERIES)

General

Upon completion of positioning in each block in the program, an external operation function signal can be output to allow the machine to perform specific operation.

G81 IP_ ; (The IP_ is axis move command)

Every time positioning for the IP_ move command is completed, the CNC sends an external operation signal to the machine. An external operation signal is output for each positioning operation until canceled by G80 or a group 01 G code.

No external operation signals are output during execution of a block that contains neither X nor Y.

Basic procedure

1 Once positioning for a move command has been completed, the CNC sets the external operation signal EF to 1.
2 When the EF signal is set to 1, the PMC executes drilling or another operation. Once the operation has been completed, the PMC sets completion signal FIN to 1.
3 The CNC resets the EF signal to 0 upon the elapse of the time (TFIN) specified in parameter No. 3011 after the FIN signal is set to 1.
4 When the EF signal is set to 0, the PMC resets the FIN signal to 0.
5 The CNC starts executing the next block.

The timing diagram is shown below:

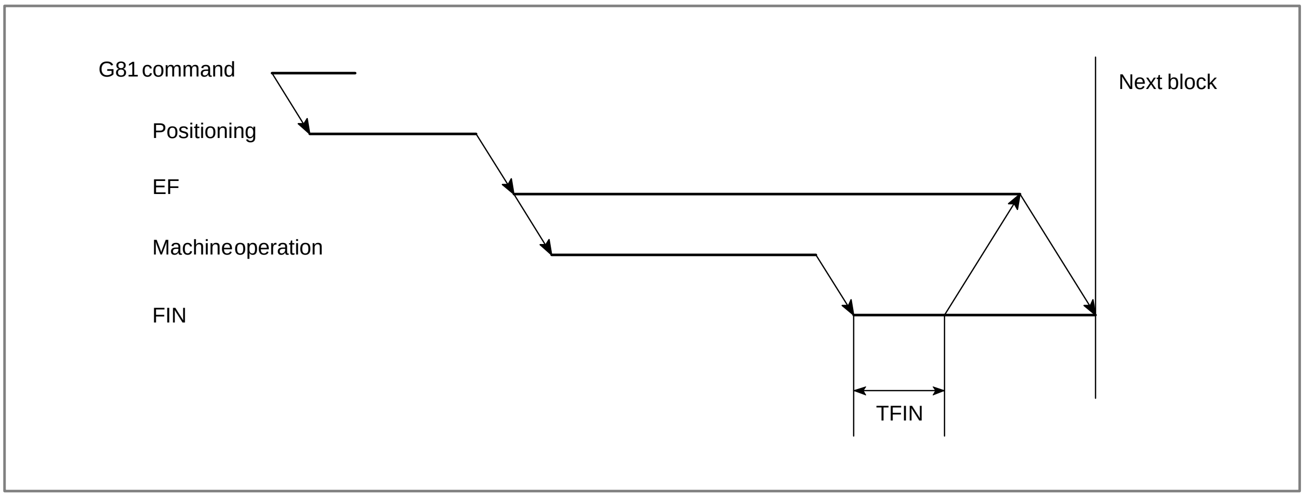


Fig. 11.7 Timing diagram of basic procedure

Signal

External Operation Signal EF<F008#0>

[Classification] Output signal

[Function] Reports that the positioning of G81 has been completed in the external motion function, and that a special external operation is required.

[Output condition] For details of the output condition and procedure, see the "basic procedure", described previously.
For details of completion signal FIN, see section 8.1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008								EF

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101							EXC	

[Data type] Bit

EXC G81:

0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

Caution

CAUTION
1 When this function is used, canned cycles (G73, G74, G76, and G82 to G89) cannot be used.
2 When the high–speed M, S, T, or B interface is used, the signals used by this function are transferred in high–speed mode. See Section 8.4.

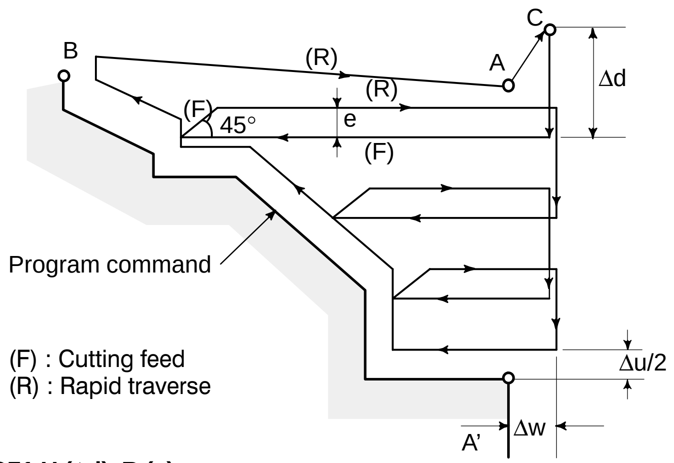
Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.13.7	External operation function (G81)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.13.4	External operation function (G81)

11.8 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T SERIES)

General

The option canned cycles makes CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available. The following example shows stock removals in turning type I.
If a finished shape of A to A' to B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance $\Delta u/2$ and Δw left.



G71 U (Δd) R (e) ;
G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t)

N (ns)··········
················
F____
S____
T____
N (nf)··········;

The move command of a finished shape of A to A' to B is specified in the blocks from sequence number ns to nf.

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).

Δw : Distance and direction of finishing allowance in Z direction.

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Signal

Chamfering signal CDZ<G053#7>

[Classification] Input signal

[Function] Executes chamfering in a threading cycle. Specify the chamfering distance in parameter No. 5130.

[Operation] When the signal is set to 1, chamfering is not executed in the threading cycle.
When the signal is set to 0, chamfering is executed in the threading cycle.

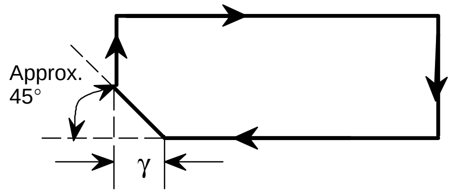


Fig. 11.8 (a) Straight thread cutting cycle

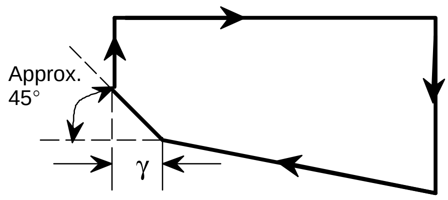


Fig. 11.8 (b)Taper thread cutting cycle

Set the chamfering distance γ to the parameter No. 5130. The chamfering distance can be specified in G76. The chamfering angle is made smaller than 45° by the remaining pulses in the automatic acceleration/deceleration circuit and servo system.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053	CDZ							

Parameter

- **Various setting for multiple repetitive canned cycle**

	#7	#6	#5	#4	#3	#2	#1	#0
5102						QSR	MRC	

[Data type] Bit

MRC When a target figure other than a monotonically increasing or monotonically decreasing figure is specified in a multiple repetitive turning canned cycle (G71, G72):

0 : No alarm occurs.
1 : P/S alarm No. 064 is occurs.

NOTE
This parameter is valid for multiple repetitive turning canned cycle type I.

QSR Before a multiple repetitive canned cycle (G70 to G73) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:

0 : Not made.
1 : Made. (If the sequence number specified in address Q cannot be found, an alarm occurs and the canned cycle is not executed.)

- **Chamfering distance in thread cutting cycles G76 and G92**

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1

[Valid data range] 0 to 127

This parameter sets the chamfering distance in thread cutting cycles G76 and G92.

- **Depth of cut in multiple repetitive canned cycles G71 and G72**

5132	Depth of cut in multiple repetitive canned cycles G71 and G72

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the depth of cut in multiple repetitive canned cycles G71 and G72.

- **Escape in multiple repetitive canned cycles G71 and G72.**

5133	Escape in multiple repetitive canned cycles G71 and G72.

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the escape in multiple repetitive canned cycles G71 and G72.

• Escape in multiple repetitive canned cycles G73

5135	Escape in multiple repetitive canned cycle G73 in X–axis direction
5136	Escape in multiple repetitive canned cycle G73 in Z–axis direction

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the escape in multiple repetitive canned cycle G73 of an X, then Z axis.

• Division count in multiple repetitive canned cycle G73

5137	Division count in multiple repetitive canned cycle G73

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 0 to 99999999

This parameter sets the division count in multiple repetitive canned cycle G73.

• Return in multiple canned cycles G74 and G75

5139	Return in multiple canned cycles G74 and G75

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the return in multiple repetitive canned cycles G74 and G75.

- **Minimum depth of cut in multiple repetitive canned cycle G76**

5140	Minimum depth of cut in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the minimum depth of cut in multiple repetitive canned cycle G76.

- **Finishing allowance in multiple repetitive canned cycle G76**

5141	Finishing allowance in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.001	mm
Inch input	0.0001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter sets the finishing allowance in multiple repetitive canned cycle G76.

- **Repetition count of final finishing in multiple repetitive canned cycle G76**

5142	Repetition count of final finishing in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in multiple repetitive canned cycle G76.

- **Tool nose angle in multiple repetitive canned cycle G76.**

5143	Tool nose angle in multiple repetitive canned cycle G76.

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 when FS10/11 tape format is used
0, 29, 30, 55, 60 and 80 when FS10/11 tape format is not used.

This parameter sets the tool nose angle in multiple repetitive canned cycle G76.

Alarm and message

Number	Message	Description
061	ADDRESS P/Q NOT FOUND IN G70–G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76	1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 The negative value is specified to Δi or Δk in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6 Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which is not monotone increase or decrease was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73	1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G–CODE IN G71–G73	An unallowable G code was commanded beween two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q was specified. Modify the program.
069	FORMAT ERROR IN G70–G73	The final move command in the blocks specified by P and Q of G70, G71, G72, or G73 ended with chamfering or corner R.

Cautions for multiple repetitive canned cycle (G70 to G76)

1 Necessary parameters (such as P, Q, X, Z, U, W, and R) must be set correctly for an individual block that specifies a multiple repetitive canned cycle.

2 In G71, G72, and G73 blocks having a sequence number specified using P, always specify G00 or G01 in group 01, or otherwise a P/S alarm (No. 65) will be issued.

3 None of G70, G71, G72, and G73 can be specified in the MDI mode. If any of them is specified, a P/S alarm (No. 67) will be issued. G74, G75, and G76 can be specified, however.

4 M98/M99 cannot be specified in a G70, G71, G72, G73 block, or any block having a sequence number between those specified using P or Q in a G70, G71, G72, or G73 block.

5 The following commands cannot be specified in any block having a sequence number between those specified using P or Q in a G70, G71, G72, or G73 block.

- One–shot G code except dwell (G04)
- G code other than G00, G01, G02, or G03 in group 01
- G codes in group 06
- M98/M99

6 It is possible to stop operation during multiple repetitive canned cycle (G70 to G76) execution to allow manual intervention. Before restarting the multiple repetitive canned cycle, be sure to return the tool to the point of manual intervention, because the distance through which the tool is caused to move by manual intervention is not included in the absolute value (the actual tool position deviates by that distance).

7 When G70, G71, G72, or G73 is executed, P and Q cannot specify the same sequence number in one program.

8 For a multiple repetitive canned cycle, no figure can be specified using the direct drawing dimension programming or chamfering/corner R format.

9 Also for G74, G75, and G76, it is impossible to use floating–point input for P and Q. The unit of travel distance and cutting depth is the least input increment.

10 If #1 = 2500 is executed in a custom macro, 2500.000 is assigned to #1, and P#1 is equivalent to P2500.

11 It is impossible to apply tool–nose radius compensation in G71, G72, G73, G74, G75, G76, and G78.

12 A multiple repetitive canned cycle program cannot be used for DNC operation.

13 No interrupt–type custom macro can be used during multiple repetitive canned cycle execution.

14 No multiple repetitive canned cycle can be executed in the advanced preview control mode.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	II.13.1 II.13.2	Canned cycle Multiple repetitive canned cycle
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	II.13.1 II.13.2	Canned cycle Multiple repetitive canned cycle

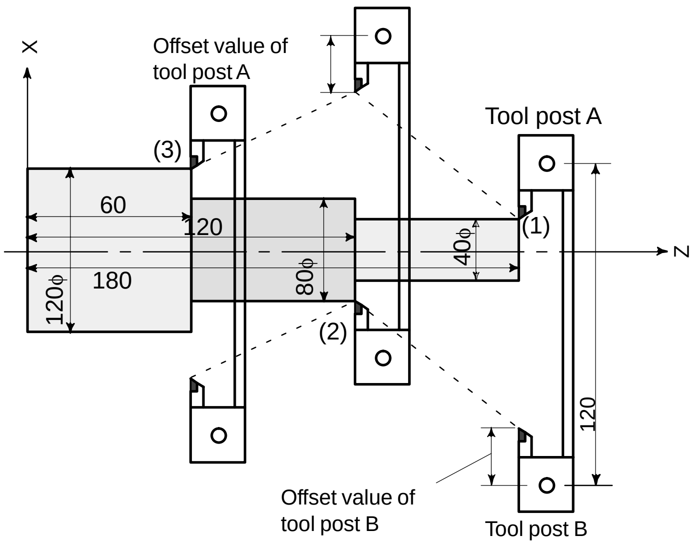
11.9 MIRROR IMAGE FOR DOUBLE TURRETS (T SERIES)

General

Mirror image can be applied to the X–axis with G code.

G68 : Start double turret mirror image
G69 : Mirror image cancel

When G68 is active, the coordinate system is shifted to the other turret, and the X–axis sign is reversed from the programmed command. To use this function, set the distance between the two turrets in a parameter (No. 1290).
Program example for double turrets.



X40.0 Z180.0 T0101 ;	Position turret A at (1)
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position turret B at (2)
G69 ;	Shift the coordinate system by the distance B to A, and cancel mirror image.
X120.0 Z60.0 T0101 ;	Position turret A at (3)

Parameter

- **Distance between two turrets**

1290	Distance between two turrets for mirror image

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 999999999

Set the distance between two turrets for mirror image.

Reference Item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	II.13.6	Mirror image for double turrets

11.10 INDEX TABLE INDEXING FUNCTION (M SERIES)

General

Indexing the table on a machining center is a complished by specifying a positioning angle.
Before and after indexing, the table is automatically unclamped or clamped .

Basic Procedure

The control axis that indexes the table can be named A, B or C. It will be referred to as “B” in the following discussion.

The positioning angle for the table is commanded by the numbers following “B” in the part program block. Both absolute and incremental commands are possible. The value for absolute “B” is based on the increment system.

(Example)	G00G90B100000;	Absolute command (Positioning angle 10 degrees)
	G00G91B20.0;	Incremental command (Move distance 20 degrees)

There are two variations of the procedure (type A and type B) to set the index table position. The difference is in the ON/OFF timing of the position control servo. The sequence of events timing charts and the difference between the variations are described below.

(1) Assume Bbbbb is ordered by the command program.

(2) The CNC turns the B axis unclamp signal BUCLP <F061#0> to “1”. (Type B -- When BUCLP is turned to “ 1 ”, the position control servo for the B axis is turned ON.)

(3) When the unclamp process is completed, the PMC sets the axis unclamp signal *BEUCL <G038#6> to “0”.

(4) The CNC then sets the B axis unclamp signal BUCLP to “0” indicating it received the *BEUCL signal.

(5) When the PMC is notified that BUCLP has been set to “0”, the PMC should set *BEUCL to “1”.
In type B, B–axis unclamp signal BUCLP is set to “0”, B–axis position control is made with servos active, B–axis is rotated, and the B axis is stopped at the specified position. B axis always moves at rapid traverse.

(6) When the B axis stops at the specified position, the CNC sets the B–axis clamp signal BCLP<F061#1>to 1. In type A, signal BCLP is set to “1” and B–axis position control is made with servo off.

(7) When BCLP is set to “1” on the PMC side, the B axis is clamped mechanically (with a clutch or shot pin, for example). When the clamp is completed, the B axis clamp completion signal *BECLP <G038#7> is set to “0”.

(8) When *BECLP is turned to “0”, the CNC then sets BCLP to “0”. (Type B -- When BCLP turns to “0”, the B axis position control servo is turned off.)

(9) On the PMC side, when BCLP changes to “0”, *BECLP is set to “1”. This completes the sequence.

The time charts for these operations are shown in the figures below.

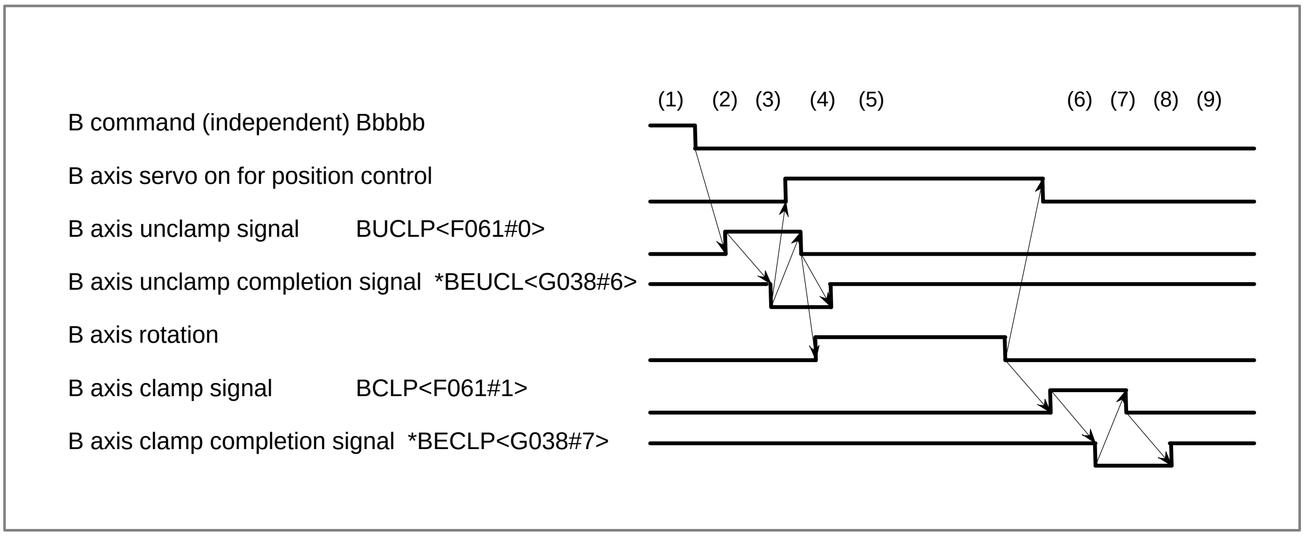


Fig. 11.10 (a) Time chart for positioning index table (type A)

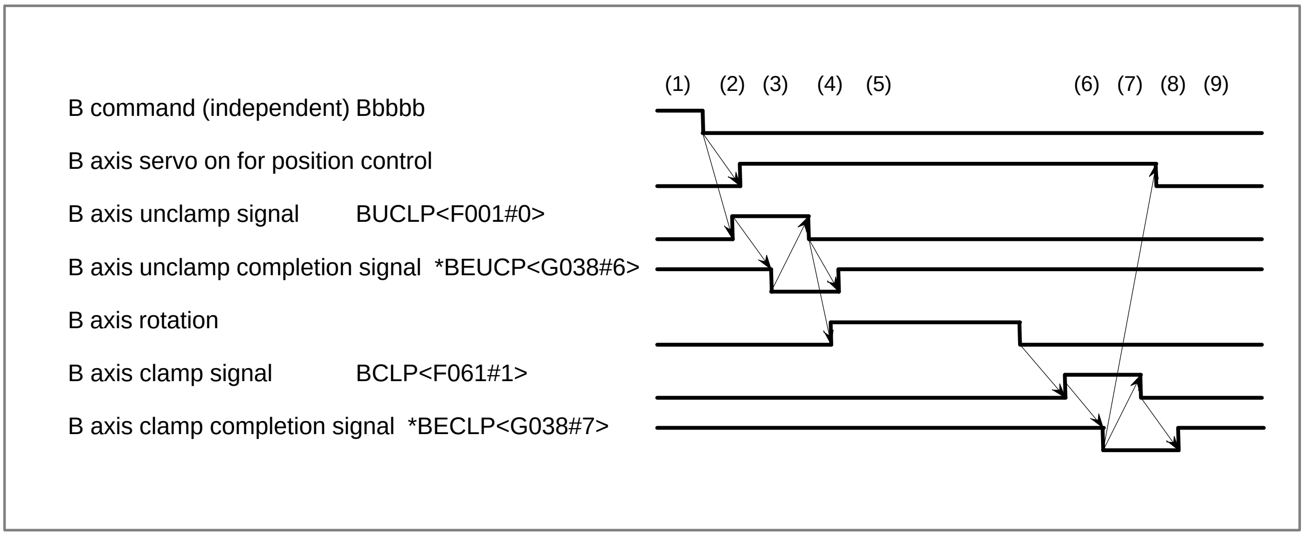


Fig. 11.10 (b) Time chart for positioning index table (type B)

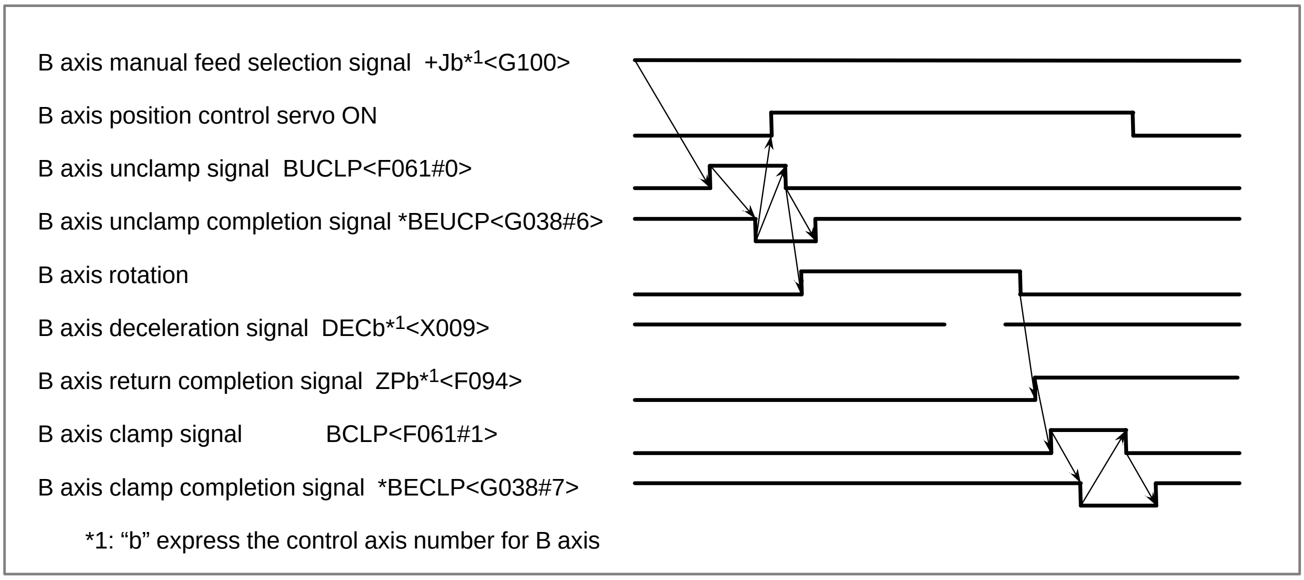


Fig. 11.10 (c) Manual reference position return of B axis (type A)

Type A and Type B

Type A differs from type B in the timing of the servo on/off signal.
Type A is suitable for a system in which the B–axis is clamped with shot pins.
Type B is suitable for a system in which the B–axis is clamped with a clutch.

Minimum indexing angle

When the B–axis is clamped with shot pins, the mechanism can be indexed at only a limited number of positions. The minimum indexing angle can be specified in parameter No. 5512. If an angle is not a multiple of this minimum indexing angle, then alarm No. 135 is issued.

Direction of rotation

The direction of rotation can be set to one of the following.

– The direction with the shorter travel distance (INC, bit 3 of parameter No. 5500)
– Direction specified with a command
– The positive direction. Only when a particular M code is specified in the same block, the axis rotates in the negative direction (parameter No. 5511).

Absolute/incremental programming

Setting G90, bit 4 of parameter No. 5500, specifies absolute programming, and override the G90/G91 G–codes.

Disabling the index table function

The index table function can be temporarily disabled, using the ITI parameter (bit 0 of parameter No. 5501) without turning the power off and on again. When the function is disabled, manual operations (such as jog feed, incremental feed, and handle feed) can be performed on the index table axis even if the index table option is enabled.

Signal

B axis clamp signal BCLP<F061#1>

[Classification] Output signal

[Function] Instructs the PMC side to clamp the B axis mechanically with a clutch or shot pin.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis clamp completion signal *BECLP<G038#7>

[Classification] Input signal

[Function] Notifies the CNC of completion of the B axis clamp operation.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp signal BUCLP<F061#0>

[Classification] Output signal

[Function] Instructs the PMC side to release the B axis from the mechanical clamp.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp completion signal *BEUCP<G038#6>

[Classification] Input signal

[Function] Notifies the CNC of completion of the release of the B axis from the mechanical clamp.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G038	*BECLP	*BEUCP						

	#7	#6	#5	#4	#3	#2	#1	#0
F061							BCLP	BUCLP

Parameter

- **Setting linear or rotation axis**

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

[Data type] Bit axis

ROTx, ROSx Define linear or rotation axis

ROSx	ROTx	Description
0	0	Linear axis · Inch/metric conversion is done. · All coordinate values are linear axis type. (Not rounded in 0 to 360°) · Stored pitch error compensation is linear axis type (Refer to parameter No. 3624)
0	1	Rotation axis (A type) · Inch/metric conversion is not done. · Machine coordinate values are rounded in 0 to 360° Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of the rotation type. (Refer to parameter No. 3624) · Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.
1	0	Setting is invalid (unused)
1	1	Rotation axis (B type) · Inch/metric conversion is not done. · Machine coordinate values is of linear axis type (i.e. not rounded in 0 to 360°). · Rounding of absolute coordinate values and relative coordinate values is decided by parameter No. 1008#0 and #2. · Stored pitch error compensation is of linear axis type (Refer to parameter No. 3624). · Cannot be used with the rotation axes roll over function and the index table indexing function (M series).

- **Setting for positioning the index table**

	#7	#6	#5	#4	#3	#2	#1	#0
5500	IDX	SIM		G90	INC	ABS	REL	DDP

[Data type] Bit

DDP Definition of index values

0 : Conventional method (Example IS–B: B1; = 0.001 deg)
1 : Calculator method (Example IS–B: B1; = 1.000 deg)

REL Relative position display of index table indexing axis

0 : Not rounded by 360 degrees
1 : Rounded by 360 degrees

ABS Displaying absolute coordinate value of index axis

0 : Not rounded by 360 degrees
The index axis rotates 720 degrees (two rotations) when G90 B720.0; is specified from the 0–degree position. It rotates in reverse direction 720 degrees (two rotations) when G90 B0.; is specified. The absolute coordinate value then becomes 0 degree.
1 : Rounded by 360 degrees
The index axis is positioned to 40 degrees when G90 B400.0; is specified from the 0–degree position. The index axis does not rotate by two or more turns when this parameter is set to 1. It also does not move when G90 B720.0; is specified from the 0–degree position.

INC Rotation in the G90 mode. This applies when negative direction M–code (parameter No. 5511) is not set

0 : Not set to the shorter distance around the circumference
1 : Move the shorter distance around the circumference (Set ABS, #2 of parameter No. 5500, to 1.)

G90 Index table indexing command

0 : An absolute/increment command according to the G90/G91 mode
1 : Always an absolute command

SIM When the same block includes a command for an index table indexing axis and a command for another controlled axis:
0 : A P/S alarm (No.136) is issued.
1 : The commands are executed. (In a block other than G00, G28, and G30, however, a P/S alarm (No.136) is issued.)

IDX Index table indexing sequence

0 : Type A
1 : Type B

- **The parameter specifies whether the index table index function is disabled.**

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

[Data type] Bit

ITI The index table index function is:

0 : Enabled.
1 : Disabled.

- **Negative direction rotation command M code**

5511	Negative direction rotation command M code

[Data type] Byte

[Valid data range] 0 to 255

0 : No M code is defined to set the index table rotation to the negative direction. The rotation direction is specified using a command and parameter (INC, #3 of parameter No. 5500).

1 to 255:
Defines an M code to set the index table rotation to the negative direction. The rotation is set to the negative direction only when this M code is specified in the same block as an index table indexing command. If the M code is not specified in the same block, the rotation is always set to the positive direction.

> **NOTE**
> Set ABS (bit 2 of parameter No. 5500) to 1.

- **Unit of index table indexing angle**

5512	Unit of index table indexing angle

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm

[Valid data range] 0 to 360000

This parameter sets the unit of index table indexing angle. The commanded movement must be a multiple of the value entered in this parameter, Other wise, a P/S alarm occur.

> **NOTE**
> If zero is specified as the setting value, any command can be specified.

Alarm and message

Number	Message	Description
135	ILLEGAL ANGLE COMMAND	The commanded index positioning angle was not multiple of the value of the minimum angle. Modify the program.
136	ILLEGAL AXIS COMMAND	In index table indexing , another control axis was commanded with the index axis. Modify the program.

Caution

CAUTION

1 The secondary auxiliary function can be used, but its address must be different from that of the indexing axis.
2 If the incremental command is used for indexing of the index table, the workpiece zero point offset value on the index table axis must always be 0. That is, the machine coordinate system must always agree with the workpiece coordinate system of the index table axis.
3 The dry run signal DRN has no affect during positioning of the B axis.
4 The machine lock signal MLK is functional during positioning of the B axis.

Note

NOTE

1 To specify a rotation axis as the index table indexing axis. (Set the ROTx bit (bit 0 of parameter No. 1006)="1".)
2 The servo off signal for the index table indexing axis is invalid.
3 Single direction positioning (G60) cannot be specified.
4 While the index table is being positioned, input signals that reset the CNC, such as *ESP (emergency stop), ERS (external reset), and RRW (reset & rewind), are functional. When reset is applied to the CNC, indexing stops. Further, if *SP (automatic operation stop signal) turns to "0", axis movement is stopped and the equipment enters the automatic operation stop state.
If a stop at an any position is not suitable for the machine, appropriate processing is required on the machine.
5 If a reset occurs while the system is awaiting the completion of clamping or unclamping, the clamp or unclamp signal is cleared. The CNC exits from the completion wait status.
6 Manual operation of jog feed, incremental feed and handle feed cannot be used with the indexing axis, but manual reference position return is possible. If reset is applied during the movement of the indexing axis, then manual reference position return should be performed.
7 No movement can be performed by automatic return from the reference position (G29), return to the second reference position (G30), or selection of the machine coordinate system (G53).
8 Only the fourth axis can be used as the index table indexing axis.

Reference Item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.13.8	Index table indexing function

11.11 SCALING (M SERIES)

General

A programmed figure can be magnified or reduced (scaling).
The dimensions specified with X_, Y_, and Z_ can each be magnified or reduced with the same or different rates of magnification.
The magnification rate can be specified in the program or by a parameter.

• Scaling up or down along all axes at the same rate of magnification

Least input increment of scaling magnification is: 0.001 or 0.00001 and is set by parameter SCR (No. 5400#07).
The scaled axis is selected by parameter SCLx (bit 0 of parameter No. 5401).
The value in the scaling magnification parameter (No. 5411) is used when not defined in the program.
If X,Y,Z are omitted, the tool position where the G51 command was specified serves as the scaling center.

SCALING UP OR DOWN ALONG ALL AXES AT THE SAME RATE OF MAGNIFICATION

Format	Meaning of command
G51X_Y_Z_P_ ; Scaling start ⋮ Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**X_Y_Z_** : Absolute command for center coordinate value of scaling **P_** : Scaling magnification

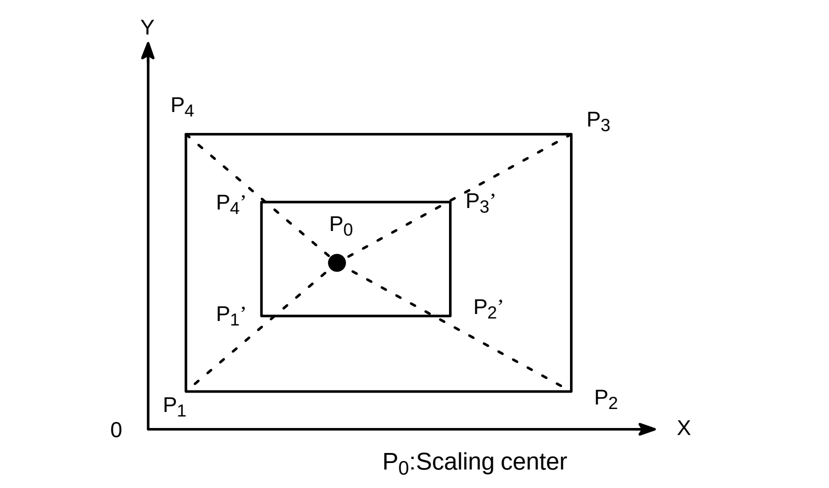


Fig. 11.11 (a) Scaling (P_1 P_2 P_3 P_4→P_1'P_2'P_3'P_4')

- **Scaling of each axis, programmable mirror image (negative magnification)**

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter XSC (No. 5400#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 5401#0) to enable scaling along each axis.
Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001(set parameter SCR (No. 5400#7)).
Magnification is set in parameter 5421 within the range ±0.00001 to ±9.99999 or ±0.001 to ±9.999.
If a negative value is set, mirror image is applied.
If magnification I, J or K is not commanded, a magnification value set to parameter (No. 5421) is used. This value must be greater than 0.

SCALING ALONG EACH AXES AT A DIFFERENT RATE OF MAGNIFICATION (MIRROR IMAGE)	
Format	Meaning of command
G51_X_Y_Z_I_J_K_; Scaling start Scaling is effective. (Scaling mode) **G50** Scaling cancel	**X_Y_Z_:** Absolute command for center coordinate value of scaling **I_J_K_:** Scaling magnification for X axis, Y axis and Z axis respectively

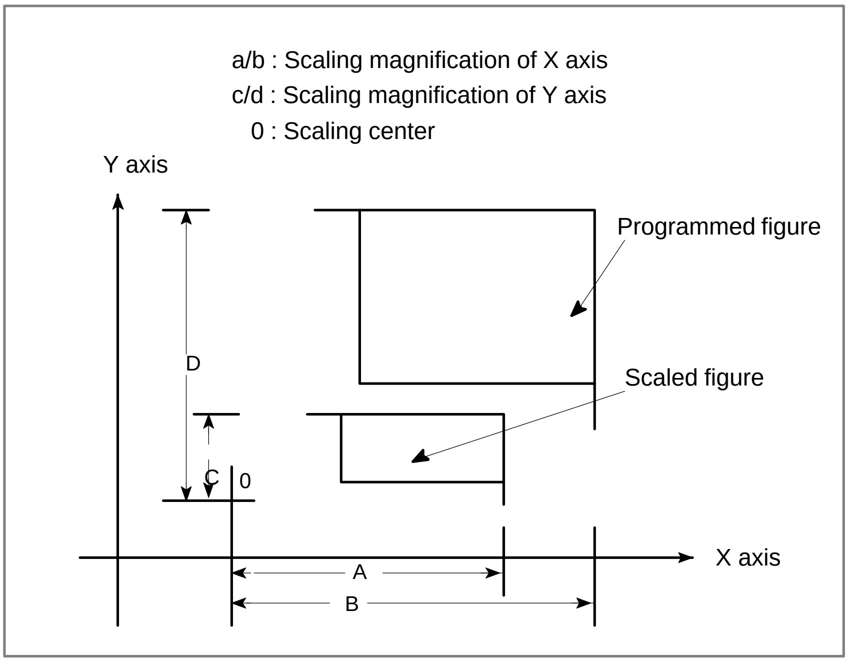


Fig. 11.11 (b) Scaling of each axis

Parameter

• Setting valid/invalid and magnification of scaling

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC						

[Data type] Bit

XSC Axis scaling and programmable mirror image

0 : Not active (The scaling magnification is specified by P.)
1 : Active

SCR Scaling magnification unit

0 : 0.00001 times (1/100,000)
1 : 0.001 times

• Valid/invalid setting to each axis scaling

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Data type] Bit axis

SCLx Scaling for every axis

0 : Not active
1 : Active

• Default scaling magnification value

5411	Default scaling magnification value

This parameter can be set at "Setting Screen."

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] 1 to 999999

This parameter sets the scaling magnification. This setting value is used when a scaling magnification (P) is not specified in the program.

> **NOTE**
> Parameter No. 5421 becomes valid when scaling for every axis is valid. (XSC, #6 of parameter No. 5400 is "1".)

- **Scaling magnification for every axis**

5421	Scaling magnification for every axis

This parameter can be set at “Setting Screen.”

[Data type] Two–word axis

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] −999999 to −1, 1 to 999999

This parameter sets the scaling magnification for every axis.

Alarm and message

Number	Message	Description
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 P_p, or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.14.7	Scaling (G50, G51)
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.14.7	Scaling (G50, G51)

11.12 COORDINATE SYSTEM ROTATION

General

A programmed shape can be rotated. By using this function it is possible to modify a program using a rotation command. This is useful when a workpiece has been placed with some angle rotated from the programmed position on the machine. This is also useful when machining the same pattern.

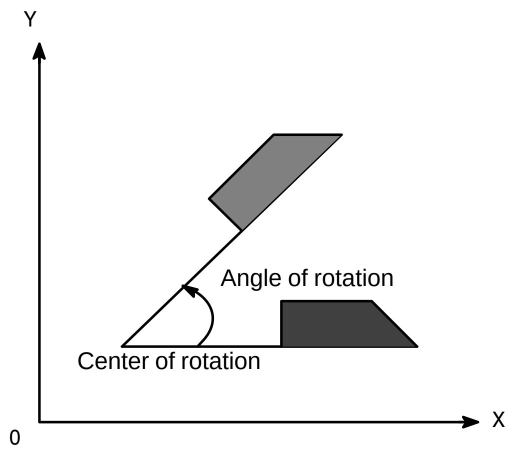


Fig. 11.12 (a) Coordinate system rotation

FORMAT

{**G17** / **G18** / **G19**} **G68 (G68.1)**α_β_**R_** ; Start rotation of a coordinate system.

Coordinate system rotation mode
(The coordinate system is rotated.)

G69 (G69.1); Coordinate system rotation cancel command

Note:G68/G69 for M series, G68.1/G69.1 for T series.

MEANING OF COMMAND

G17 (G18 or G19)	: Select the plane that contains the figure to be rotated.
α_β_	Absolute command for two of the x_,y_,and Z_ axes that correspond to the current plane selected. The command specifies the coordinates of the center of rotation.
R_	A positive angular displacement indicates counter clockwise rotation. Parameter 5400#0 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg
Valid data range : –360.000 to 360.000

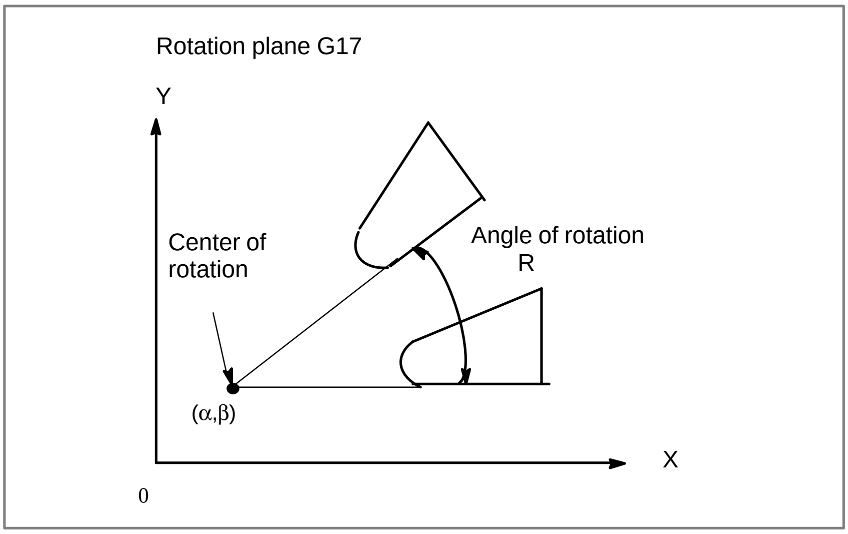


Fig. 11.12 (b) Coordinate system rotation

Parameter

- **Angle specification method of coordinate system rotation**

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Data type] Bit

RIN Coordinate rotation angle command (R)

0 : Specified by an absolute method
1 : Specified by G90 or G91

- **Angular displacement used when no angular displacement is specified for coordinate system rotation**

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation

[Data type] Two–word

[Unit of data] 0.001 degrees

[Valid data range] –360000 to 360000

This parameter sets the angular displacement for coordinate system rotation. When the angular displacement is not specified with address R in the G68 block, this parameter value is used as the angular displacement.

Alarm and message

Number	Message	Description
144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
5302	ILLEGAL COMMAND IN G68 MODE	A command to set the coordinate system is specified in the coordinate system rotation mode.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.14.8	Coordinate system rotation (G68, G69)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.14.8	Coordinate system rotation (G68, G69)

11.13 MACRO COMPILER/ EXECUTER

General

There are two types of NC programs; those which, rarely changed, and those that changed with every execution. These programs are created by the custom macro.
The custom macro is an executable form of a part program created by a machine tool builder that is stored in the CNC's flash ROM. It is executed from the flash ROM.

Features

(1) Since the program is stored in executable form program, the execution speed is high. Machine time is then reduced, and precision is improved.
(2) Since the program is stored in FLASH ROM, there is no lost of data of battery failure or corruption. Reliability is improved.
(3) The stored program is not displayed on a program screen.
(4) The custom macro is stored in FLASH ROM.
(5) The user can call the macro without knowing the stored program. A custom macro can be created and executed in the program edit memory.
(6) An original screen can be created by using the graphic display or by selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machine program creation and edit control, reader/punch interface control, and PMC data read/write functions.

Note

NOTE
When the macro executor is active, the order-made macro cannot be specified.

Reference item

Macro compiler/executer programming manual (B–61803E–1)

11.14 SMALL HOLE PECK DRILLING CYCLE (M SERIES)

General

In peck drilling, the tool enters and retracts from hole based on an overload torque detection signal (skip signal) until the designed hole depth is reached. The spindle speed and cutting feedrate are changed on each entry of the hole.

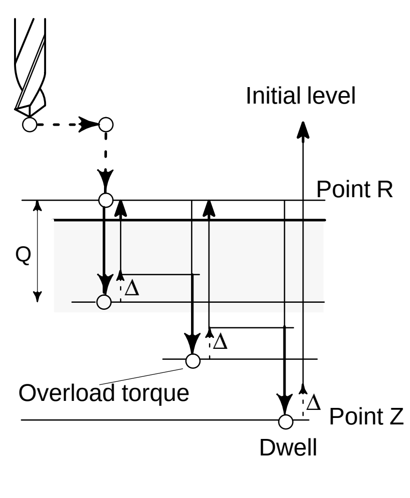
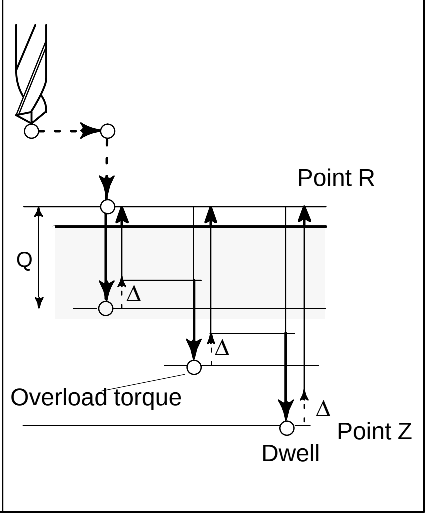
The cycle is realized by:

* X– and Y–axis positioning
* Positioning at point R along the Z–axis
* Cutting along the Z–axis (first time, depth of cut Q, incremental)

Repeated until point Z is reached:

Retracting
(bottom of hole → minimum clearance Δ , incremental)
Retracting
(→ to point R, absolute)
Forwarding
(point R → to point with hole bottom + clearance Δ, absolute)
Cutting
(second and subsequent times, cut of depth Q + Δ, incremental)

* Dwell
* Return to point R along the Z–axis (or initial point) = end of cycle

G83(**G98**)	**G83**(**G99**)
	

Change of cutting conditions

The cutting conditions are changed at each pecking operation (forwarding → cutting → retracting) during one G83 cycle. Bits 1 and 2 of parameter No. 5160 affects the cutting conditions.

- **Changing of cutting feedrate**

The cutting feedrate programmed with the F word is changed during the second and subsequent cutting operations. Parameter Nos. 5166 and 5167 specify the ratio of change for the case in which a skip signal was received during the previous cutting operation and the case in which no skip signal was received during the previous cutting operation, respectively.

$$\boxed{\text{Cutting feed rate} = F \times \alpha}$$

(First time) $\alpha = 1.0$

(Second and subsequent times) $\alpha = \alpha \times \beta \div 100$
(where β is the ratio of change for the first time)

Skip during the previous cutting: β = b1% (parameter No. 5166)
No skip during the previous cutting: β = b2% (parameter No. 5167)

The feedrate is not changed when the ratio α becomes less than the ratio specified in parameter No. 5168. The upper limit to the newly specified cutting feedrate is the maximum cutting feedrate.

- **Changing of spindle speed**

The spindle speed programmed with the S word is changed during the second and subsequent cutting operations. Parameter Nos. 5164 and 5165 specify the ratio of change in which a skip signal was received during the previous cutting operation and in which no skip signal was received during the previous cutting operation.

$$\boxed{\text{Spindle speed} = S \times \gamma}$$

(First time) $\gamma = 1.0$

(Second and subsequent times) $\gamma = \gamma \times \delta \div 100$
(where δ is the ratio of change for the first time)

Skip during the previous cutting: δ = d1% (parameter No. 5164)
No skip during the previous cutting: δ = d2% (parameter No. 5165)

If the spindle speed is set at the lower limit, then it is not changed. The upper limit to the newly specified spindle speed is the maximum S analog data.

Signal

Overload torque signal SKIP<X004#7>

[Classification] Input signal

[Function] Retracts a tool if an overload torque occurs.

[Operation] When this signal becomes “1”, the control unit operates as follows:

- Retracts the tool, and changes the spindle speed and cutting feedrate, then continue machining.
- This signal is valid, when the drill axis is between points R and Z and is moving forward or cutting.

NOTE
This signal is used also as a skip signal. (See Section 14.3.)

Small–diameter peck drilling in progress signal PECK2<F066#5>

[Classification] Output signal

[Function] Indicates whether small–diameter peck drilling is in progress.

[Output condition] This signal becomes “1” under the following conditions.

- When the tool returns to R point/initial level, after entering the hole.
- The signal does not become “1” during positioning at the hole position.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP							

	#7	#6	#5	#4	#3	#2	#1	#0
F066			PECK2					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

[Data type] Bit

OLS When an overload torque signal is received in a peck drilling cycle, the feed and spindle speed are

0 : Not changed.
1 : Changed.

NOL When the the depth of cut is reached, the feed and spindle speed are:

0 : Not changed.
1 : Changed.

5163	M code specifying the peck drilling cycle mode

[Data type] Two–word

[Valid data range] 1 to 99999999

This parameter defines the M–code for the peck drilling cycle.

5164	Percentage of the spindle speed to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 255

This parameter sets the percentage of the spindle speed change caused by the overload torque signal when the tool is retracted during a peck drilling cycle.

S2 = S1 × d1 ÷ 100
S1: Spindle speed to be chaged
S2: Spindle speed changed
d1 is set as a percentage.

5165	Percentage of the spindle speed to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 255

This parameter sets the percentage of the spindle speed change during a peck drilling cycle.

S2 = S1 × d2 ÷ 100
S1: Spindle speed to be chaged
S2: Spindle speed changed
d2 is set as a percentage.

5166	Percentage of cutting feedrate to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 255

This parameter sets the percentage of the cutting feedrate change caused by the overload torque signal during a peck drilling cycle.

F2 = F1 × b1 ÷ 100
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b1 is set as a percentage.

5167	Percentage of the cutting feedrate to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 255

This parameter sets the percentage of the cutting feedrate change during a peck drilling cycle.

F2 = F1 × b2 ÷ 100
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b2 is set as a percentage.

5168	Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 255

This parameter sets the lower limit of the percentage of feedrate change for in a peck drilling cycle.

FL = F × b3 ÷ 100
F: Specified cutting feedrate
FL: Changed cutting feedrate
Set b3 as a percentage.

5170	Macro variable where the total number of retractions is recorded

[Data type] Word

[Valid data range] 100 to 149

This parameter identifies the macro variable where the total number of times the tool is retracted is recorded. The macro variables cannot be variables 500 to 599.

5171	Macro variable to which the total umber of retractions due to an overload signal is recorded

[Data type] Word

[Valid data range] 100 to 149

This parameter identifies the custom macro variable where the number of times the tool is retracted after the overload signal is recorded. The macro variable cannot be variables 500 to 599.

5172	Speed of retraction to point R when no I address is defined

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 4000

This parameter sets the speed of retraction to point R when no I address is defined.

5173	Speed advancing to the bottom of a hole when no I address is defined

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 4000

This parameter sets the speed for advancing to the bottom of a previously machined hole when no address I is defined.

5174	Clearance in a peck drilling cycle

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter input)	0.01	0.001	0.0001	mm
Linear axis (inch input)	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the clearance in a peck drilling cycle.

Alarm and message

• Diagnostic display

520	Total number of retract operations during cutting since G83 was issued
521	Total number of retract operations due to reception of the overload torque signal since G83 was issued

The indications of Nos. 520 and 521 are cleared by G80.

522	Coordinates at which the drill axis started retracting (least input command)
523	Difference between the previous and current coordinates at which the drill axis started retracting (least input increment: previous – current)

Warning

> **WARNING**
> Forwarding or retracting is not performed by rapid traverse positioning. Instead, it is performed with the same interpolation as for cutting feed. This means exponential acceleration/deceleration is performed. However, the tool life management function does not count the tool life during forwarding or retracting.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II. 13.1.7	Small hole peck drilling cycle
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II. 13.1.7	Small hole peck drilling cycle

12 DISPLAY/SET/EDIT

12.1 DISPLAY/SET

12.1.1 Clock Function

General

Time is displayed in the hour/minute/second format on each display screen. Some screens allows display of the year, month, and day.
The custom macro system variable can be used to read the time.

Time information can be read and written.

System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 2147483648 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to decimal number. For example, Janualy 23, 2004 is represented as 20040123.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.8	Displaying and Setting Run Time,Parts Count, and Time

12.1.2 Displaying Operation History

General

This function displays a history of the key stroke and signal operations, performed by the CNC operator, when a failure or CNC alarm occurs.

The following history data is recorded:

(1) MDI key operation sequences
Example: A to Z, <POS>, <PAGE ↑>, [SF1]

(2) On/off status transitions of selected input and output signals
Example: G0000.7↑ , SBK ↑

(3) CNC alarm information
Example: P/S0010

(4) Time (date, time) stamp
Example: 04/01/20
09:15:30

The history data can be read or written to an input/output device.

Operation history signal selection parameter

Setting bit 4 (OHS) of parameter No. 3206 to 1 enables operation history signals to be selected as parameters and signal selection data to be input/output (read/punched) as parameters. In this case, selecting/deselecting these signals on the operation history signal selection screen causes the corresponding parameter values to change automatically. Changing the parameter values also changes the displays on the operation history signal selection screen.

Signal selection data and corresponding parameters

The input/output signals selected as operation history targets are displayed on the ”operation history signal selection screen.” Input/output signals can be selected as operation history targets on the ”operation history signal selection screen” and, if bit 4 of parameter No. 3206 is 1, they can specified directly using parameters. Up to 20 addresses can be used for signal selection. Selecting one signal uses three types of data, that is, a signal type (X, G, F, or Y), address, and bit. These data types correspond to parameters as listed below.

Table 12.1.2(a) Signal selection data and corresponding parameters

No.	Signal type	Address	Bit
01	No.12801	No.12841	No.12881
02	No.12802	No.12842	No.12882
03	No.12803	No.12843	No.12883
…	…	…	…
20	No.12820	No.12860	No.12900

The following table lists parameter values for respective signal types.

Table 12.1.2(b) Signal types and corresponding parameter values

Signal type	Parameter value
Not selected	0
G0000 to G0255	1
F0000 to F0255	3
Y0000 to Y0127	5
X0000 to X0127	6

Method of selecting signals based on parameters

The following explains how to use parameters to select signals as operation history targets.

Procedure

1 Set bit 4 of parameter No. 3206 to 1. The current operation history signal selection data is reflected to parameters.

2 Determine which parameter No. to be set while referencing Table 1.

3 Select or deselect signal types (from parameter Nos. 12801 to 12820) while referencing Table 2. To deselect a signal type, reset the related parameter to 0 (the address and bit combined with the parameter that is reset are reset simultaneously with that parameter). When a signal type is selected, the parameter for setting the address combined with that signal type is set to a minimum possible value, and the parameter for setting the bit combined with that signal type is initialized to 0. If an attempt is made to specify an invalid value, the warning message ”Data out of range” is displayed; so, you should retry.

4 Set up an address (parameter Nos. 12841 to 12860). Setting up an address causes the parameter for setting the bit combined with that address to be initialized to 0. If an attempt is made to set an invalid value, or the parameter for setting the parameter type combined with that address is 0, the warning message ”Data out of range” is displayed; so, you should retry.

5 Set or reset bits (Nos. 12881 to 12900). To select a signal as a history target, set the related bit to 1. Otherwise, reset it to 0. If the parameter for setting a signal type combined with that bit is 0, the warning message ”Data out of range” is displayed; so, you should retry.

6 To continue signal selection, repeat steps 2 to 5.

Example: To select the automatic operation start signal (G7.2) as operation history target No. 2, set up the related parameters as listed below:

No.12802=1
No.12842=7
No.12882=00000100
If parameter No. 12802 is 1, parameter Nos. 12842 and 12882 are initialized to 0. If parameter No. 12842 is 7, parameter No. 12882 is initialized to 0. Be sure to set parameter Nos. 12802, 12842, and 12882 in the stated sequence.

Caution

CAUTION
A parameter clear operation (turning the power on with the <RESET> key held) down does not deselect signals from operation history targets. To deselect them, perform a release operation ([DELET] or [ALLDEL] soft keys) on the "operation history signal selection screen" or, when bit 4 of parameter No. 3206 is 1, reset parameter Nos. 12801 to 12820 to 0.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3106	OHS			OHD				

[Data type] Bit

OHD The operation history screen is:
0 : Not displayed.
1 : Displayed.

OHS Operation history sampling is:
0 : Performed.
1 : Not performed.

NOTE
Normally, set 0 (sampling is performed).

	#7	#6	#5	#4	#3	#2	#1	#0
3112			OPH					

NOTE
CNC power must be cycled for this perameter to take effect.

OPH Enable the operation history function.
0 : Enabled.
1 : Disabled.

3122	Time interval used to record data in the operation history

[Data type] Word

[Unit of data] Minutes

[Valid data range] 0 to 1439

Time data is recorded in the operation history at set intervals. When 0 is specified in this parameter, 10 minutes is assumed as the default.

	#7	#6	#5	#4	#3	#2	#1	#0
3206				OHS				

[Data type] Bit

OHS 0 : Operation history signal selection is not linked to parameters. Signals are selected as or deselected from operation history targets on the operation history signal selection screen. Changing any parameter does not affect operation history signal selection.
1 : Operation history signal selection is linked to parameters. Operation history signal selection can be made both on the operation history signal selection screen and through parameters.

> **NOTE**
> Setting this parameter to 1 causes the current operation history signal selection data to be reflected to parameter Nos. 12801 to 12900.

12801	Operation history signal selection signal type (No. 01)
12802	Operation history signal selection signal type (No. 02)
12803	Operation history signal selection signal type (No. 03)
:	:
12820	Operation history signal selection signal type (No. 20)

[Data type] Byte

[Valid data range] 0 to 10

Specify signal types for operation history signal selection. The following table lists the signal types and corresponding parameter values.

Signal type	Parameter value
Not selected	0
G0000 to G0255	1
F0000 to F0255	3
Y0000 to Y0127	5
X0000 to X0127	6

To deselect a signal type, reset the related parameter to 0 (the address and bit combined with the parameter that is reset are reset simultaneously with that parameter). When a signal type is specified, the parameter for setting the address to be combined and the parameter for setting the bit are initialized to 0.

Example) If parameter No. 12801 is 3, parameter Nos. 12841 and 12881 are initialized, respectively, to 0 and 00000000.
If an attempt is made to specify an invalid value, the warning message ”Data out of range” is displayed; so, you should retry.

12841	Operation history signal selection address (No. 01)
12842	Operation history signal selection address (No. 02)
12843	Operation history signal selection address (No. 03)
:	:
12860	Operation history signal selection address (No. 20)

[Data type] Word

[Valid data range] 0 to 255

Set up addresses for operation history signal selection. Setting up an address causes the parameter for setting the bit combined with that address to be initialized to 0. If an attempt is made to set an invalid value, or the parameter for setting the parameter type combined with that address is 0, the warning message ”Data out of range” is displayed; so, you should retry.

	#7	#6	#5	#4	#3	#2	#1	#0	
12881									(No. 01)
12882									(No. 02)
12883									(No. 03)
:				:					
12900									(No. 20)

[Data type] Bit

Set or reset bits for operation history signal selection. To select a signal as a history target, set the related bit to 1. Otherwise, reset it to 0. If the parameter for setting a signal type combined with that bit is 0, the warning message ”Data out of range” is displayed; so, you should retry.

Note

NOTE

1 While the operation history screen is displayed, history data is not recorded.
2 When the duration of the on/off state of an input signal is 16 msec or shorter, that state is not recorded in the history. In addition, some signals are not recorded.
3 Approximately 8000 keystrokes can be stored in the memory.
4 Recorded history data is held in memory even after the power is turned off. Clearing memory will erase the history data.
5 For the operation history function, sampling is disabled when bit 7 (OHS) of parameter No. 3106 is set to 1.
6 Be careful to set the date and time correctly.
If no data exists in the regular recording interval, then this time period is not recorded.

12.1.3 Help Function

General

The help function displays on the screen detailed information about alarms issued in the CNC and about CNC operations.

- **Detailed information of alarms**

The help screen displays detailed information about the alarms and how to recover from them. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

The help screen displays a list of parameter Nos. for each function, as an aid.

Note

NOTE

The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.13	HELP FUNCTION
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.13	HELP FUNCTION
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.13	HELP FUNCTION
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.13	HELP FUNCTION

12.1.4 Displaying Alarm History

General

Up to 50 of the most recent CNC alarms are stored and displayed on the screen.

The following items are displayed for each alarm easily.

(1) The date the alarm was issued

(2) Alarm No.

(3) Alarm message (some contains no message)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.7.2	ALARM HISTORY DISPLAY
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.7.2	ALARM HISTORY DISPLAY
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.7.2	ALARM HISTORY DISPLAY
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.7.2	ALARM HISTORY DISPLAY

12.1.5 Servo Tuning Screen

General

On the servo tuning screen, parameters for each axis are listed for the basic adjustment of the servo motor.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111								SVS

[Data type] Bit

SVS Servo tuning screen

0 : Not displayed
1 : Displayed

Reference item

Series 0*i*–C/0*i* Mate–C	MAINTENANCE MANUAL (B–64115EN)	Servo Tuning Screen

12.1.6 Spindle Setting and Tuning Screen

General

On the spindle setting and tuning screen, parameters are listed for the basic adjustment of the serial spindle. The screen is only for the main spindle connected to the first amplifier.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111						SVP	SPS	

[Data type] Bit type

SPS Spindle setting and tuning screen

0 : Not displayed
1 : Displayed

SVP Display of spindle data

0 : Instantaneous values are displayed.
1 : Peak–hold values are displayed.

Reference item

Series 0*i*–C/0*i* Mate–C	MAINTENANCE MANUAL (B–64115EN)	Spindle setting and tuning screen

12.1.7 Waveform Diagnosis Display

General

Waveform diagnosis is classified into two main types.

(1) One–shot type

One–shot waveform diagnosis provides graphs of waveforms.

In one–shot waveform diagnosis, the start of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the adjustment of servo motors and spindle motors, and includes the following.

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data
b. Combined feedrate for the first, second, and third axes
c. Spindle motor speed, load meter reading, and position deviation, converted to spindle position
d. On/off status of the machine signal, specified by a signal address

(2) Stored type

In stored waveform diagnosis, changes in the following data are recorded. When a servo alarm is issued, the recorded data can be read and displayed graphically as waveforms.

The end of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the determination of a fault.

Stored data can be output via the reader/punch interface, and includes:

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112								SGD

[Data type] Bit

SGD Servo waveform

0 : Not displayed
1 : Displayed

> **NOTE**
> When this parameter is set, CNC power must be cycled.

3120	Time from the output of an alarm to the termination of sampling

[Data type] Word

[Unit of data] ms

[Valid data range] 1 to 32760

When the waveform diagnosis function is used, this parameter sets the time form the output of a servo alarm until data collection. Storage operation is stopped because of the alarm. (This means that the termination of data collection can be delayed by a specified time.)

3121	Store–type waveform diagnosis data select

[Data type] Byte

[Valid data range] 0 to 1

The six types of sampling data in store–type waveform diagnosis are:

0 : Thermal simulation data.
1 : Spindle load meter.

Note

NOTE

1 Once the memory area becomes full, the oldest data is deleted to allow new data.
2 Waveform diagnosis data is held in memory even after the power is turned off.
3 Be careful to set the date and time correctly.
4 Waveform diagnosis is enabled when bit 0 (SGD) of parameter No. 3112 is set to 1.
Usual graphic display function can not be used when waveform diagnosis is enable.

Reference item

Series 0*i*–C/0*i* Mate–C	MAINTENANCE MANUAL (B–64115EN)	Waveform diagnostic display

12.1.8 Self–diagnosis

General

To determine the cause of an alarm, check the following.

First, it has to be determined if the breakdown is in the CNC, the PMC or the machine.

The CNC checks the following.

1) Abnormality of detection system
2) Abnormality of position control unit
3) Abnormality of servo system
4) Overheat
5) Abnormality of CPU
6) Abnormality of ROM
7) Abnormality of RAM
8) Abnormality in data transfer between MDI
9) Abnormality of part program storage memory
10) Abnormality in tape reader read function
11) Abnormality in data transfer between PMC

Input/output signals between the PMC and the CNC, and the CNC status can be displayed on the screen.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

12.1.9 Display of Hardware and Software Configuration

General

The required hardware/software configuration for the CNC can be displayed on the screen.

The system configuration screen displays the following information:

(1) Printed circuit board configuration

a. The type and function of the printed circuit board mounted in each slot.

b. When a CPU is mounted on a printed circuit board, the software series and edition are displayed.

(2) Software configuration

The series and editions of installed software, including the CNC software, servo software, PMC management software, and ladder programs, are displayed. For the CNC software, assembly information is also displayed.

(3) Module configuration

The configuration of the modules or hardware mounted on each printed circuit board (such as the type of a module, and where it is mounted) is displayed.

Reference item

Series 0*i*–C/0*i* Mate–C	MAINTENANCE MANUAL (B–64115EN)	System Configuration Screen

12.1.10 Position Display Neglect

General

Disabling the current position display is accomplished by setting bit 0 (NDPx) of parameter No. 3115.

Bit 1 (NDAx) of parameter No. 3115 enables the display of positions in the machine coordinate system.

Signal

Position Display Neglect Signal NPOS1 to NPOS4 <G198#0 to #3>

[Classification] Input signal

[Function] Disables the display of the current position.
A separate signal is provided for each controlled axis. The number at the end of each signal name denotes the controlled axis number.

NPOS1

1···1st axis current position display
2···2nd axis current position display
3···3rd axis current position display
4···4th axis current position display

[Operation] When a signal is set to 0, the current position in the corresponding axis is displayed. When the signal is set to 1, the current position and its corresponding axis is not displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G198					NPOS4	NPOS3	NPOS2	NPOS1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Data type] Bit axis

NDPx Display of the current position for each axis

0 : The current position is displayed.
1 : The current position is not displayed.

NDAx Position display using absolute coordinates and relative coordinates is:

0 : Performed.
1 : Not performed. (Machine coordinates are displayed.)

12.1.11 Run Hour and Parts Count Display

General

This function displays the integrated power–on time, the integrated cycle operation time, the integrated cutting time and timer (started by an input signal from PMC) on the screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, by the operator.

In addition to the above, this function displays the count of the total number of parts machined, the number of parts required and the number of completed parts on the screen. Each time M02, M30 or a parameter set M code is executed, the count of the total number of parts machined and the number of parts completed is incremented by 1.

If a program is prepared so as to execute M02, M30 or a parameter set M code each time one part machining is completed, the number of parts machined can be counted automatically.

If the count of the number of parts machined reaches the number of parts required, a signal is output to the PMC side.

It is possible for the operator to change and preset the number of parts required and the number of parts completed.

Signal

Target part count reached signal PRTSF<F062#7>

[Classification] Output signal

[Function] Reports to the PMC that the specified number of parts have been machined.

[Output condition] The PRTSF signal is set to 1 when:

- Machining of the specified number of parts has been completed.
  When the required number of parts is zero, this signal is not set.

The PRTSF signal is set to 0 when:

- Machining of the specified number of parts has not yet been completed.
- The system is reset.

General–purpose integrating meter start signal TMRON <G053#0>

[Classification] Input signal

[Function] The CNC has an meter which is started by an input signal from the PMC. Additionally, there are meters for counting the automatic operation time and counting cutting time. The count for these meter can be displayed on the screen. The count can be preset by the operator.

[Operation] When the signal is set to 1, the meter starts counting.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053								TMRON

	#7	#6	#5	#4	#3	#2	#1	#0
F062	PRTSF							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Data type] Bit

PCM M code that counts the total number of machined parts and the number of completed parts

0 : M02, or M30, or an M code specified by parameter No. 6710
1 : Use only the M code specified by parameter No. 6710 and not M02 or M30

6710	M code that counts the total number of machined parts and the number of completed parts

[Data type] Byte

[Valid data range] 0 to 255 except 98 and 99

The total number of machined parts and the number of completed parts are counted (+1) when the M code set is executed.

> **NOTE**
> A value of 0 is invalid (the number of parts is not counted for M00). Data 98 and 99 cannot be set.

6711	Number of machined parts

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

The number of machined parts is counted (+1) together with the total number of completed parts when the M02, M30, or a M code specified by parameter No. 6710 is executed.

> **NOTE**
> When bit 0 (PCM) of parameter No. 6700 is set to 1, the number of parts is not counted with M02 and M30.

6712	Total number of machined parts

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

This parameter sets the total number of machined parts.

The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No. 6710 is executed.

> **NOTE**
> When bit 0 (PCM) of parameter No. 6700 is set to 1, the number of parts is not counted with M02 and M30.

6713	Number of required parts

This parameter can be entered on the setting screen.

[Data type] Word

[Unit of data] One piece

[Valid data range] 0 to 9999

This parameter sets the number of required parts.

Required parts finish signal PRTSF is output to the PMC when the number of machined parts reaches the number of required parts. The number of parts is regarded as infinity when the number of required parts is zero. The PRTSF signal is then not output.

6750	Integrated value of power–on period

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the accumulated time when the machine is powered.

6751	Operation time (integrated value of time during automatic operation)

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6752	Operation time (integrated value of time during automatic operation)

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the accumulated time during automatic operation (neither stop nor hold time included).

The actual operation time is the sum of the values set in parameter Nos. 6751 and 6752.

6753	Accumulated cutting time

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6754	Accumulated cutting time

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the accumulated cutting time that is performed in machine feed such as linear interpolation (G01) and circular interpolation (G02 or G03).

The actual cutting time is the sum of the values set in parameter Nos. 6753 and 6754.

6755	Accumulated time of meter drive signal (TMRON) ON

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6756	Accumulated time of meter drive signal (TMRON) ON

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter is accumulated time while input PMC signal TMRON is on.

The actual accumulated time is the sum of the values set in parameter Nos. 6755 and 6756.

6757	Operation time (accumulated automatic operation time)

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6758	Operation time (accumulated automatic operation time)

This parameter can be entered on the setting screen.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter is the automatic operation drive time (neither stop nor hold state included). The actual operation time is the sum of the values set in parameter Nos. 6757 and 6758. The operation time is automatically preset to 0 during the power–on sequence.

Reference item

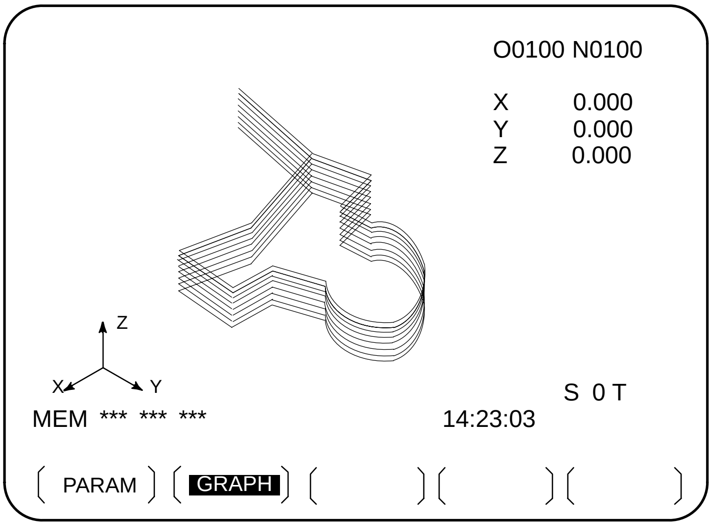
Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.8	Displaying and Setting Run Time,Parts Count, and Time

12.1.12 Graphic Display/ Dynamic Graphic Display

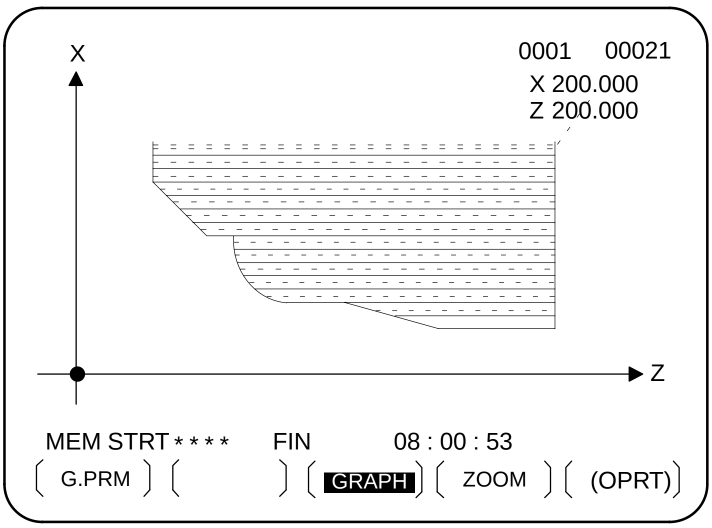
General

Graphic Display

It is possible to draw the programmed tool path on the screen, and check the progress of machining.
In addition, it is also possible to enlarge/reduce the drawing.
The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.



M series



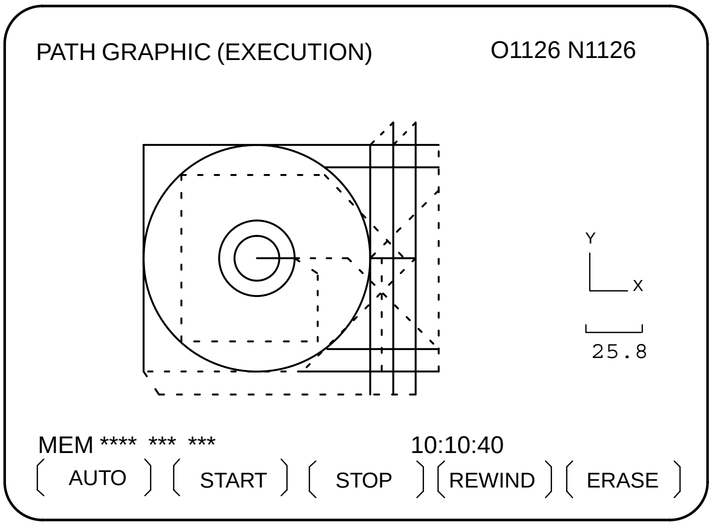
T series

Dynamic graphic display

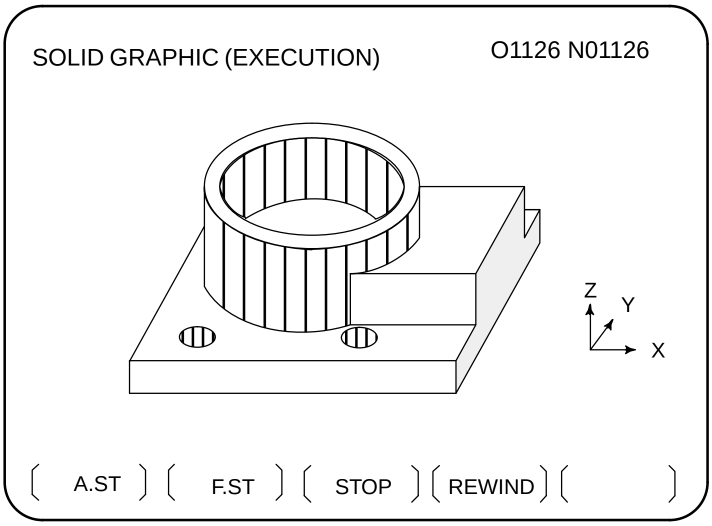
There are two functions of Dynamic Graphics.

Path graphic	This is used to draw the path of the tool center.
Solid graphic	This is used to draw the workpiece figure machined by the tool movement. The T series does not have this function.

The path graphic function is used to precisely check the part program. The solid graphic function is used to draw the workpiece figure to be machined by a program. Switching between the two functions is possible.



Tool path Graph



Part Machined

Signal

Check drawing–under–way signal CKGRP <F062#5>

[Classification] Output signal

[Function] Indicates that a dynamic graphics display (animated graphics display) is being drawn for check purposes.

[Operation] The signal becomes 1 when:

- Check drawing begins.

The signal becomes 0 when:

- A reset occurs after check drawing has ended.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F062			CKGRP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003	MVG							

[Data type] Bit

MVG While drawing using the dynamic graphics function (with no machine movement), the axis–in–movement signal is:

0 : Output
1 : Not output

> **NOTE**
> In case of M series the signal is not output.

	#7	#6	#5	#4	#3	#2	#1	#0
6500		NZM			DPA			
			DPO					

[Data type] Bit

DPA Current position display on the graphic display screen

0 : Displays the actual position with tool nose radius compensation
1 : Displays the programmed position

DPO Current position on the workpiece drawing or tool path drawing screen

0 : Is not displayed
1 : Displayed

NZM 0 : Screen image enlargement by a conventional method is enabled.
1 : The screen image is enlarged by specifying the center of the screen and the magnification. (Screen image enlargement by the conventional method is disabled.)

	#7	#6	#5	#4	#3	#2	#1	#0
6501			CSR					
			CSR	FIM	RID	3PL	TLC	ORG

[Data type] Bit

ORG Movement when coordinate system is altered during drawing
0 : Draws in the same coordinate system
1 : Draws in the new coordinate system (only for the path drawing)

TCL Solid drawing
0 : Do not compensate for the tool length
1 : Compensates for the tool length

3PL Tri–plane drawing in solid drawing
0 : Drawn by the first angle
1 : Drawn by the third angle

RID In solid drawing
0 : Draws a plane without edges.
1 : Draws a plane with edges.

FIM Machining profile drawing in solid drawing
0 : Displays in the coarse mode
1 : Displays in the fine mode

CSR While the screen image is enlarged, the shape of the graphic cursor is:
0 : A square. (■)
1 : An X. (×)

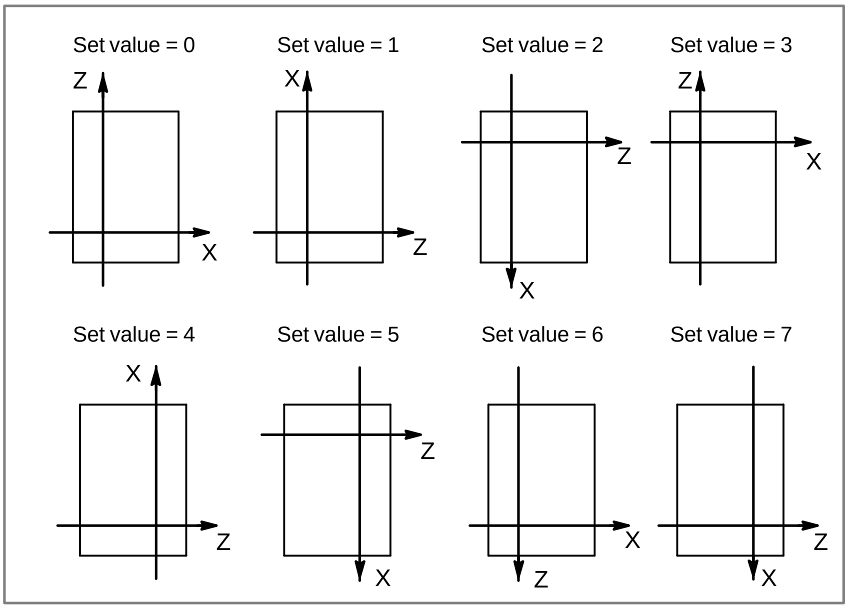
6510	Drawing coordinate system

[Data type] Byte

[Valid data range] 0 to 7

This parameter specifies the drawing coordinate system for the graphic function.

The following show the relationship between the parameter values and the drawing coordinate systems.



6511	
	Right margin in solid drawing

6512	
	Left margin in solid drawing

6513	
	Upper margin in solid drawing

6514	
	Lower margin in solid drawing

[Data type] Word

[Unit of data] Dot

These parameters set the drawing margins in pixels on the screen. The unit is a dot.

Parameter No.	Margin area	Standard set value	
		DPO=0	DPO=1
6511	Right	0	200
6512	Left	0	0
6513	Upper	25	25
6514	Lower	0	0

Set DPO with parameter No. 6500#5.

6515	
	Change in cross–section position in tri–plane drawing

[Data type] Byte

[Unit of data] Dot

[Valid data range] 0 to 10

This parameter sets the change in the cross–section position when a soft key is continuously pressed in tri–plane drawing. When zero is specified, it is assumed to be 1.

Note

> **NOTE**
> When the dynamic graphics function is used, the graphics function cannot be used.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY

12.1.13 Displaying Operating Monitor

General

The load meter can be displayed for each servo axis and the serial spindle.

- **Display of the servo axes**

The load meter can be displayed for up to three servo axes by setting parameters 3151 to 3153.

- **Display of the spindle axes**

When serial spindles are used, the load meter and speedometer can be displayed only for the main serial spindle.

- **Speedmeter**

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM					

[Data type] Bit

OPM Operating monitor

0 : Not displayed
1 : Displayed

OPS The speedometer on the operating monitor screen indicates:

0 : Spindle motor speed
1 : Spindle speed

3151	Axis number for the first load meter
3152	Axis number for the second load meter
3153	Axis number for the third load meter
3154	Axis number for the 4th load meter
3155	Axis number for the 5th load meter
3156	Axis number for the 6th load meter
3157	Axis number for the 7th load meter
3158	Axis number for the 8th load meter

[Data type] Byte

[Valid data range] 0, 1, . . . , the number of the control axes

Set the numbers of the axes for which measurement values on the load meters for the eight servo motors are displayed. Set the parameters to 0 for those axes for which a load meter need not be displayed.

2086	Rated current parameter (RTCURR)

[Data type] Word axis

4127	Load meter displayed value for maximum output

[Data type] Word axis

Note

NOTE
The load meter display depends on servo parameter 2086 and spindle parameter 4127.
These parameters are set automatically.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.1.7	Operating Monitor Display
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.1.7	Operating Monitor Display
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.1.7	Operating Monitor Display
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.1.7	Operating Monitor Display

12.1.14 Software Operator's Panel

General

The software operator's panel function replaces part of the control switches on the machine operator's panel.

The control switches for the functions listed in the following table can be replaced with soft switches. Also available are eight general–purpose soft switches which can be assigned by the machine tool builder. These eight general–purpose soft switches can be optionally named by the machine tool builder. For control switches in groups 1 to 7, parameter (no.7200) can be used to select whether the control switches on the machine operator's panel or soft switches on the MDI of the control unit are used.

Group1 :Mode selection

Group2 :Selection of jog feed axis, manual rapid traverse

Group3 :Selection of manual pulse generator feed axis, selection of manual pulse magnification

Group4 :Jog feedrate override, feedrate override, rapid traverse override

Group5 :Optional block skip, single block, machine lock, dry run

Group6 :Protect key

Group7 :Feed hold

Group8 :General purpose

The states of all soft switches are input to the PMC by output signals. Based on these output signals, the PMC should turn "1" or "0" input signals related to the soft switch functions.
When the soft switch provided for single block operation is turned on, for instance, the control unit does not select the single block operation internally. The single block operation is selected just when the PMC sets the input signal for single block operation to 1.

Signal

Group	Function	Output signal	Related input signal
1	Mode selection	MD1O <F073#0> MD2O <F073#1> MD4O <F073#2> ZRNO <F073#4>	MD1 MD2 MD4 ZRN
2	Jog feed axis select	+J10 – +J40 –J10 – –J40 <F081>	+J1 – +J4 –J1 – –J4
	Manual rapid traverse	RTO <F077#6>	RT
3	Handle feed	HS1AO <F077#0> HS1BO <F077#1> HS1CO <F077#2> HS1DO <F077#3>	HS1A HS1B HS1C HS1D
	Handle feed magnification	MP1O <F076#0> MP2O <F076#1>	MP1 MP2
4	Jog feed rate override	*JV0O – *JV15O <F079, F080>	*JV0 – *JV15
	Feedrate override	*FV0O – *FV7O <F078>	*FV0 – *FV7
	Rapid traverse override	ROV1O <F076#4> ROV2O <F076#5>	ROV1 ROV2
5	Optional block skip	BDTO <F075#2>	BDT
	Single block	SBKO <F075#3>	SBK
	Machine lock	MLKO <F075#4>	MLK
	Dryrun	DRNO <F075#5>	DRN
6	Protect key	KEYO <F075#6>	KEY1 – KEY4
7	Feed hold	SPO <F075#7>	*SP
8	General purpose (Switch from 1st line to the 8th line)	OUT0 – OUT7 <F072>	

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O			MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	–J4O	+J4O	–J3O	+J3O	–J2O	+J2O	–J1O	+J1O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Data type] Bit

OP1 Mode selection on software operator’s panel

0 : Not operational
1 : Operational

OP2 JOG feed axis select and manual rapid traverse buttons on software operator’s panel

0 : Not operational
1 : Operational

OP3 Manual pulse generator’s axis select and manual pulse generator’s magnification switches on software operator’s panel

0 : Not operational
1 : Operational

OP4 JOG feedrate override and rapid traverse override switches on software operator’s panel

0 : Not operational
1 : Operational

OP5 Optional block skip, single block, machine lock, and dry run switches on software operator’s panel

0 : Not operational
1 : Operational

OP6 Protect key on software operator’s panel

0 : Not operational
1 : Operational

OP7 Feed hold on software operator's panel

0 : Not operational
1 : Operational

7210	Jog–movement axis and its direction on software operator's panel "↑"
7211	Jog–movement axis and its direction on software operator's panel "↓"
7212	Jog–movement axis and its direction on software operator's panel "→"
7213	Jog–movement axis and its direction on software operator's panel "←"
7214	Jog–movement axis and its direction on software operator's panel "↙"
7215	Jog–movement axis and its direction on software operator's panel "↗"
7216	Jog–movement axis and its direction on software operator's panel "⤴"
7217	Jog–movement axis and its direction on software operator's panel "↷"

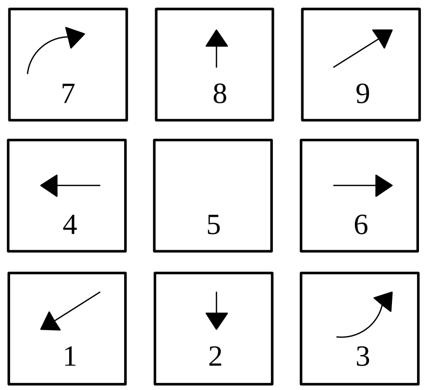
[Data type] Byte

[Valid data range] 0 to 8

On software operator's panel, set a feed axis corresponding to an arrow key on the MDI panel when jog feed is performed.

Set value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on the MDI panel



Example

Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. [8↑] to the positive direction of the Z axis, [2↓] to the negative direction of the Z axis, [6→] to the positive direction of the X axis [4←] to the negative direction of the X axis, [1↙] to the positive direction of the Y axis, [9↗] to the negative direction of the Y axis

Parameter No. 7210 = 5 (Z axis, positive direction)
Parameter No. 7211 = 6 (Z axis, negative direction)
Parameter No. 7212 = 1 (X axis, positive direction)
Parameter No. 7213 = 2 (X axis, negative direction)
Parameter No. 7214 = 3 (Y axis, positive direction)
Parameter No. 7215 = 4 (Y axis, negative direction)
Parameter No. 7216 = 0 (Not used)
Parameter No. 7217 = 0 (Not used)

7220	Name of general–purpose switch on software operator's panel
⋮	⋮
7283	Name of general–purpose switch on software operator's panel

[Data type] Byte

Example

These parameters set the names of the general–purpose switches (SIGNAL 1 through SIGNAL 8) on the software operator's panel.

```
OPERATOR' S  PANEL        O1234  N5678

SIGNAL 1  :    OFF   ON
SIGNAL 2  :    OFF   ON
SIGNAL 3  :    OFF   ON
SIGNAL 4  :    OFF   ON
SIGNAL 5  :    OFF   ON
SIGNAL 6  :    OFF   ON
SIGNAL 7  :    OFF   ON
SIGNAL 8  :    OFF   ON
```

These names are set using character codes that are displayed in parameter Nos. 7220 to 7283.

Parameter No. 7220:

Sets the character code (083) corresponding to S of SIGNAL 1.

Parameter No. 7221:

Sets the character code (073) corresponding to I of SIGNAL 1.

Parameter No. 7222:

Sets the character code (071) corresponding to G of SIGNAL 1.

Parameter No. 7223:

Sets the character code (078) corresponding to N of SIGNAL 1.

Parameter No. 7224:

Sets the character code (065) corresponding to A of SIGNAL 1.

Parameter No. 7225:

Sets the character code (076) corresponding to L of SIGNAL 1.

Parameter No. 7226:

Sets the character code (032) corresponding to (space) of SIGNAL 1.

Parameter No. 7227:

Sets the character code (049) corresponding to 1 of SIGNAL 1.

Parameter Nos. 7228 to 7235:

Set the character codes of SIGNAL 2 shown in the figure above.

Parameter Nos. 7236 to 7243:

Set the character codes of SIGNAL 3 shown in the figure above.

Parameter Nos. 7244 to 7251:

Set the character codes of SIGNAL 4 shown in the figure above.

Parameter Nos. 7252 to 7259:

Set the character codes of SIGNAL 5 shown in the figure above.

Parameter Nos. 7260 to 7267:

Set the character codes of SIGNAL 6 shown in the figure above.

Parameter Nos. 7268 to 7275:

Set the character codes of SIGNAL 7 shown in the figure above.

Parameter Nos. 7276 to 7283:

Set the character codes of SIGNAL 8 shown in the figure above.

The character codes are shown in character code list on the following page.

Character to Code Correspondence Table

Character	Code	Com-ment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation marks
H	072		#	035	Number
I	073		$	036	Dollar mark
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Positive sign
Q	081		,	044	Comma
R	082		–	045	Negative sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	Commercial at mark
1	049		[	091	Left square bracket
2	050		¥	092	Yen mark
3	051		]	093	Right square bracket
4	052		^	094	
5	053		_	095	Underline

Note

NOTE

1 Only the modes shown below can be selected by soft switches. When the mode for DNC operation is to be required, then, all control switches for mode selection should be on the machine operator's panel or a general–purpose soft switch should be used to select the mode for DNC operation.

Soft switches available for mode selection

- Manual data input
- Automatic operation
- Memory edit
- Manual handle feed / incremental feed
- Jog feed
- Manual reference position return

2 Only one soft switch is available for the protection key. But, four input signals are available for protection key (KEY1, KEY2, KEY3 and KEY4). Generally, four input signals are simultaneously turned to "1" or "0" according to the state of the protection soft switch.

3 When the soft switch for feed hold is turned on, output signal SPO is turned to "1", and the PMC turns feed hold signal *SP to "0".

In contrast to the above, when the soft switch for feed hold is turned off, output signal SPO is turned "0" and the PMC turns signal *SP to "1". For soft switches other than feed hold and general soft switches, when an output signal corresponding to a soft switch is turned to "1", the corresponding input signal is turned to "1".

NOTE

4 The following table lists the jog feedrate override values which can be selected by soft switches.

	*JV00 – *JV150 (*JV0 – *JV150) 15 ↓ 12 ↓ 8 ↓ 4 ↓ 0 ↓ bit				Override values (%)
0	1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1	1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.1
2	1 1 1 1	1 1 1 1	1 1 1 1	0 0 0 1	0.14
3	1 1 1 1	1 1 1 1	1 1 1 0	1 0 1 1	0.2
4	1 1 1 1	1 1 1 1	1 1 1 0	0 1 0 0	0.27
5	1 1 1 1	1 1 1 1	1 1 0 1	1 0 1 0	0.37
6	1 1 1 1	1 1 1 1	1 1 0 0	1 0 1 1	0.52
7	1 1 1 1	1 1 1 1	1 0 1 1	0 1 1 1	0.72
8	1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.0
9	1 1 1 1	1 1 1 1	0 1 1 1	0 0 1 1	1.4
10	1 1 1 1	1 1 1 1	0 0 1 1	0 1 1 1	2.0
11	1 1 1 1	1 1 1 0	1 1 1 1	0 0 0 1	2.7
12	1 1 1 1	1 1 1 0	1 0 0 0	1 1 0 1	3.7
13	1 1 1 1	1 1 0 1	1 1 1 1	0 1 1 1	5.2
14	1 1 1 1	1 1 0 1	0 0 1 0	1 1 1 1	7.2
15	1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.0
16	1 1 1 1	1 0 1 0	1 0 0 0	0 1 1 1	14.0
17	1 1 1 1	1 0 0 0	0 0 1 0	1 1 1 1	20.0
18	1 1 1 1	0 1 0 1	0 1 1 1	0 0 1 1	27.0
19	1 1 1 1	0 0 0 1	1 0 0 0	1 0 1 1	37.0
20	1 1 1 0	1 0 1 1	1 0 1 0	1 1 1 1	52.0
21	1 1 1 0	0 0 1 1	1 1 0 1	1 1 1 1	72.0
22	1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.0
23	1 1 0 0	1 0 0 1	0 1 0 0	1 1 1 1	140.0
24	1 0 1 1	0 0 0 1	1 1 0 1	1 1 1 1	200.0

NOTE

5 The following table lists the feedrate override values which can be selected by soft switches.

	*FV0O – *FV7O (*FV0 – *FV7) 7 ↓ 4 ↓ 0 ↓		Override values (%)
0	1 1 1 1	1 1 1 1	0
1	1 1 1 1	0 1 0 1	10
2	1 1 1 0	1 0 1 1	20
3	1 1 1 0	0 0 0 1	30
4	1 1 0 1	0 1 1 1	40
5	1 1 0 0	1 1 0 1	50
6	1 1 0 0	0 0 1 1	60
7	1 0 1 1	1 0 0 1	70
8	1 0 1 0	1 1 1 1	80
9	1 0 1 0	0 1 0 1	90
10	1 0 0 1	1 0 1 1	100
11	1 0 0 1	0 0 0 1	110
12	1 0 0 0	0 1 1 1	120
13	0 1 1 1	1 1 0 1	130
14	0 1 1 1	0 0 1 1	140
15	0 1 1 0	1 0 0 1	150
16	0 1 0 1	1 1 1 1	160
17	0 1 0 1	0 1 0 1	170
18	0 1 0 0	1 0 1 1	180
19	0 1 0 0	0 0 0 1	190
20	0 0 1 1	0 1 1 1	200

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.10	Displaying and Setting the Software Operator's Panel
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.14	Displaying and Setting the Software Operator's Panel
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.10	Displaying and Setting the Software Operator's Panel
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.13	Displaying and Setting the Software Operator's Panel

12.1.15 Multi–language Display

General

The language displayed on the screen is set by a parameter.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3102		SPN	HNG	ITA	CHI	FRN	GRM	JPN
	DTH	SPN	HNG	ITA	CHI	FRN	GRM	JPN

	#7	#6	#5	#4	#3	#2	#1	#0
3119							POR	

	#7	#6	#5	#4	#3	#2	#1	#0
3190		CH2	CZE	SWE	HUN	POL		

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

[Data type] Bit

Select the language to be used for the display.

CH2	CZE	SWE	HUN	POL	POR	DTH	SPN	HNG	ITA	CHI	FRN	GRM	JPN	Language
0	0	0	0	0	0	0	0	0	0	0	0	0	0	English
0	0	0	0	0	0	0	0	0	0	0	0	0	1	Japanese
0	0	0	0	0	0	0	0	0	0	0	0	1	0	German
0	0	0	0	0	0	0	0	0	0	0	1	0	0	French
0	0	0	0	0	0	0	0	0	0	1	0	0	0	Chinese (traditional characters)
0	0	0	0	0	0	0	0	0	1	0	0	0	0	Italian
0	0	0	0	0	0	0	0	1	0	0	0	0	0	Korean
0	0	0	0	0	0	0	1	0	0	0	0	0	0	Spanish
0	0	0	0	0	0	1	0	0	0	0	0	0	0	Dutch
0	0	0	0	0	1	0	0	0	0	0	0	0	0	Portuguese
0	0	0	0	1	0	0	0	0	0	0	0	0	0	Polish
0	0	0	1	0	0	0	0	0	0	0	0	0	0	Hungarian
0	0	1	0	0	0	0	0	0	0	0	0	0	0	Swedish
0	1	0	0	0	0	0	0	0	0	0	0	0	0	Czech
1	0	0	0	0	0	0	0	0	0	0	0	0	0	Chinese (simplified characters)

12.1.16 External Operator Message Logging and Display

General

Parameter setting enables one external operator message (consisting of up 255 characters) or, simultaneously, up to 4 external operator message (consisting of up to 63 characters) to be displayed.
External operator messages can be logged in a history file.

These messages can be displayed on the external operator message history screen.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Data type] Bit

OMH The external operator message history screen is:

0 : Not displayed.
1 : Displayed.

		#7	#6	#5	#4	#3	#2	#1	#0
3113		MS1	MS0						MHC

MHC External operator message history data:

0 : Cannot be cleared.
1 : Can be cleared.
(Such data can be cleared using the [CLEAR] soft key.)

MS0, MS1 Set the number of items and the item length.

MS1	MS0	Number of history data characters	Number of history data items
0	0	255	8
0	1	200	10
1	0	100	18
1	1	50	32

CAUTION
When the values of MS0 and MS1 are changed, all preserved external operator message history data is cleared.

		#7	#6	#5	#4	#3	#2	#1	#0
3207									OM4

[Data type] Bit

OM4 A message displayed on the external operator message screen can have:
0 : Up to 256 characters, and just a single message can be displayed.
1 : Up to 64 characters, and up to four messages can be displayed.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.7.1	External operator message logging and display
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.7.1	External operator message logging and display
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.7.1	External operator message logging and display
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.7.1	External operator message logging and display

12.1.17 Erase Screen Display/Automatic Erase Screen Display

General

Displaying the same characters in the same positions on the screen causes a LCD to degrade relatively quickly. To help prevent this, the screen can be cleared by pressing specific keys. It is also possible to specify the automatic clearing of the screen if no keys are pressed during a period specified by a parameter.
However, it is disabled for the open CNC.

Erase screen display

Holding down the CAN key and pressing a function key clears the screen.

Hold down the CAN key and press a function key (such as POS and PROG).

Press a function key, to restore the display.

Automatic erase screen display

The CNC screen is automatically cleared if no keys are pressed during the period (in minutes) specified by parameter (No. 3123). The screen is restored by pressing any key.

- **Clearing the screen with automatic erase screen display**

The CNC screen is cleared once the period (minutes) specified with parameter No. 3123 has elapsed, provided the following conditions are satisfied:

Conditions for clearing the CNC screen

- Automatic erase screen display cancel signal *CRTOF is “0”.
- Parameter No. 3123 is non–zero.
- None of the following keys have been pressed:
  MDI keys
  Soft keys
  External input keys
- No alarm has been issued.

- **Restoring the screen with automatic erase screen display**

A cleared CNC screen is restored once at least one of the following conditions is satisfied:

Conditions for restoring the CNC screen

- Automatic erase screen display cancel signal *CRTOF is “1”.
- Any of the following keys has been pressed:
  MDI keys
  Soft keys
  External input keys
- An alarm has been issued.

- **Clearing the screen using CAN + function key**

If parameter No. 3123 is set to 0, clearing of the screen using the CAN key and a function key is disabled.

- **Alarm**

The screen is not cleared if an alarm is issued.

> **WARNING**
> Pressing any key while the screen is being cleared restores the screen. Do not press the DELETE, INSERT, or ALTER key to restore the screen.

Signal

Automatic erase screen display cancel signal *CRTOF <G062#1>

[Classification] Input signal

[Function] Enables or disables the automatic screen erase function. This signal is used to switch the control mode.

[Operation] When this signal is set to 0, the control unit:

· Enables the automatic screen erase function.

When this signal is set to 1, the control unit:

· Disables the automatic screen erase function, displays the screen, and initializes the timer.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G062							*CRTOF	

Parameter

3123	Screen display timeout

[Data type] Bytes

[Unit of data] Minutes

[Valid data range] 1 to 255

This parameter specifies the period that must elapse before erase screen display is applied. If 0 is set, the screen is not cleared.

Limitation

For the display unit with PC, this function can not be used.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.8	Cleaning the screen
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.8	Cleaning the screen
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.8	Cleaning the screen
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.8	Cleaning the screen

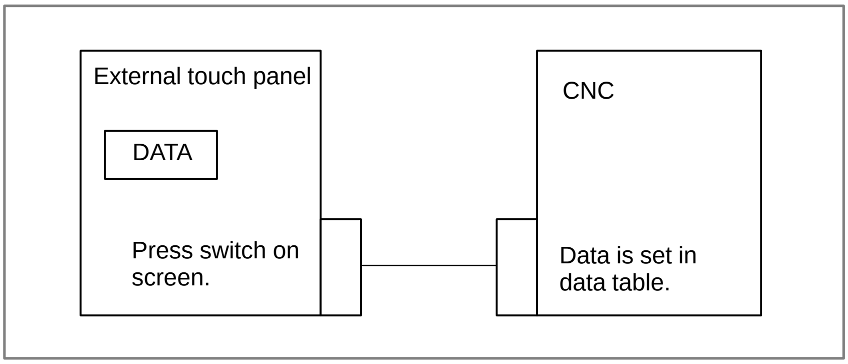
12.1.18 External Touch Panel Interface

General

Support of the external touch panel interface allows an SNP–X protocol compliant external touch panel.

External touch panels are similar to the machine operator's panel. By using an external touch panel, the user can read and write the signals controlled by the PMC (input signals, (X), output signals (Y), internal relays (R), keep relays (K), data tables (D), timers (T), and counters (C)).

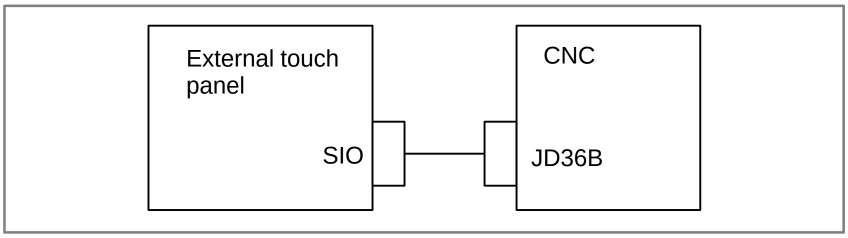
The external touch panel features a plotting capability. The user can perform plotting and address (signal) assignment freely. For example, when a screen with data table settings assigned is created, data can be set in the data table by using switches on the screen.

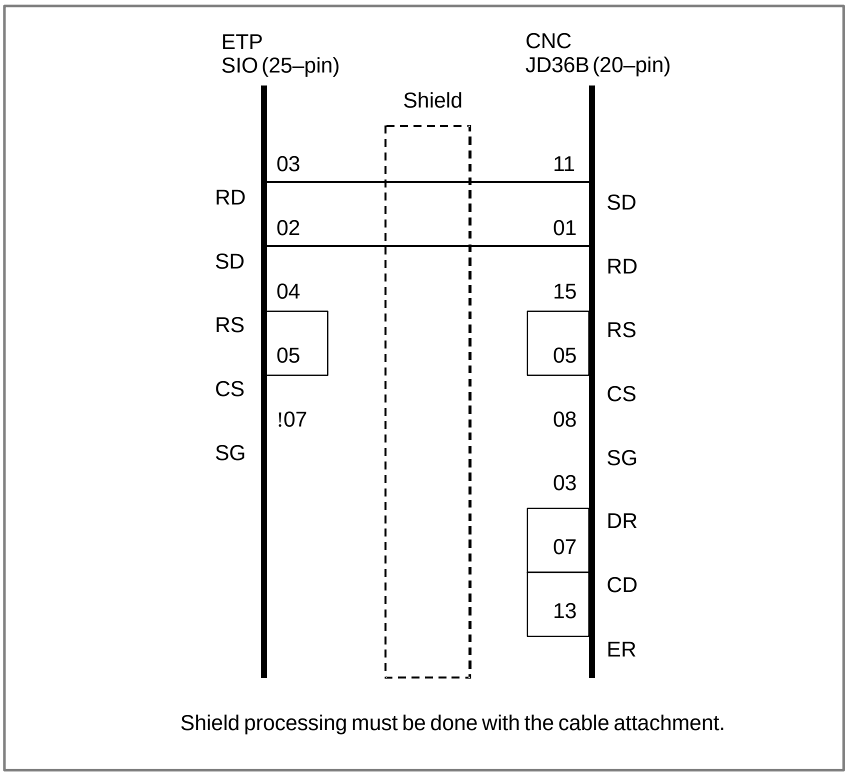


Function details

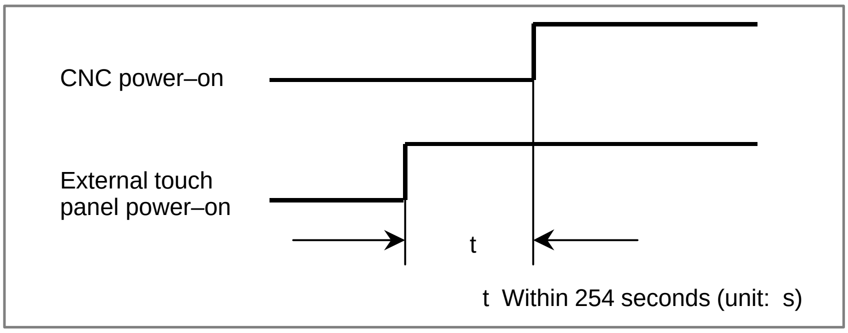
• Connection

The customer needs to prepare the cable for connecting the RS–232C serial port 2 (JD36B) of the main CPU board of the CNC and the SIO connector of the external touch panel.





• Power–on sequence



As the start time in the system setting for the external touch panel, set ”t+1”.

When the CNC power–on operation is earlier than the power–on operation of the external touch panel, there is no problem.

The period of the CNC power–on operation refers to the period of time until the power–on reset state terminates.

- **CNC data that can be read from and written to the external touch panel**

Data at the CNC addresses listed below can be read into and written from the external touch panel. The address range varies depending on the PMC model.

• Input address:	X0000 to X0127 (read only)
• Output address:	Y0000 to Y0127
• Internal relay:	R0000 to R1499
• Keep relay:	K0000 to K0019
• Data table:	D0000 to D2999
• Timer:	T0000 to T0079
• Counter:	C0000 to C0079

When the PMC–SA1 are used, data at the following addresses can be read into and written from the external touch panel:

• Input address:	X0000 to X0127 (read only)
• Output address:	Y0000 to Y0127
• Internal relay:	R0000 to R0999
• Keep relay:	K0000 to K0019
• Data table:	D0000 to D1859
• Timer:	T0000 to T0079
• Counter:	C0000 to C0079

- **Protocol**

The CNC uses SNP–X protocol direct commands only. So, as the protocol on the external touch panel (ETP) side, also use SNP–X protocol direct commands only.
The processing in response to a request for writing 3–byte or longer data is the same as the processing in response to a request for writing 2–byte or shorter data. For details of the SNP–X protocol (such as device code specifications), refer to ”SNP–X Protocol Document”.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3119					TPD			

[Data type] Bit

TPD Connection of an external touch panel is:

0 : Enabled.
1 : Disabled.

CAUTION
As described in "Connection" in "Function details" above, RS–232C serial port 2 (JD36B) of the main CPU board of the CNC is used for ETP connection.
When using an external touch panel, set bit 3 (TPLDS) of parameter No. 3119 to 0. Then, the JD36B can be used specifically for the external touch panel regardless of the existing I/O channel setting (I/O device selection) in parameter No. 20 (including Nos. 21 to 23). For other I/O device connection, use another port such as the JD36A.
When TPLDS is set as described above, the existing settings in parameter Nos. 100 and 121 to 123 are ignored for channel 2 (JD36B), and the following settings are assumed:

- Baud rate: 19200 bps
- Stop bit: 1 bit
- Parity check: Even parity

12.1.19 Periodic Maintenance Screen

General

The periodic maintenance screen shows the current statuses of those items that require periodic replacement (backup battery, LCD backlight, touch panel, etc.). An item whose service life has expired is indicated by the machine run time.

Screen displays and settings

1 Press the SYSTEM function key.

2 Press the ▷ next menu page key several times until the [MAINTE] soft key appears.

3 Press the [MAINTE] soft key. The periodic maintenance screen appears.

The periodic maintenance screen consists of two pages: status screen and setup screen, either of which can be selected using the [CHANGE] soft key.

- **Status screen displays and settings**

On the status screen, it is possible to register up to 10 maintenance items. The remaining lifetime and count status for each registered item are displayed on the screen.

```
PERIODICAL MAINTENANCE                O0001 N12345
   (STATUS)
       ITEM NAME                           REMAIN
 *01  BATTERY FOR CONTROLLER                   0H
 @02  BATTERY FOR PULSECODER                5000H
  03  FAN MOTOR                       10000H
 @04  LCD BACK LIGHT                         720H
  05
  06
  07
  08
  09
  10
>_
EDIT *** ***** *** ****     19:27:05
[      ][ MAINTE ][       ][       ][ (OPRT) ]
```

↓

```
[CHANGE][ ENTRY ][ CLEAR ][ +INPUT ][ INPUT ]
```

↓

```
[       ][       ][       ][  CAN  ][  EXEC  ]
```

(1) Maintenance item names
The names of maintenance items are set up here. They can be registered using either the corresponding menu or MDI keys.

1) Menu–based setup

1 To display the setting menu, place the cursor on the desired item, and press the [ENTRY] soft key. There are two types of setup menus: machine maintenance menu and CNC maintenance menu.

2 Pressing the [MACHIN] or [NC] soft key displays a menu that contains typical machine or CNC model names.

3 Place the cursor on the desired item and press the [SELECT] soft key. Now press the [EXEC] soft key to return to the status screen where the selected item name is set up.

4 Pressing the [CAN] soft key restores the previous screen.

5 Pressing the [MAINTE] soft key displays the status screen.

On the machine maintenance screen, items can be registered according to the procedure below.

(a) Program–based registration
An item can be registered by executing a program in the following format.

Format

G10 L61 Px [n]

x : Registration number
n : Item name
[Alphanumeric character]

NOTE

For the CNC edit function, specific strings of two or more characters are registered as reserved words of custom macro commands.

(Example: GO for GOTO, WH for WHILE, and SI for SIN)

Therefore, when a program to register item names is created in the above format using the CNC edit function, input strings may be converted to the reserved words of custom macro commands. Thus, desired strings may not be input.

(Example: When SIGNAL is input, it is converted to SINGNAL.)

When the string to be displayed as an item number cannot be input with the CNC edit function, set the string by editing the data output from the periodic maintenance screen on the PC, and then loading the data through the periodic maintenance screen, as described in ”Inputting/outputting registered data” later in this section (12.1.20 Periodic maintenance screen).

For the reserved words of the custom macro commands, refer to ”Editing Custom Macros” in Operator’s Manual.

(b) MDI–based registration

An item name can be registered by first keying in data in the following format, then pressing the [INPUT]. In addition, pressing the [+INPUT] soft key can modify an existing item.

Format

Alphanumeric–character*two–byte–character* alphanumeric–character

Two–byte character codes conform to the FUNUC codes.

To key in a two–byte character code, key in an asterisk (*) before and after the character code.

Up to 24 characters can be registered as an item name when it consists of only alphanumeric characters, while up to 12 characters can be registered when it consists of only two–byte characters.

Example) To register ”LCD backlight”
>LCD*110E10F410CC114010B610FE_

NOTE

1 The asterisk (*) is used as a control code, so it cannot be used in the item name. Likewise, square brackets “[” and “]” and parentheses “(” and “)” must be excluded from the item name.
2 If a null item is selected on the machine maintenance screen, the warning message “EDIT REJECTED” will appear. If a null item name is selected on the NC maintenance screen, a blank will be set up.
3 When a blank item is selected on a screen of the machine, a warning “Unable to edit” is issued.
When a blank item is selected on a screen of the NC, a blank is set.

To delete a registered item name, place the cursor on it, and press the [CLEAR] soft key, then the [EXEC] soft key.

[Machine maintenance screen]

```
PERIODICAL MAINTENANCE           O0001 N12345
   (MACHINE)
        ITEM NAME
  01
  02
  03
  04
  05
  06
  07
  08
  09
  10
>_
EDIT *** ***** *** ****     19:27:05
[      ][ STATUS ][ MACHIN ][  NC  ][ (OPRT) ]
```

```
[ SELECT ][        ][ CLEAR ][ +INPUT ][ INPUT ]
```

```
[        ][        ][        ][  CAN   ][ EXEC  ]
```

```
[        ][  READ  ][ PUNCH  ][        ][       ]
```

```
[        ][        ][        ][  CAN   ][ EXEC  ]
```

[CNC maintenance screen]

```
PERIODICAL MAINTENANCE              O0001 N12345
   (NC)
       ITEM NAME
  01  BATTERY FOR CONTROLLER
  02  BATTERY FOR PULSECODER
  03  FAN MOTOR
  04  LCD BACK LIGHT
  05
  06
  07
  08
  09
  10
>_
EDIT *** ***** *** ****     19:27:05
[      ][ STATUS ][ MACHIN ][  NC  ][ (OPRT) ]
```

```
[ SELECT ][        ][        ][        ][        ]
```

```
[        ][        ][        ][  CAN  ][  EXEC  ]
```

NOTE

On the CNC maintenance screen, it is impossible to register, delete, input, and output item.

2) MDI–based setup
An item can be registered by first keying in data in the following format, then pressing the [INPUT]. In addition, pressing the [+INPUT] soft key can modify an existing item name.

NOTE

The asterisk (*) is used as a control code, so it cannot be used in the item name. Likewise, square brackets “[” and “]” and parentheses “(” and “)” must be excluded from the item name.

To delete a registered item, place the cursor on it, and press the [CLEAR] soft key, then the [EXEC] soft key.
When an item is deleted, the related service life, remaining lifetime, and count type are also deleted.

(2) Remaining lifetime
The remaining lifetime of an item is the time allowed before it must be replaced. It is displayed in a count–down format. When the remaining lifetime becomes less than the percentage specified in parameter No. 8911 of the corresponding service life, the remaining lifetime display turns red. Count–down continues even after the service life has expired.

NOTE
No setup can be made on the status screen. Setup is possible only on the setting screen.

(3) Count status
The count status of each item is displayed at the left of the corresponding item number as listed below:

Display	Count status
Blank	Counting is at a halt.
@	Counting is under way.
*	Service life has expired.

- **Setting screen displays and settings**

On the setting screen, it is possible to specify the service life, remaining lifetime, and count type for each registered item. It also displays the same count status as displayed on the status screen.

```
PERIODICAL MAINTENANCE              O0001 N12345
   (SETTING)
        LIFE        REMAIN      COUNT TYPE
 *01   10000H           0H       ALL TIME
 @02   20000H        5000H       LIV TIME
  03   32767H       10000H       --------
 @04    1500H         720H       RUN TIME
  05
  06
  07
  08
  09
  10
>_
EDIT *** ***** *** ****     19:27:05
[ CHANGE ][ TYPE ][ CLEAR ][ +INPUT ][ INPUT ]
```

↓

```
[ EFFECT ][  ALL  ][  LIV  ][  RUN  ][  CUT  ]
```

```
[        ][  READ  ][ PUNCH ][  CAN  ][  EXEC  ]
```

(1) Service life

To specify the service life, key in the corresponding data and press [INPUT]. The same data is set up as both the service life and remaining lifetime. In addition, the count type is indicated as: "______"

Pressing the [+INPUT] soft key causes newly entered data to be added to the existing service life and the remaining lifetime.

The valid data range for this item is: 0 to 65535 (hours)

NOTE

1 If an attempt is made to enter data for a null item name, the warning message "EDIT REJECTED" appears.
2 If an attempt is made to enter data that does not fall into the valid data range, the warning message "DATA IS OUT OF RANGE" appears.
3 If an attempt is made to enter data that would set the service life or remaining lifetime below 0, it will be set to 0.
4 If the [CLEAR] or [TYPE] soft key is pressed, the warning message "EDIT REJECTED" appears.

(2) Remaining lifetime

The remaining lifetime of an item is the time allowed before it must be replaced. It is displayed in a count–down format. When the remaining lifetime becomes less than the percentage specified in parameter No. 8911 of the corresponding service life, the remaining lifetime display turns red. Count–down continues even after the service life has expired.

To specify the remaining lifetime, key in the corresponding data and press [INPUT].

Pressing the [+INPUT] soft key causes newly entered data to be added to the current remaining lifetime.

The valid data range for this item is: 0 to the corresponding service life

Pressing the [CLEAR] soft key, then the [EXEC] soft key causes the remaining lifetime to be set with the same value as the service life.

NOTE

1 If an attempt is made to enter data for a null item, the warning message "EDIT REJECTED" appears.
2 If an attempt is made to enter data that does not fall into the valid data range, the warning message "DATA IS OUT OF RANGE" appears.
3 If an attempt is made to enter data that would set the remaining lifetime below 0, it will be set to 0.
4 If the [TYPE] soft key is pressed, the warning message "EDIT REJECTED" appears.

(3) Count type

Pressing the [TYPE] causes the following count types to be displayed as soft keys. Select the desired one and press the [EXEC] soft key to set it up.

Software	Meaning	Display
[EFFECT]	No counting takes place (the counter is halted).	– – –
[ALL–AX]	Counting continues unconditionally (nonstop counting).	ALL
[LIV]	Counting continues as long as the power is on.	LIV
[RUN]	Counting continues during run time.	RUN
[CUT]	Counting continues during cutting time.	CUT

NOTE

1 If an attempt is made to enter data for a null item, the warning message "EDIT REJECTED" appears.

2 The [INPUT] and [+INPUT] soft keys do not function for count type.

3 Leap year is not a ccounted for in the counting, and can cause a 24–hour error in the counting.

Inputting and outputting the registered data

The registered data related to the maintenance service life can be output to an external unit, using the [PUNCH] soft key. Similarly, data can be input from an external unit, using the [READ] soft key. These input/output operations can be performed on the status, setting, and menu (machine maintenance only) screens.

```
EDIT *** ***** *** ****     19:27:05
[        ][  READ  ][ PUNCH ][        ][        ]

[        ][        ][        ][  CAN  ][ EXEC  ]
```

- **Data output**

In EDIT mode, pressing the [PUNCH] soft key causes registered data to be output in the following formats.

○ Status and setting screens

Format

G10 L60 P01 Aa Rr [n] Qq ;
G10 L60 P02 Aa Rr [n] Qq ;
G10 L60 P03 Aa Rr [n] Qq ;
:

○ Menu screen (machine maintenance only)

Format

```
G10 L61 P01 [n] ;
G10 L61 P02 [n] ;
G10 L61 P03 [n] ;
         :
```

a : Service life

r : Remaining lifetime

n : Item name
[Alphanumeric character]

q : Count type
0 = No counting.
1 = Nonstop counting.
2 = Counting continues as long as the power is on.
3 = Counting continues throughout run time.
4 = Counting continues throughout cutting time.

- **Data input**

In EDIT mode, pressing the [READ] soft key causes input data to be registered according to a specified format (G10). Registration is possible even if the format (G10) is already in program memory as long as the programmable data input option is available.

> **NOTE**
> Registration may not be performed correctly unless the input format (G10) matches the output format.

Signal

Periodic maintenance life expiration output signal LIFOVR <F093#0>

[Classification] Output signal

[Function] This signal indicates whether the service life of consumable parts (that need periodic replacement) specified on the periodic maintenance screen has expired.
The signal is valid only when items to be subjected to maintenance have been specified.

[Operation] The signal becomes ”1” when:

· The service life of one or more consumable parts that need periodic replacement expires.

The signal becomes ”0” when:

· The consumable parts whose service life has expired are re–set on the periodic maintenance screen.

Parameter

8911	Lifetime versas service life parcentage

[Data type] Byte

[Unit of data] 1%

[Valid data range] 0 to 100

On the periodic maintenance screen, the remaining lifetime display turns red for warning purposes, if the remaining lifetime goes below a specified percentage of the corresponding service life.

12.1.20 Actual Speed Display

General

The actual speed is displayed on the current position display screen, program check screen, and program screen (MDI mode).
PMC controlled axis movement data can be added to the actual speed display.
Reflection of movement along an arbitrary axis in the actual speed display can also be suppressed by parameter setting.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3105							PCF	DPF

[Data type] Bit

DPF Display of the actual speed on the current position display screen, program check screen and program screen (MDI mode)
0 : Not displayed
1 : Displayed

PCF Addition of the movement of the PMC–controlled axes to the actual speed display
0 : Added
1 : Not added

NOTE
For each setting, movement along any axis other than those controlled by the CNC (see the description of parameter No. 1010) is not reflected in the actual speed display.

	#7	#6	#5	#4	#3	#2	#1	#0
3115					NDFx			

[Data type] Bit axis

NDFx To the actual speed display, axis movement data is:
0 : Added.
1 : Not added.

> **NOTE**
> Even if the parameter PCF (bit 1 of parameter No.3105) is set to 0, so as to add PMC controlled axis movement data to the actual speed display, the movement data for a PMC controlled axis for which NDFx is set to 1 is not added to the actual speed display.

12.1.21 Parameter Set Supporting Screen

General

The parameter set supporting screen is a parameter setting and tuning screen intended to help:

- Readily start up the machine by collecting and displaying the minimum required parameters for machine launching.
- Smoothly make adjustments by easily displaying the servo, spindle, and machining parameter tuning screens.

Operation

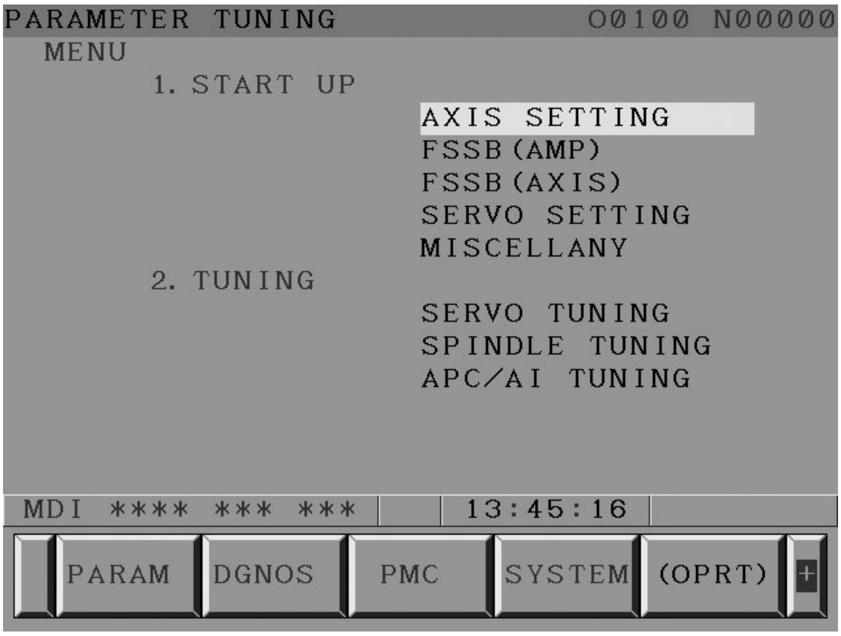
The following two methods can be used to display the parameter set supporting screen.

(1) Press the [SYSTEM] function key several times until the parameter set supporting screen appears.

Parameter → Diagnosis → Parameter set supporting screen

Setting the CPR parameter (bit 2 of parameter No. 3195) to 1 disables the function key from displaying the parameter set supporting screen. If you want to use the conventional method for displaying the screen, use this parameter so that you can follow the method given below.

(2) Press the [+] soft key several times until the [PRMTUN] soft key appears.
Press the [PRMTUN] soft key to display the parameter set supporting screen.



This screen is called the parameter set supporting screen menu.

Display item overview

The items displayed on the parameter set supporting screen are outlined below.

[START UP]
[START UP] lets you specify the minimum required parameters for machine launching.

AXIS SETTING:	The axis, coordinate, feedrate, and acceleration/deceleration CNC parameters are specified.
FSSB (AMP):	The FSSB amp setting screen is displayed.
FSSB (AXIS):	The FSSB axis setting screen is displayed.
SERVO SETTING:	The servo setting screen is displayed.
MISCELLANY:	CNC parameters, such as DI/DO and serial spindle parameters, are specified.

[TUNING]
[TUNING] lets you specify display screens for servo, spindle, and high–speed, high–precision machining tuning.

SERVO TUNING:	The servo tuning screen is displayed.
SPINDLE TUNING:	The spindle tuning screen is displayed.
APC/AI TUNING:	The machining parameter tuning screen (for APC, AI–APC, and AI contour control) is displayed.

Selecting items

Press the [(OPRT)] soft key to display the following soft keys:

[<] [SELECT] [ ] [ ] [ ] [ ] [>]

Place the cursor on the item you want to select.
Press the [SELECT] soft key.

START UP

START UP lets you specify the minimum required parameters for machine launching.
Let us explain the AXIS SETTING and MISCELLANY items. For detailed descriptions about the FSSB (AMP), FSSB (AXIS), and SERVO SETTING items, refer to the respective manuals.

Screen displays

The minimum required CNC parameters for machine launching are collected and subdivided into some groups. They are displayed in groups as follows:

AXIS SETTING item

(BASIC) group: Parameters related to basic setting are displayed.

(COORDINATE) group: Parameters related to coordinates are displayed.

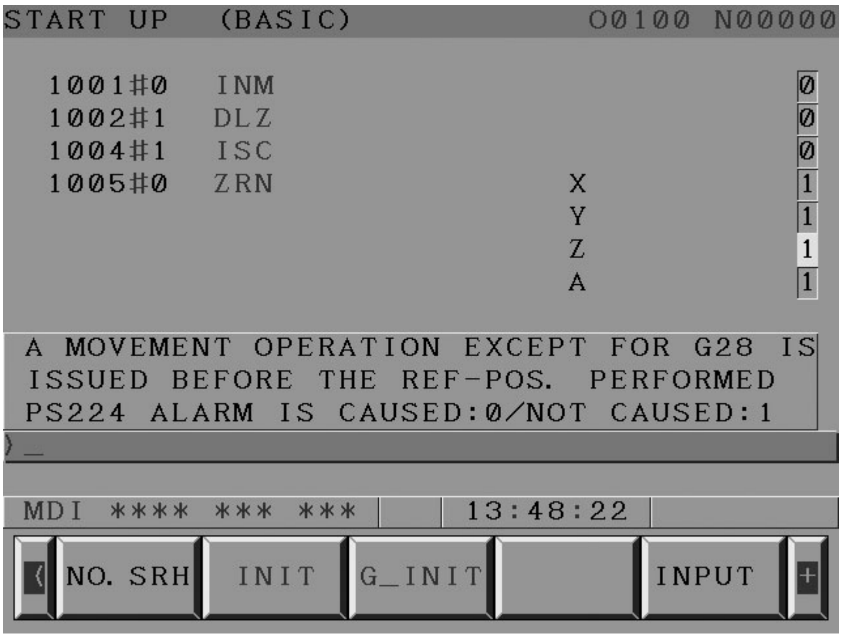
(FEED RATE) group: Parameters related to feedrate are displayed.

(ACC./DEC.) group: Parameters related to acceleration/deceleration are displayed.

MISCELLANY item

(MISC) group: Parameters related to DI/DO and serial spindle settings are displayed.

A brief help message corresponding to a parameter selected with the cursor is displayed.
If a standard value (recommended by FANUC) is available for the parameter, it is also displayed.



Entering parameters

Make sure that the setting screen is "parameter write enabled."
Select the MDI mode, and place the cursor on the parameter you want to specify.
Enter data, using numeric keys, and press the [INPUT] soft key or the [INPUT] key on the MDI.

[<] [NO. SRH] [INIT] [G_INIT] [ ] [INPUT] [>]

Displaying the conventional parameter screen

Press the [–] soft key several times until the following soft keys appear.

[<] [PARAM] [DGNOS] [PMC] [SYSTEM] [(OPRT)] [>]

Press the [PARAM] soft key to display the conventional parameter screen.

Displaying the menu screen

To return to the parameter tuning menu screen after an item is selected, operate as follows:

[<] [NO. SRH] [INIT] [G_INIT] [ ] [INPUT] [>]

Press the [+] soft key to display the [MENU] soft key.

[<] [MENU] [ ] [ ] [ ] [ ] [>]

Pressing the [MENU] soft key displays the parameter set supporting menu screen.
This operation can let you return to the parameter set supporting menu screen also from the FSSB (AMP), FSSB (AXIS), SERVO SETTING, SERVO TUNING, and SPINDLE TUNING screens.

Setting initial values

Standard values can be specified for parameters, using soft keys.

There are two methods. The first method is to specify a standard value only for the parameter selected with the cursor. The second method is to specify standard values for all parameters in a group.
The standard values are those recommend by FANUC. It is impossible to specify standard values specific to an individual user.
The conventional parameter screen allows no standard value to be set up.

• Setting initial values for individual parameters

Select the MDI mode.

Place the cursor on the parameter for which you want to specify a standard value.

Make sure that the following soft keys are displayed, and press the [INIT] soft key.

[<] [NO. SRH] [INIT] [G_INIT] [ ] [INPUT] [>]

The standard value you entered is displayed on the key input line, and the following warning message appears: "Do you really want to perform initialization?"

Press the [EXEC] soft key to set up the standard value.

[<] [ ] [ ] [ ] [CAN] [EXEC] [>]

If you want to quit setting the standard value, press the [CAN] soft key.
If no standard value is available for the parameter selected with the cursor, pressing the [INIT] soft key results in the following warning message being displayed: "No standard value is available."

- **Specifying standard values for an entire group**

Select the MDI mode.

Make sure that the following soft keys are displayed, and press the [G_INIT] soft key.

[<] [NO. SRH] [INIT] [G_INIT] [ ] [INPUT] [>]

Instead of a help message, the following message appears:

The following soft key group appears, and the following warning message appears: ”Do you really want to perform initialization?”

[<] [ ] [ ] [ ] [CAN] [EXEC] [>]

Pressing the [EXEC] soft key causes standard values to be set up for the selected group.
In this case, no standard value is displayed on the key input line. Instead, the setting is automatic. Be extremely careful when using this method.
If you want to quit setting the standard values, press the [CAN] soft key.
Before starting to set up standard values for an entire group, put the machine at an emergency stop for safety purposes.

TUNING

TUNING lets you display the servo, spindle, and machining parameter tuning screens readily, so you can make adjustments easily.
On the parameter tuning screen menu, place the cursor on the desired tuning screen item and press the [SELECT] soft key to display the corresponding screen.

SERVO TUNING: The servo tuning screen is displayed.

SPINDLE TUNING: The spindle tuning screen is displayed.

APC/AI TUNING: The machining parameter tuning screen (for APC, AI–APC, and AI contour control) is displayed.

> **NOTE**
> The machining parameter tuning screen (for APC, AI–APC, and AI contour control) will not be displayed unless the machining condition select option is available.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3195						CPR		

[Data type] Bit

CPR Pressing the [SYSTEM] function key:

0 : Displays the parameter setting assistance screen.
1 : Does not display the parameter setting assistance screen.

Notes

(1) The parameter set supporting screen cannot be used to read or punch parameters.

(2) Standard value setting can be done only for AXIS SETTING on the parameter set supporting screen and items on other screens. The conventional parameter screen cannot be used to specify standard values.

(3) If no standard value is available for the parameter you select with the cursor, pressing the [INIT] soft key results in the following warning message being displayed: ”There is no standard value.”

(4) If no standard value is available for the group you select, pressing the [G_INIT] soft key results in the following warning message being displayed: ”There is no standard value.”

(5) The machining parameter tuning screen does not appear unless the a machining condition select option is available.

Appendix

Parameters required for machine launching

Menu item	Group	Parameter No.	Brief description
AXIS SETTING	BASIC	1001#0	Linear–axis least command increment 0: Millimeter machines 1: Inch machines
		1002#1	Reference position return without dogs 0: Disable 1: Enable (all axes)
		1004#1	Least input increment and least command increment settings 0: IS–B 1: IS–C
		1005#0	Automatic operation (other than G28) with no origin established is responded with: 0: An alarm (No. 224) 1: No alarm
		1005#1	Reference position return without dogs 0: Disable (individual axes) 1: Enable (individual axes)
		1006#0	Selecting linear or rotation axis 0: Linear axis 1: Rotation axis
		1006#3	Specifying a move amount type for individual axes 0: Radius type 1: Diameter type
		1006#5	Direction of a reference position return for individual axes 0: Positive direction 1: Negative direction

Menu item	Group	Parameter No.	Brief description
AXIS SETTING	BASIC	1008#0	Rotation–axis roll–over 0: Disable 1: Enable
		1008#2	Whether to round relative coordinates to a rotation unit: 0: Not to round 1: To round
		1010	Maximum number of axes that the CNC can control
		1020	Individual–axis program name
		1022	Axis assignment in the basic coordinate system
		1023	Individual–axis servo axis No.
		1815#1	Whether to use a separate pulse coder: 0: Not to use 1: To use
		1815#4	Whether the machine position has been associated with the position of the absolute position detector: 0: Associated 1: Not associated
		1815#5	The position detector used is: 0: Not an absolute position detector 1: An absolute position detector
		1825	Individual–axis servo loop gain
		1826	In–position check effective area for individual axes
		1828	Positioning deviation limit for individual axes during movement
		1829	Positioning deviation limit for individual axes at stop
	COORDINATE	1240	First–reference position machine coordinates for individual axes
		1241	Second–reference position machine coordinates for individual axes
		1260	Move amount per rotation–axis rotation
		1320	Positive–direction boundary coordinates for stored stroke check 1
		1321	Negative–direction boundary coordinates for stored stroke check 1

Menu item	Group	Parameter No.	Brief description
AXIS SETTING	FEED RATE	1401#6	Whether to enable dry run for rapid traverse 0: To disable 1: To enable
		1410	Dry run speed
		1420	Rapid traverse rate for individual axes
		1421	Rapid traverse override FO speed for individual axes
		1422	Maximum cutting feedrate (common to all axes)
		1423	Jog feedrate for individual axes
		1424	Manual rapid traverse feedrate for individual axes
		1425	FL speed at reference position return for individual axes
	ACCEL-ERATION/ DECEL-ERATION	1610#0	The cutting feed acceleration/deceleration used is: 0: Exponential type acceleration/deceleration 1: Post–interpolation linear–type acceleration/deceleration
		1620	Rapid–traverse linear–type acceleration/deceleration time constant for individual axes
		1622	Cutting feed acceleration/deceleration time constant for individual axes
		1624	Jog feed acceleration/deceleration time constant for individual axes
		1625	Jog feed exponential acceleration/deceleration FL speed for individual axes
MISCEL-LANY	MISCELLA-NY	3017	Reset signal output time
		3030	Allowable number of M code digits
		3701#1	Whether to use spindle serial interface for all axes 0: To use 1: Not to use
		7110	Number of manual pulse generators in use

12.1.22 Machining Condition Selecting

General

This function automatically specifies machining conditions when you simply select a precision level that matches your machining purposes when machining. The machining conditions are obtained based on the selected precision level and two prescribed parameter groups (precision–first and velocity–first).
The precision level can be selected from ten steps ranging from 1 (velocity–first) to 10 (precision–first).
It can be selected also on the screen or from the NC program.

> **NOTE**
> 1 This is an optional function.
> 2 This function is usable only with APC, AI contour control, and AI–APC.
> 3 This function is unusable on inch output machines (bit 0 of parameter No. 1001 = 1).

Screen

This function consists of the following two screens.

(1) Machining parameter tuning screen

On this screen, the following parameters are set up for both velocity–first (precision level 1) and precision–first (precision level 10) modes.

- Pre–interpolation acceleration/deceleration rate
- Bell–shaped acceleration/deceleration change time
- Allowable acceleration
- Post–interpolation acceleration/deceleration rate
- Corner feedrate difference
- Maximum allowable cutting feedrate
- Two items that can be set up at the user's discretion

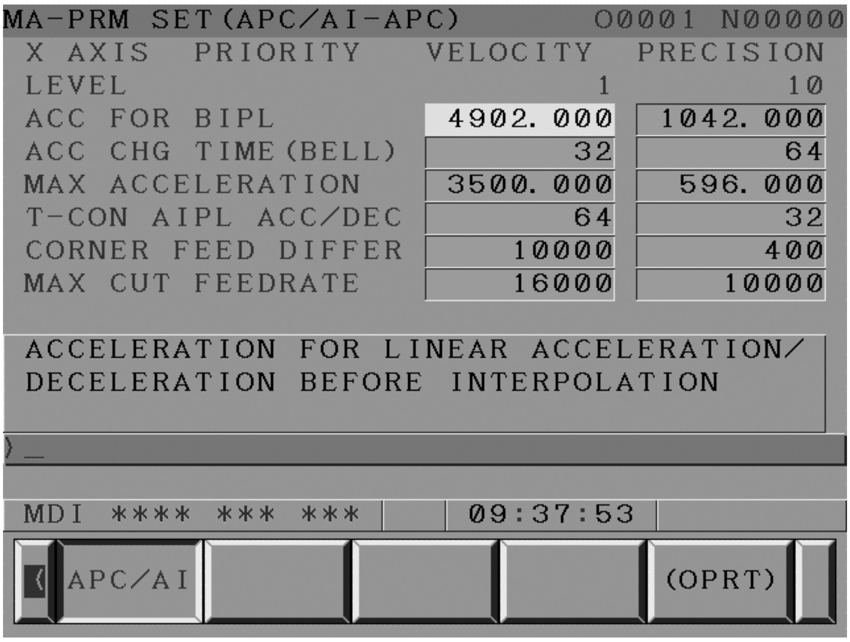
(2) Precision level select screen

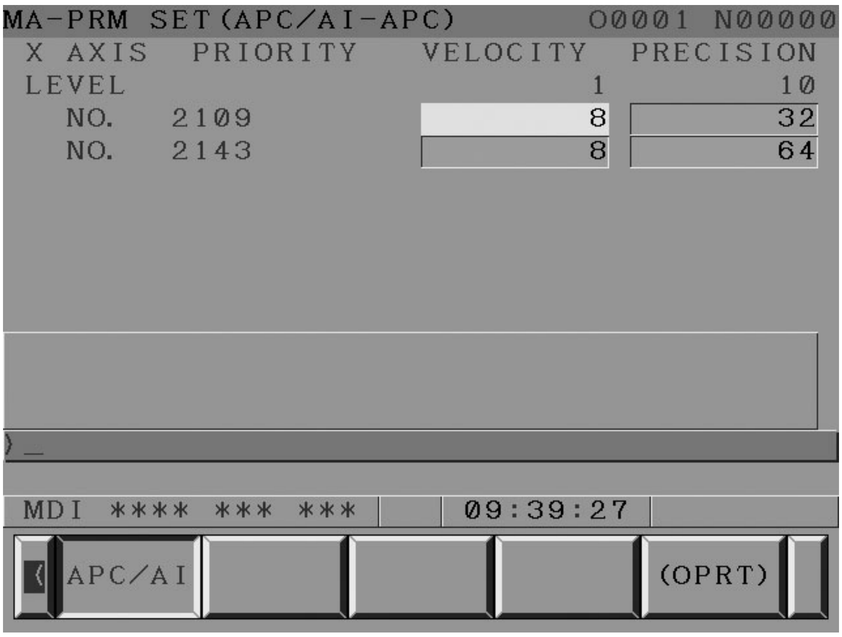
On this screen, a precision level that matches your machining purposes can be specified. Parameter values are calculated and displayed according to the precision level corresponding to the velocity–first parameter set (precision level 1) or the precision–first parameter set (precision level 10), whichever is selected.
The precision level can be changed also using program commands.

> **NOTE**
> 1 The bell–shaped acceleration change time is valid only if the pre–interpolation bell–shaped acceleration/deceleration option is available.
> 2 In APC, only the arc radius–based velocity clamp is changed according to the allowable acceleration, because no acceleration–based velocity determination function is available.

Machining parameter tuning screen

On this screen, the velocity–first parameter set (precision level 1) and precision–first parameter set (precision level 10) can be set up.
The screen is displayed by pressing: <SYSTEM> function key → [>] soft key (several times) → [M–TUN].
It can be called also from the parameter tuning screen (by selecting APC/AI TUNING on this screen).
Setting the CPR parameter (bit 0 of parameter No. 13601) to 1 hides this screen.





The velocity–first and precision–first parameters can be specified when PWE = 1 (parameter write enabled).

Pressing the [INIT] soft key and then the [EXEC] soft key initializes a cursor–selected item with a standard parameter. Pressing the [G_INIT] soft key and then the [EXEC] soft key initializes all items of a cursor–selected parameter set (velocity–first or precision–first). (Also this operation is usable only when PWE = 1.)

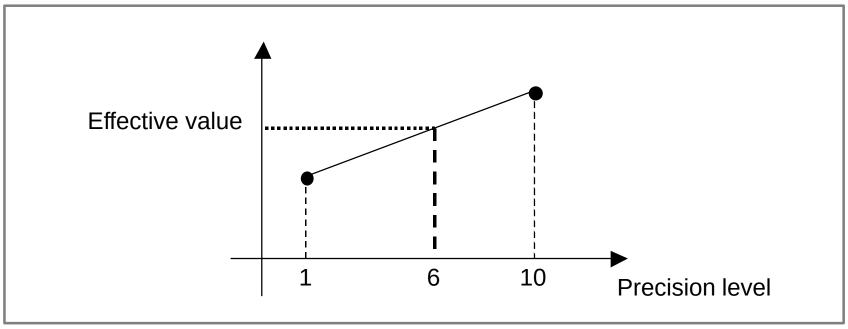
The following table lists the initial values.

Initial values

Item	Velocity–first (LV1)	Precision–first (LV10)	Unit
Pre–interpolation acceleration/ deceleration rate	4902.000	1042.000	mm/sec^2
Bell–shaped acceleration change time	32	64	msec
Allowable acceleration change	0	0	mm/sec^2
Post–interpolation acceleration/ deceleration time constant	24	24	msec
Corner feedrate difference	1000	400	mm/min
Maximum allowable cutting feedrate	10000	10000	mm/min

Precision level select screen

On this screen, it is possible to select a precision level at the midpoint between the velocity–first parameter setting (precision level 1) and the precision–first parameter setting (precision level 10). As shown below, the level changes linearly between the two extreme points. Selecting a level between them can set up an optimum parameter.



This screen is displayed by pressing: <OFFSET/SETTING> function key → [>] soft key (several times) → [PR_LEV].

The specified precision level value is not cleared by turning off the power (instead, it is saved to parameter No. 13634).

The precision level is initially '1'. It can be re–set even when PWE = 0 (parameter write disabled) or CPR (bit 0 or parameter No. 13601) = 1 (that is, the machining parameter tuning screen is hidden).

Screen soft keys

- SYSTEM
  - [M–TUN]
    - [APC/AI] : Machining parameter tuning screen (APC, AI–APC, and AI contour control)
      - [INIT] : Initial setting for a cursor–selected item
        - [EXEC]
        - [CAN]
      - [M–TUN] : Initial setting for a cursor–selected axis
        - [EXEC]
        - [CAN]
- OFFSET SETTING
  - [M–TUN]
    - [APC/AI] : Precision level select screen (APC, AI–APC, and AI contour control)

NOTE

1 The machine condition select function requires any of the APC, AI–APC, and AI contour control functions.
2 Using the bell–shaped acceleration change time for AI contour control requires the ”pre–read pre–interpolation bell–shaped acceleration/deceleration” option.
3 The effective value of items is rewritten when:
   – The precision level is changed, or
   – The velocity–first or precision–first parameter for the respective items is changed (including the [INIT] and [G_INIT] soft keys).
4 If a parameter corresponding to a certain effective value is changed, the parameter value may differ from a value obtained from the velocity–first level, precision–first level, or the current precision level.
5 The velocity–first or precision–first parameter group, whichever is selected with the cursor, for the currently displayed axis can be changed at a time by pressing: [G_INIT] R [EXEC].
6 Entering values on the machining parameter tuning screen or precision level select screen causes automatic proportional allotment to be performed for the parameter to be tuned. If the automatic proportional allotment fails, leading to an incorrect calculation result, a waning message meaning that automatic setting has failed appears.
7 On the machining condition select function screen, an independent PMC axis can be neither displayed nor tuned.

Changing the precision level, using a program

The precision level can be changed not only on the precision level select screen but also using the following program.

```
G05. 1 Q1 Rx;    (level selecting, using an AI contour control or
                  AI–APC mode command)
G08 P1 Rx;       (level selecting, using an APC mode command)
x...Level (1 to 10)
```

NOTE

Once the precision level is specified, it remains in effect even when the AI contour control, AI–APC, and APC modes are canceled.

Setting item

- **Pre–interpolation acceleration/deceleration rate**

This item is used to set up a linear–portion acceleration for pre–interpolation acceleration/deceleration (in mm/s^2).

Setting range: 50.000 to 99999.999 (mm/s^2)
Size: 2–word type

The parameter value set up on the machining parameter tuning screen is saved to the following parameters:

No. 13610: Velocity–first parameter
No. 13611: Precision–first parameter

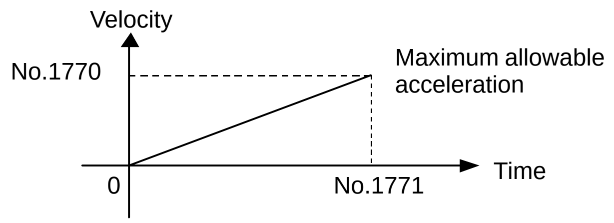
In addition, the following parameters are set up from the precision level:

No. 1770/1771 (arbitrary integral ratio)

$$\text{Effective value} = \frac{\text{No.1770}}{\text{No.1771}} \times \frac{100}{6}$$ (arbitrary integral ratio for millimeter machines)

No. 1770: Maximum allowable machining rate for pre–interpolation linear acceleration/deceleration (in mm/min)

No. 1771: Time elapsed before the maximum allowable machining feedrate for pre–interpolation linear acceleration/deceleration is attained (in ms)



> **NOTE**
> The acceleration setting is common to all axes.

- **Bell–shaped acceleration change time**

This item is used to set up a time constant for the bell–shaped portion of pre–read pre–interpolation acceleration/deceleration (in ms).

Setting range: 0 to 100 [ms]
Size: Byte type

The parameter value set up on the machining parameter tuning screen is reflected on the following parameters:

No. 13612: Velocity–first parameter
No. 13613: Precision–first parameter

In addition, the following parameter is set up according to the precision level:

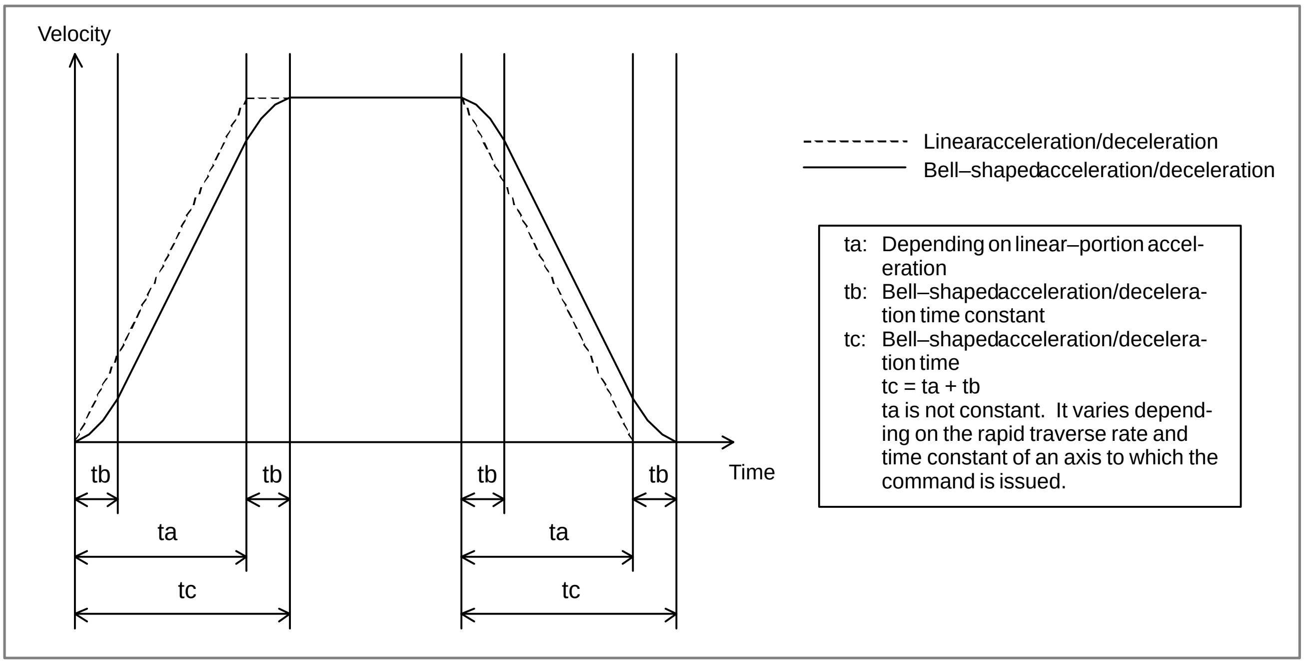
(AI contour control)
No. 1772: Bell–shaped acceleration/deceleration time constant with a constant pre–read pre–interpolation acceleration time (in ms)

If this item is set with a nonzero value, the following parameter is also set up:

BEL (bit 7 of parameter No. 1603) = 1 (AI contour control only)

NOTE

1 This item is disabled for APC and AI APC.
2 The time constant mentioned above is common to all axes. Changing this item results in the settings of all axes being changed.



- **Allowable acceleration**

This item is used to set up an allowable acceleration in acceleration–based velocity setup (in mm/s^2).

Setting range: 0 to 99999.999 (mm/s^2 or degrees/s^2)
Size: 2–word type

For AI contour control and AI APC, the arc radius–based feedrate clamp setting is also changed automatically.
The parameter value specified on the machining parameter tuning screen is reflected on the following parameters:
(APC control/AI contour control/AI APC)
No. 13620: Velocity–first parameter
No. 13621: Precision–first parameter

In addition, the following parameter is set up according to the precision level:

- APC (for rotation axes and millimeter machine linear axes)

$$\text{Effective value} = \frac{(\text{No.}1730)^2}{\text{No.}1731} \times \frac{10}{36} \quad (\text{IS–B})$$

$$= \frac{(\text{No.}1730)^2}{\text{No.}1731} \times \frac{100}{36} \quad (\text{IS–C})$$

Note: Parameter Nos. 1730 and 1731 are set/re–set automatically only when MCR (bit 0 of parameter No. 13600) = 0.

- AI contour control and AI APC (for rotation axes and millimeter machine linear axes)

$$\text{Effective value} = \frac{\text{No.1432}}{\text{No.1785}} \times \frac{100}{6} = \frac{(\text{No.1730})^2}{\text{No.1731}} \times \frac{10}{36} \quad \text{(IS–B)}$$

$$= \frac{(\text{No.1730})^2}{\text{No.1731}} \times \frac{100}{36} \quad \text{(IS–C)}$$

Note: Parameter Nos. 1730 and 1731 are set/re–set automatically only when MCR (bit 0 of parameter No. 13600) = 0.

No. 1785: Time elapsed before the maximum allowable cutting feedrate (No. 1432) is attained (in ms)

No. 1482: Maximum allowable cutting feedrate for individual axes during APC mode (in mm/min or deg/min)

No. 1730: Upper limit to the feedrate for an arc with a radius of R (in mm/min)

No. 1731: Arc radius corresponding to the upper limit to the feedrate (in 0.001 mm)

NOTE

1 For AI contour control and AI APC, the arc radius–based feedrate clamp is automatically set or re–set if MCR (bit 0 of parameter No. 13600) = 0.

2 Parameter No. 1785 (acceleration–based velocity determination) is unusable during APC. Only the arc radius–based velocity clamp (parameter Nos. 1730 and 1731) is set up. This setting is disabled if MCR (bit 0 of parameter No. 13600) = 1.

3 If a different allowable acceleration is specified for each axis, the arc radius–based velocity clamp parameter is set up using the least nonzero allowable acceleration.

- **Post–interpolation acceleration/deceleration time constant**

This item is used to specify a post–interpolation acceleration/deceleration time constant (in ms).
Its type (linear or bell–shaped) is selected according to the settings of parameters BS2 (bit 3 of parameter No. 1602) and LS2 (bit 6 of parameter No. 1602).

Parameter No. 1602		Acceleration/deceleration type
LS2(#6)	BS2(#3)	
1	0	Post–interpolation linear acceleration/deceleration is selected for cutting feed.
0	1	Post–interpolation bell–shaped acceleration/deceleration is selected for cutting feed.

Data range: 8 to 512 [ms]
Size: Word type (common to all axes)

The parameter value specified on the machine parameter tuning screen is reflected on the following parameters (common to all modes):

No. 13622: Velocity–first parameter
No. 13623: Precision–first parameter

In addition, the following parameter is set up according to the precision level (common to all modes):

No. 1769: Post–interpolation linear cutting feed acceleration/ deceleration or post–interpolation bell–shaped cutting feed acceleration/deceleration (specific to each axis)

NOTE
If this parameter is 0 for all axes, parameter No. 1768 (post–interpolation linear acceleration/deceleration time constant common to all axes) is enabled.

- **Corner feedrate difference**

This item is used to specify an allowable feedrate difference for corner feedrate difference–based velocity determination.

If it is anticipated that this setting may be exceeded by the velocity component of an axis at a block boundary, a feedrate that can prevent the setting from being exceeded is obtained and the axis is decelerated to the feedrate, using pre–interpolation acceleration/deceleration.

[Unit of data]

[Valid data range]

Type	Increment system	Unit of data	Valid data range	
			IS–B	IS–C
APC/ AI contour control/ AI APC	Millimeter machine	1 mm/min	6 to 15000	6 to 12000
	Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Size: Word axis type

The parameter value specified on the machining parameter tuning screen is reflected on the following parameters:
(APC control/AI contour control/AI APC)

No. 13624: Velocity–first parameter
No. 13625: Precision–first parameter

In addition, the following parameter is set up according to the precision level:
(APC control/AI contour control/AI APC)

No. 1788: Allowable individual–axis feedrate difference for feedrate difference–based automatic deceleration at corner

NOTE
1 This setting item is axis–specific. The item for an axis is not interlocked with that for any other axis. So it is necessary to set up the item for all axes individually.
2 If this item is specified as 0 for all axes, no deceleration is performed at corners.
3 If this parameter is 0 for all axes, parameter No. 1780 (feedrate difference–based allowable feedrate difference of the automatic deceleration at corners common to all axes) is valid for APC.

- **Maximum allowable machining feedrate**

This item is used to specify an axis–specific maximum allowable machining feedrate.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	0 to 240000	0 to 100000
Rotaion axis	1 deg/min	0 to 240000	0 to 100000

Size: 2–word axis type

The parameter value specified on the machining parameter tuning screen is reflected on the following parameters (common to all axes):

No. 13626: Velocity–first parameter
No. 13627: Precision–first parameter

In addition, the following parameter is set up according to the precision level (the setting is common to all axes):

No. 1432: Maximum allowable individual–axis feedrate to be applied during APC control/AI contour control/AI APC mode

- **Arbitrary items**

It is possible to register two different types of arbitrary parameters. Each of them can be associated with a CNC parameter or servo parameter (except for a bit type). They are assigned parameter Nos., using a parameter.

- Setting
  The following table summarizes how each arbitrary item is specified with a parameter No., velocity–first (precision level 1), and precision–first (precision level 10) settings.

	Parameter No.	Velocity–first (precision level 1) value setting	Precision–first (precision level 10) value setting
Arbitrary item (1)	No. 13628	No. 13630	No. 13632
Arbitrary item (2)	No. 13639	No. 13631	No. 13633

- Display
  The target tuning parameter number is displayed.

NOTE

Any of the following parameter Nos. cannot be specified for arbitrary items.
– Bit–type parameters
– Spindle parameters (Nos. 4000 to 4799)
– Power–off parameter
– Nonexistent parameters

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13600								MCR

[Data type] Bit

MCR When the permissible acceleration is adjusted with the machining condition selection function (machining parameter adjustment screen or precision level selection screen), parameter Nos. No.1730 and 1731, which are related to feedrate clamping by arc radius, are:

0 : Changed.
1 : Not changed.

	#7	#6	#5	#4	#3	#2	#1	#0
13601								MPR

[Data type] Bit

MPR The machining parameter adjustment screen is:

0 : Displayed.
1 : Not displayed.

> **NOTE**
> 1 When this parameter has been set, the power must be turned off before operation is continued.
> 2 Even when this parameter is set to 1, the precision level selection screen is displayed.

13610	Acceleration rate of acceleration/deceleration before interpolation when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 1)
13611	Acceleration rate of acceleration/deceleration before interpolation when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 10)

[Data type] 2–word

[Unit of data] %

Increment system	Unit
Millimeter machine	0.001mm/sec^2

[Valid data range] 50000 to 99999999

These parameters set the acceleration rate of acceleration/deceleration before interpolation in advanced preview control, AI advanced preview control, or AI contour control. Two levels including precision level 1, which places emphasis on speed, and precision level 10, which places emphasis on precision, can be set.

13612	Acceleration change time when AI contour control is used (bell–shaped) (precision level 1)
13613	Acceleration change time when AI contour control is used (bell–shaped) (precision level 10)

[Data type] Byte

[Unit of data] msec

[Valid data range] 1 to 100

These parameters set an acceleration change time (bell–shaped) with emphasis placed on speed (precision level 1) and an acceleration change time (bell–shaped) with emphasis placed on precision (precision level 10) in AI contour control.

13620	Permissible acceleration when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 1)
13621	Permissible acceleration when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 10)

[Data type] 2–word axis

[Unit of data]

Increment system	Unit
Millimeter machine	0.001mm/sec^2

[Valid data range] 0 to 99999999

These parameters set a permissible acceleration with emphasis placed on speed (precision level 1) and a permissible acceleration with emphasis placed on precision (precision level 10) in advanced preview control, AI advanced preview control, or AI contour control.

13622	Time constant of acceleration/deceleration after interpolation (precision level 1)
13623	Time constant of acceleration/deceleration after interpolation (precision level 10)

[Data type] Word axis

[Unit of data] msec

[Valid data range] See the description of parameter No. 1768.

These parameters set a time constant of linear acceleration/deceleration after interpolation with emphasis placed on speed (precision level 1) and a time constant of linear acceleration/deceleration after interpolation with emphasis placed on precision (precision level 10).
The linear or bell–shaped type is selected by bit 3 (BS2) and bit 6 (LS2) of parameter No. 1602.

Parameter No. 1602		Acceleration/deceleration
LS2(#6)	BS2(#3)	
1	0	Selects linear acceleration/deceleration after cutting feed interpolation.
0	1	Selects bell–shaped acceleration/deceleration after cutting feed interpolation.

NOTE
1 For bell–shaped acceleration/deceleration, the function for bell–shaped acceleration/deceleration after cutting feed interpolation is required.
2 The same parameters are used in advanced preview control, AI advanced preview control, and AI contour control.

13624	Difference in corner speed when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 1)
13625	Difference in corner speed when advanced preview control, AI advanced preview control, or AI contour control is used (precision level 10)

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

These parameters set a permissible speed difference with emphasis placed on speed (precision level 1) and a permissible speed difference with emphasis placed on precision (precision level 10) when the speed is determined by a corner speed difference in advanced preview control, AI advanced preview control, or AI contour control.

13626	Maximum machining speed (precision level 1)
13627	Maximum machining speed (precision level 10)

[Data type] 2–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	6 to 24000	6 to 100000
Rotaion axis	1 deg/min	6 to 24000	6 to 100000

These parameters set the maximum machining speed for each axis.

13628	Parameter number for arbitrary item 1 when advanced preview control, AI advanced preview control, or AI contour control is used
13629	Parameter number for arbitrary item 2 when advanced preview control, AI advanced preview control, or AI contour control is used

[Data type] Word

[Valid data range] 1 to 65535

These parameters specify parameter numbers corresponding to arbitrary items 1 and 2.

NOTE

1 You cannot specify the numbers of the following parameters:
   – Bit parameters
   – Spindle parameters (Nos. 4000 to 4799)
   – Parameters requiring power disconnection (P/S 0 alarm is issued for these parameters.)
   – Nonexistent parameters

2 When such a parameter is set, the power must be turned off before operation is continued.

13630	Value of the parameter corresponding to arbitrary item 1 with emphasis placed on speed (precision level 1) when advanced preview control, AI advanced preview control, or AI contour control is used
13631	Value of the parameter corresponding to arbitrary item 2 with emphasis placed on speed (precision level 1) when advanced preview control, AI advanced preview control, or AI contour control is used
13632	Value of the parameter corresponding to arbitrary item 1 with emphasis placed on speed (precision level 10) when advanced preview control, AI advanced preview control, or AI contour control is used
13633	Value of the parameter corresponding to arbitrary item 2 with emphasis placed on speed (precision level 10) when advanced preview control, AI advanced preview control, or AI contour control is used

[Data type] 2–word axis

[Unit of data] Depending on the type of the parameter for an item

[Valid data range] Depending on the type of the parameter for an item

13634	Precision level currently selected when advanced preview control, AI advanced preview control, or AI contour control is used

[Data type] Byte

[Valid data range] 1 to 10

The currently selected level is set.

12.1.23 Other Functions

Suppressing display of the screens displayed by the <SYSTEM> function key

When bit 0 (SYY) of setting parameter No. 3208 is set to disable the <SYSTEM> function key on the MDI panel, display of the screens and soft keys of the functions that belong to the <SYSTEM> function key can be suppressed.

Screen switching by the macro executor is not disabled.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3208								SKY

[Data type] Bit

SKY The <SYSTEM> function key is:

0 : Not disabled.
1 : Disabled.

12.2 EDIT

12.2.1 Part Program Storage Length

General

One of the following part program size can be selected.

Part program size \ CNC model		Series 0*i*		Series 0*i* Mate
		Package A	Package B	
320m	(128Kbyte)	—	○	—
640m	(256Kbyte)	○	—	○

○: Standard, —: Not Available

NOTE
The memory space values are listed as guidelines.
The actual size of a program that can be registered varies with the number or sizes of the registered programs.

Alarm and message

Number	Message	Description
070	NO PROGRAM SPACE IN MEMORY	Not enough program space remaining. Delete any unnecessary programs, then retry.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.3.1	Displaying Memory Used and a List of Programs
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.3.1	Displaying Memory Used and a List of Programs

12.2.2 No. of Registered Programs

General

The number of registered programs is 400.

Alarm and message

Number	Message	Description
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 400. Delete unnecessary programs and execute program registeration again.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR’S MANUAL (T series) (B–64114EN)	III.11.3.1	Displaying Memory Used and a List of Programs
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR’S MANUAL (T series) (B–64134EN)	III.11.3.1	Displaying Memory Used and a List of Programs

12.2.3 Memory Protection Key

General

A key called the data protection key is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously.

Signal

Memory protection signal KEY1 to KEY4 <G046#3 to #6>

[Classification] Input signal

[Function] Enables the changing of the memory contents from the MDI panel. Four signals are provided. The operations that can be performed on the contents of memory by each signal vary depending on the setting of bit 7 (KEY) of parameter No. 3290.

When KEY = 0

- KEY1: Enables the input of tool compensation values and the workpiece zero point offset values, and workpiece coordinate systems shift amount.
- KEY2: Enables the input of setting data and macro variables, and tool life management data.
- KEY3: Enables program loading and editing.
- KEY4: Enables PMC data (counter data tables)

When KEY = 1

- KEY1: Enables program loading and editing, as well as the input of PMC parameters.
- KEY2 to KEY4: Not used

[Operation] When a signal is set to 0, the associated operations are disabled.
When a signal is set to 1, the associated operations are enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046		KEY4	KEY3	KEY2	KEY1			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3290	KEY							

KEY For memory protection keys:
0 : The KEY1, KEY2, KEY3, and KEY4 signals are used.
1 : Only the KEY1 signal is used.

> **NOTE**
> The functions of the signals depend on whether KEY = 0 or KEY = 1.

Alarm and message

Warning message	Contents
WRITE PROTECT	Data input is invalid because of memory protect signal.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11	SETTING AND DISPLAYING DATA
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11	SETTING AND DISPLAYING DATA
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11	SETTING AND DISPLAYING DATA
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11	SETTING AND DISPLAYING DATA

12.2.4 Password Function

General

The password function locks NE9 (bit 4 of parameter No. 3202), used to protect program Nos. 9000 to 9999, by using the PASSWD (No. 3210) and KEYWD (No. 3211) parameters. When NE9 is locked, NE9 cannot be set to 0. Therefore, the protection for programs numbered 9000 to 9999 cannot be released unless the correct keyword is entered.

NE9 is locked when different values are set in the PASSWD and KEYWD parameters. The values set in the two parameters are not displayed. NE9 is unlocked when the value preset in the PASSWD parameter is set in the KEYWD parameter. When 0 is indicated for the PASSWD parameter, a value has not yet been set for PASSWD.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				

[Data type] Bit

NE9 Editing of subprograms with program numbers 9000 to 9999

0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)

(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

3210	Password(PASSWD)

[Data type] Two–word

Set a password to this parameter. Its value is not displayed.

> **CAUTION**
> This parameter shows 0, when no value is set to this parameter. Once a key is locked, parameter NE9 cannot become 0 and PASSWD cannot be changed unless you perform an unlock operation or perform the memory all clear operation. When an attempt is made to modify the password by MDI input operation in this state, the warning message “WRITE PROTECTED” is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), P/S alarm No. 231 is issued.

3211	Keyword(KEYWD)

[Data type] Two–word

When the value set as the password (set in parameter No. 3210) is set the same as this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No. 3202 becomes 0.

> **NOTE**
> The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

Alarm and message

Number	Message	Description
231	FORMAT ERROR IN G10 L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1) Address N or R was not entered. 2) A number not specified for a parameter was entered. 3) The axis number was too large. 4) An axis number was not specified in the axis–type parameter. 5) An axis number was specified in the parameter which is not an axis type. 6) An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PASSWD) when they are protected by a password. Correct the program.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	III.9.9	PASSWORD FUNCTION
	OPERATOR’S MANUAL (T series) (B–64114EN)	III.9.9	PASSWORD FUNCTION
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	III.9.9	PASSWORD FUNCTION
	OPERATOR’S MANUAL (T series) (B–64134EN)	III.9.9	PASSWORD FUNCTION

12.2.5 Background Editing

General

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory.
During background editing, all programs cannot be deleted at once.

Alarm and message

Number	Message	Description
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. Use background editing correctly.
239	BP/S alarm	Background editing was performed while the external punch was being executed in external I/O device control.
240	BP/S alarm	Background editing was done while in MDI operation.

NOTE
Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.9.8	BACKGROUND EDITING
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.9.8	BACKGROUND EDITING
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.9.8	BACKGROUND EDITING
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.9.8	BACKGROUND EDITING

12.2.6 Playback

General

In The **TEACH IN JOG** mode (TJOG) and **TEACH IN HANDLE** mode (THND), a machine position along the X, Y, and Z axes obtained by manual operation is stored in memory as a program position to create a program.
The words other than X, Y, and Z, which include O, N, G, R, F, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	

[Data type] Bit type

THD Manual handle feed in TEACH IN JOG mode

0 : Valid
1 : Invalid

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
CONNECTION MANUAL (This manual)		2.6	MODE SELECTION

12.2.7 Conversational Programming with Graphic Function

General

Programs can be created block after block on the conversational screen while displaying the G code menu.
Blocks in a program can be modified, inserted, or deleted using the G code menu and converstional screen.

13 INPUT/OUTPUT OF DATA

13.1 READER/PUNCHER INTERFACE

General

The data shown below can be input/output through reader/puncher interface.

1. Program
2. Offset data
3. Parameter
4. Pitch error compensation data
5. Custom macro common variables.

Setting the IO4 parameter (bit 4 of No. 0110) enables I/O channels to be subjected to separate control. In this case, the I/O channels can be categorized into four types, that is, foreground input, foreground output, background input, and background output. When a DNC operation is in the foreground mode, for example, programs can be input/output in the background editing mode.

Parameter

The parameters described below must be set up to use an I/O unit interface (RS–232–C serial port) or memory card interface for inputting and outputting data (such as programs and parameters) between external input/output units and memory cards.
The I/O CHANNEL setting parameter is used to select a desired input/output unit by specifying the channel (RS–232–C serial port 1 or RS–232–C serial port 2) to which the input/output unit is connected.
The same setting is made when using the memory card interface.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel of I/O device interface must be set in parameters for that channel in advance.
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the input/output device interface parameters for the channels of I/O device interface.

Input/output channel number (parameter No. 0020)
↓

0020 I/O CHANNEL			
Specify a channel for an input/output device. I/ O CHANNEL =0 : Channel1 =1 : Channel1 =2 : Channel2	I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
		0102	Number specified for the input/output device
		0103	Baud rate
	I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
		0112	Number specified for the input/output device
		0113	Baud rate
	I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
		0122	Number specified for the input/output device
		0123	Baud rate

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

This parameter can be entered on the setting screen

[Data type] Bit

TVC TV check

0 : Not performed
1 : Performed

ISO Code used for data output

0 : EIA code
1 : ISO code

0020	I/O CHANNEL: Selection of an input/output device or selection of input device in the foreground

Setting entry is acceptable.

[Data type] Byte

[Valid data range] 0 to 35

I/O CHANNEL: Selection of the input/output device to be used

The CNC provides the following interfaces for data transfer to and from the host computer and external input/output devices:

- Input/output device interface (RS–232–C serial port 1, 2)
- DNC2 interface

Data input/output can be performed with a personal computer connected via FOCAS1/Ethernet or FOCAS1/HSSB. Data input/output can be performed with the Power Mate CNC via the FANUC I/O Link.

This parameter selects the interface used to transfer data to and from an input/output device.

Setting	Description
0, 1	RS–232–C serial port 1
2	RS–232–C serial port 2
4	Memory card interface
5	Data server interface
6	The DNC operation is performed or M198 is specified by FOCAS1/ Ethernet.
10	DNC2 interface, OSI–Ethernet
15	M198 is specified by FOCAS1/HSSB. (Bit 1 (NWD) of parameter No. 8706) must also be specified.)
20 21 22 to 34 35	Group 0 Group 1 Group 2 to Group 14 Group 15 Data is transferred between the CNC and a Power Mate CNC in group n (n: 0 to 15) via the FANUC I/O Link.

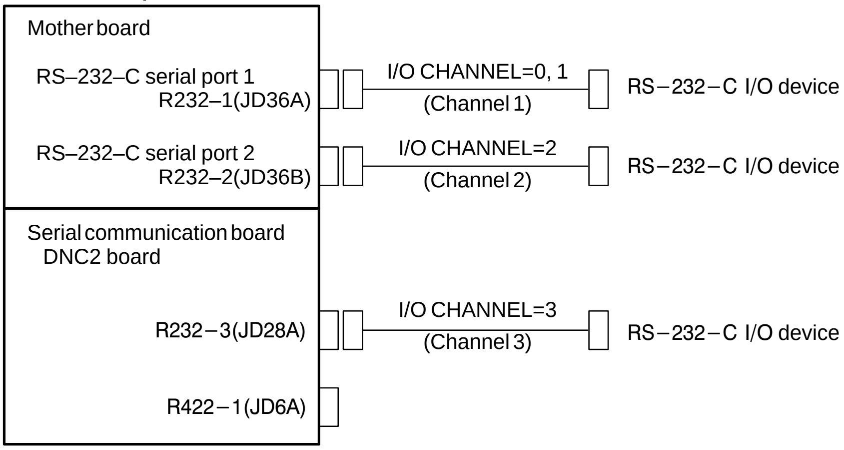
Supplemental remark 1

If the DNC operation is performed with FOCAS1/HSSB, the setting of parameter No. 20 does not matter. The DMMC signal <G042.7> is used.

Supplemental remark 2

If bit 0 (IO4) of parameter No. 110 is set to control the I/O channels separately, the I/O channels can be divided into four types: input and output in the foreground and input and output in the background. If so, parameter No. 20 becomes a parameter for selecting the input device in the foreground.

NOTE
1 An input/output device can also be selected using the setting screen. Usually, the setting screen is used.
2 The specifications (such as the baud rate and the number of stop bits) of the input/output devices to be connected must be set in the corresponding parameters for each interface beforehand. I/O CHANNEL = 0 and I/O CHANNEL = 1 represent input/output devices connected to RS–232C serial port 1. Separate parameters for the baud rate, stop bits, and other specifications are provided for each channel.



3 The input/output unit interface may be referred to as the reader/punch interface. RS–232C serial port 1 and RS–232C serial port 2 are also referred to as channel 1 and channel 2, respectively.
4 JD36A, JD36B, JD28A, and JD6A represent the connector numbers on the printed–circuit board.

0021	Setting of the output device in the foreground
0022	Setting of the input device in the background
0023	Setting of the output device in the background

This parameter can be entered on the setting screen

[Data type] Byte

[Valid data range] 0 to 2, 5, 10

These parameters are valid only when bit 0 (IO4) of parameter No. 110 is set to control the I/O channels separately.

The parameters set individual input/output devices if the I/O channels are divided into these four types: input and output in the foreground and input and output in the background. The input device in the foreground is set in parameter No. 20. For the details of the settings, see the table provided with the description of parameter No. 20.

NOTE
If different input/output devices are simultaneously used in the foreground and background, just a value from 0 to 2 can be specified for the background device.
If an attempt is made to use a busy input/output device, an alarm (P/S233 or BP/S233) will be raised. Note that the settings 0 and 1 indicate the same input/output device.

Input/output channel number (parameter No.0020)
↓

0020	I/O CHANNEL

Specify a channel for an input/output device.

I/ O CHANNEL
=0 : Channel1
=1 : Channel1
=2 : Channel2

I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
	0102	Number specified for the input/ output device
	0103	Baud rate
I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
	0112	Number specified for the input/ output device
	0113	Baud rate
I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
	0122	Number specified for the input/ output device
	0123	Baud rate

Fig.13.1 I/O Device Interface Settings

(1) Parameters Common to all Channels

0024	Port for communication with the PMC ladder development tool (FAPT LADDER–III)

This parameter can be entered on the setting screen

[Data type] Byte

This parameter sets the port to be used for communication with the PMC ladder development tool (FAPT LADDER–III).

0 : According to the setting on the PMC online screen
1 : RS–232C serial port 1 (JD36A)
2 : RS–232C serial port 2 (JD36B)
10 : High–speed interface (HSSB(COP7) or Ethernet)
11 : High–speed interface or RS–232–C serial port 1
12 : High–speed interface or RS–232–C serial port 2

	#7	#6	#5	#4	#3	#2	#1	#0
0100	ENS	IOP	ND3		NCR	CRF	CTV	

[Data type] Bit

CTV: Character counting for TV check in the comment section of a program.
0 : Performed
1 : Not performed

CRF EOB (end of block) to be output in the ISO code:
0: Depends on the setting of bit 3 (NCR) of parameter No. 100.
1: is “CR”“LF”.

Note) The EOB output patterns are as shown below:

NCR	CRF	EOB output format
0	0	“LF” “CR” “CR”
0	1	“CR” “LF”
1	0	“LF”
1	1	“CR” “LF”

NCR Output of the end of block (EOB) in ISO code
0 : LF, CR, CR are output.
1 : Only LF is output.

ND3 In DNC operation, a program is:
0 : Read block by block. (A DC3 code is output for each block.)
1 : Read continuously until the buffer becomes full. (A DC3 code is output when the buffer becomes full.)

> **NOTE**
> In general, reading is performed more efficiently when ND3 set to 1. This specification reduces the number of buffering interruptions caused by reading of a series of blocks specifying short movements. This in turn reduces the effective cycle time.

IOP Specifies how to stop program input/output operations.
0 : An NC reset can stop program input/output operations.
1 : Only the [**STOP**] soft key can stop program input/output operations. (An reset cannot stop program input/output operations.)

ENS Action taken when a NULL code is found during read of EIA code
0 : An alarm is generated.
1 : The NULL code is ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
0110								IO4

[Data type] Bit

IO4 Separate control of I/O channel numbers is:
0: Not performed.
1: Performed.

If the I/O channels are not separately controlled, set the input/output device in parameter No. 20.

If the I/O channels are separately controlled, set the input device and output device in the foreground and the input device and output device in the background in parameters No. 20 to No. 23 respectively.

Separate control of I/O channels makes it possible to perform background editing, program input/output, and the like during the DNC operation.

(2) Parameters of Channel 1 (I/O CHANNEL=0)

	#7	#6	#5	#4	#3	#2	#1	#0
0101	NFD				ASI			SB2
	NFD				ASI		HAD	SB2

[Data type] Bit

SB2 The number of stop bits
0 : 1
1 : 2

HAD An alarm raised for the internal handy file is:
0: Not displayed in detail on the NC screen. (PS alarm 86 is displayed.)
1: Displayed in detail on the NC screen.

ASI Code used at data input
0 : EIA or ISO code (automatically distinguished)
1 : ASCII code

NFD Feed before and after the data at data output
0 : Output
1 : Not output

NOTE
When input/output devices other than the FANUC PPR are used, set NFD to 1.

0102	Number specified for the input/output device (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 0, with one of the set values listed in Table 13.1 (a).

Table 13.1 (a) Set value and Input/Output Device

Set value	Input/output device
0	RS–232–C (Used control codes DC1 to DC4)
1	FANUC CASSETTE ADAPTOR 1 (FANUC CASSETTE B1/ B2)
2	FANUC CASSETTE ADAPTOR 3 (FANUC CASSETTE F1)
3	FANUC PROGRAM FILE Mate, FANUC FA Card Adaptor FANUC FLOPPY CASSETTE ADAPTOR, FANUC Handy File FANUC SYSTEM P-MODEL H
4	RS–232–C (Not used control codes DC1 to DC4)
5	Portable tape reader
6	FANUC PPR FANUC SYSTEM P-MODEL G, FANUC SYSTEM P-MODEL H

0103	Baud rate (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set baud rate of the input/output device used when the I/O CHANNEL is set to 0, with a set value in Table 13.1 (b).

Table 13.1 (b)

Set value	Baud rate (bps)	Set value	Baud rate (bps)
1	50	7	600
2	100	8	1200
3	110	9	2400
4	150	10	4800
5	200	11	9600
6	300	12	19200

(3) Parameters of Channel 1 (I/O CHANNEL=1)

	#7	#6	#5	#4	#3	#2	#1	#0
0111	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 1. The meanings of the bits are the same as for parameter 0101.

0112	Number specified for the input/output device (when I/O CHANNEL is set to 1)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 1, with one of the set values listed in Table 13.1 (a).

0113	Baud rate (when I/O CHANNEL is set to 1)

[Data type] Byte

Set the baud rate of the input/output device used when I/O CHANNEL is set to 1, with a value in Table 13.1 (b).

(4) Parameters of Channel 2 (I/O CHANNEL=2)

	#7	#6	#5	#4	#3	#2	#1	#0
0121	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 2. The meanings of the bits are the same as for parameter 0101.

0122	Number specified for the input/output device (when I/O CHANNEL is set to 2)

[Data type] Byte

Set the number specified for the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (a).

0123	Baud rate (when the I/O CHANNEL is set to 2)

[Data type] Byte

Set the baud rate of the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (b).

	#7	#6	#5	#4	#3	#2	#1	#0
0134				NCD			PRY	

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

[Data type] Bit

PRY Parity bit
0: Not used
1: Used

NCD CD (signal quality detection) of the RS–232C interface
0: Checked
1: Not checked

	#7	#6	#5	#4	#3	#2	#1	#0
0135								ASC

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

[Data type] Bit

ASC Communication code except NC data
0: ISO code
1: ASCII code

(5) Parameters of Memory Card Interface

	#7	#6	#5	#4	#3	#2	#1	#0
0138	MDN							

[Data type] Bit

MDN The DNC operation function by a memory card is:
0: Disabled.
1: Enabled. (A PCMCIA card attachment is required.)

> **NOTE**
> Use a PCMCIA card attachment suited to the CNC to secure the memory card in the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
0300								PCM

[Data type] Bit

PCM If the CNC screen display function is enabled, when a memory card interface is provided on the NC side (HSSB connection),
0: The memory card interface on the NC side is used.
1: The memory card interface on the PC side is used.

This parameter is valid when parameter No. 20 is set to 4 (memory card interface). This parameter is valid only while the CNC screen display function is active.

	#7	#6	#5	#4	#3	#2	#1	#0
0138		OWN						

[Data type] Bit

OWN When NC data or an NC program is output to a memory card, a message for file overwrite confirmation is:
0: Displayed.
1: Not displayed.

> **NOTE**
> For file overwrite operation using the screen hardcopy function, maintenance information screen, and PMC screen, the confirmation message is not displayed.

Alarm and message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
5227	FILE NOT FOUND	A file specified for communication with the built–in handy file unit is missing.
5228	SAME NAME USED	There are duplicate names for files in the built–in handy file unit.
5229	WRITE PROTECTED	A floppy disk in the built–in handy file unit is write–protected.
5230	FILE PROTECTED	In communication with the built–in handy file unit, a file is write– or read–protected.
5231	TOO MANY FILES	In communication with the built–in handy file unit, a limit to the number of files has been exceeded.
5232	DATA OVER–FLOW	There is no free space on a floppy disk in the built–in handy file unit.
5233	TO MANY FD VOLUME	In communication with the built–in handy file unit, a multivolume limit has been exceeded.
5234	FORMAT ERROR	A floppy disk format error has occurred on the built–in handy file unit.

Number	Message	Description
5235	COMMUNICATION ERROR	In communication with the built–in handy file unit, a communication error has occurred.
5236	COMMAND ERROR	In communication with the built–in handy file unit, an incorrect command was issued.
5237	READ ERROR	It is impossible to read from a floppy disk in the built–in handy file unit. It is likely that the floppy disk is abnormal, the magnetic head is dirty, or the hand file unit is defective.
5238	WRITE ERROR	It is impossible to write to a floppy disk in the built–in handy file unit. It is likely that the floppy disk is abnormal, the magnetic head is dirty, or the hand file unit is defective.
5239	SUBPROGRAM ERROR	In communication with the built–in handy file, a subprogram call function was used with a floppy disk, but there is no subprogram file on the floppy disk.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.8	DATA INPUT/OUTPUT
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.8	DATA INPUT/OUTPUT
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.8	DATA INPUT/OUTPUT
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.8	DATA INPUT/OUTPUT

13.2 DNC2 INTERFACE

Refer to an item of FANUC DNC2 DESCRIPTIONS (B–61992E) for detailed information of DNC2 interface.

13.3 EXTERNAL I/O DEVICE CONTROL

General

It is possible to request from an external source that a program be registered, collated, or output.

- Registeration/Collation
  As triggered by the external read start signal EXRD, the background edit function saves programs from an external input unit into part program memory and verifies them.
- Output
  As triggered by the external punch start signal EXWT, the background edit function outputs all programs stored in the part program memory to an external output device.

Signal

External Read Start Signal EXRD <G058#1>

[Classification] Input signal

[Function] Programs are registered through the reader/puncher interface. Read programs are collated with programs already stored in the part program memory.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- In all modes other than the MDI mode, the background edit function reads programs from an external input device, and register them on the part program memory or collates them with programs already registered in the part program memory.
  (The memory protection key KEY3 <G046#5> determines whether to register or collate.)
- Bit 1 (RAL) of parameter No. 3201 selects whether to register all programs in a file or one program at a time. Bit 0 (RDL) of parameter No. 3201 can be used to delete all programs previously stored in the part program memory. However, it is impossible to delete programs protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.
- When programs are being registed or collated, the read/punch busy signal (RPBSY) is kept to be logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external read start signal EXRD is ignored.
- When programs are being registered or collated, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the registeration or collation is discontinued.
- If the foreground processing is already using the reader/puncher interface (for example, during DNC operation or program reading in the edit mode), the external read start signal EXRD is ignored.

- There are some other conditions to determine whether a program can be registered or collated. For example, a program cannot be registered or collated, if a program with the same program number is being executed in the foreground processing.

External Punch Start Signal EXWT <G058#3>

[Classification] Input signal

[Function] Programs stored in the part program memory are output to an external unit via the reader/puncher interface.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- In all modes other than the MDI mode, the background edit function outputs all programs stored in the part program memory to an external output device.
- When programs are being output, the read/punch busy signal RPBSY becomes logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external punch start signal EXWT is ignored.
- When programs are being output, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the output is discontinued.
- If the foreground processing is already using the reader/puncher interface (for example, during DNC operation or program reading in the edit mode), the external punch start signal EXWT is ignored.
- There are some other conditions to determine whether all programs can be output. For example, a program cannot be output, if it is running or protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.

External Read/Punch Stop Signal EXSTP <G058#2>

[Classification] Input signal

[Function] When the external read/punch stop signal becomes logical 1, it stops program registeration, collation, or output via the reader/puncher interface.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- The program registeration, collation or output triggered by the external read or punch start signal is stopped immediately.

Background editing signal BGEACT <F053#4>

[Classification] Output signal

[Function] This signal indicates that the background edit function is operating.

[Output condition] This signal becomes logical 1 when:

- The [BG EDIT] soft key is pressed to put the CNC in the background edit mode.
- The MDI mode is selected.
- The external read or punch start signal starts program registeration, collation, or output.
- Program upload or download is started by the DNC2 or MMC.

This signal becomes logical 0 when:

- The [BG END] soft key is pressed to terminate the background edit mode.
- The CNC shifts from the MDI mode to another mode.
- Program registeration or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).
- Program upload or download started by the DNC2 or MMC is ended.

Read/punch busy signal RPBSY <F053#2>

[Classification] Output signal

[Function] This signal indicates that program registeration, collation, or output triggered by the external read or punch start signal is under way.

[Output condition] This signal becomes logical 1, when:

- The external read or punch start signal triggers program registeration, collation, or output.

This signal becomes logical 0, when:

- Program registeration collation or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).

Read/punch alarm signal RPALM <F053#3>

[Classification] Output signal

[Function] This signal indicates that an alarm condition has occurred during program registeration, collation, or output triggered by the external read or punch start signal.

[Output condition] This signal becomes logical 1, when:

- An alarm condition occurs during program registeration, collation, or output triggered by the external read or punch start signal.

This signal becomes logical 0, when:

- The system is reset, or the external read/punch stop signal EXSTP is input.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058					EXWT	EXSTP	EXRD	

	#7	#6	#5	#4	#3	#2	#1	#0
F053				BGEACT	RPALM	RPBSY		

Parameter

Input/output channel number (parameter No. 0020)
↓

0020	I/O CHANNEL

Specify a channel for an input/output device.

I/ O CHANNEL
- =0 : Channel1
- =1 : Channel1
- =2 : Channel2

I/O CHANNEL	Parameter	Description
I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
	0102	Number specified for the input/output device
	0103	Baud rate
I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
	0112	Number specified for the input/output device
	0113	Baud rate
I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
	0122	Number specified for the input/output device
	0123	Baud rate

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99			REP	RAL	RDL

[Data type] Bit

RDL When a program is registered by input/output device external control

0 : The new program is registered following the programs already registered.

1 : All registered programs are deleted, then the new program is registered.
Note that programs which are protected from being edited are not deleted.

RAL When programs are registered through the reader/puncher interface

0 : All programs are registered.

1 : Only one program is registered.

REP Action in response to an attempt to register a program whose number is the same as that of an existing program

0 : An alarm is generated.

1 : The existing program is deleted, then the new program is registered.
Note that if the existing program is protected from being edited, it is not deleted, and an alarm is generated.

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:

0 : Completed

1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:

0 : Completed

1 : Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				NE8

[Data type] Bit

NE8 Editing of subprograms with program numbers 8000 to 8999

0 : Not inhibited

1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)

(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

NE9 Editing of subprograms with program numbers 9000 to 9999

0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)

(2) Program punching (Even when punching of all programs is specified, programs with program numbers 9000 to 9999 are not punched.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

Alarm and message

Number	Message	Description
079	PROGRAM VERIFY ERROR	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using reader /puncher interface, though the read terminate command is specified, input does not stop after 10 characters read. I/O unit or P.C.B. is defective.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.8.4	PROGRAM INPUT/OUTPUT
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.8.4	PROGRAM INPUT/OUTPUT
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.8.4	PROGRAM INPUT/OUTPUT
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.8.4	PROGRAM INPUT/OUTPUT

13.4 EXTERNAL PROGRAM INPUT

General

By using the external program input start signal, a program can be loaded from an input unit into CNC memory.
When an input unit such as the FANUC Handy File or FANUC Floppy Cassette is being used, a file can be searched for using the workpiece number search signals, after which the program can be loaded into CNC memory.

To use external program input start signal MINP <G058#0> to start data input, the following conditions must be satisfied:

- The reader/punch interface function is enabled.
- Bit 7 (MIP) of parameter No. 3201 is set to 1.
- Programs are not protected using the memory protection signal.
- Automatic operation mode is set.
- The current state is other than the start of automatic operation, that is, cycle start lamp signal STL is set to 0. (Automatic operation signal OP may be set to 1.)

Signal

External program input start signal MINP <G058#0>

[Classification] Input signal

[Function] This signal starts the loading of a program from an input unit into CNC memory.

[Operation] When the signal is set to 1, the control unit operates as follows:

- When memory operation mode is set, but no automatic operation is being performed and program loading is not inhibited by the setting of the memory protection key, the CNC deletes all currently loaded programs, then loads a program from the external input unit into CNC memory.
- When the FANUC Handy File or FANUC Floppy Cassette is being used as the input unit, a desired file can be searched for using the workpiece number search signals (PN1 to PN16), after which the program can be loaded into CNC memory.
  File numbers are indicated using the workpiece number search signals, as follows:

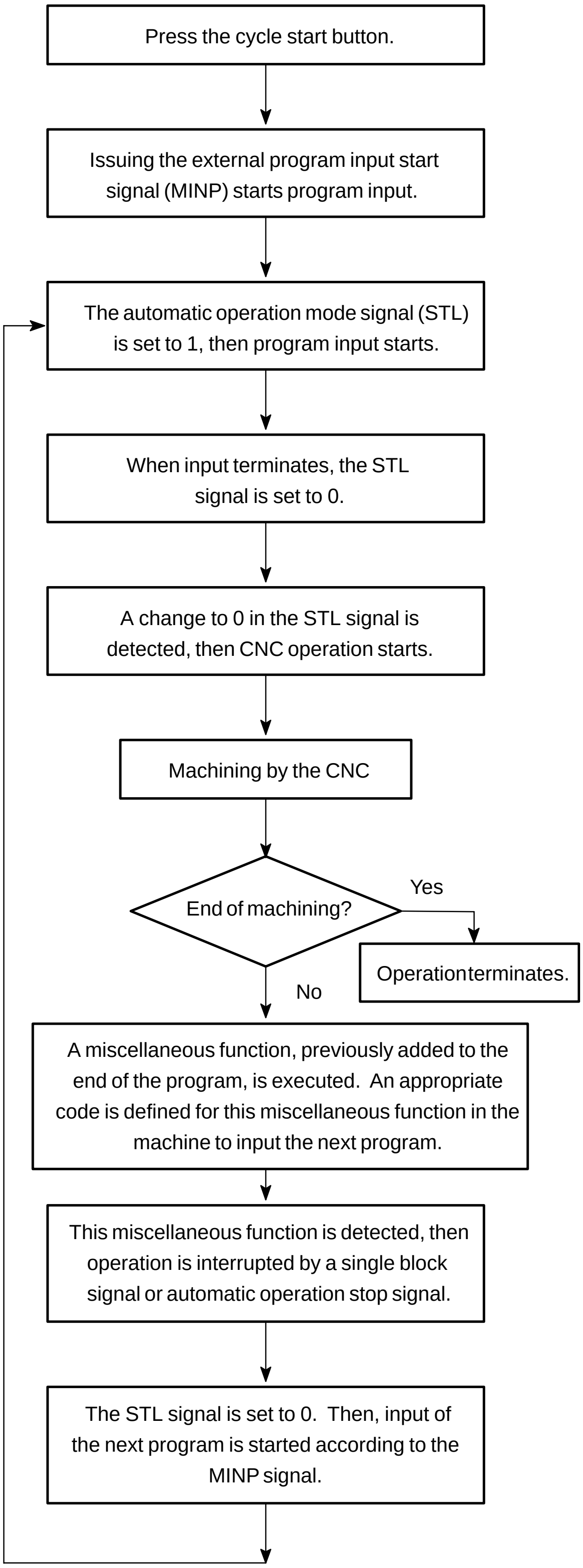
Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02

Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

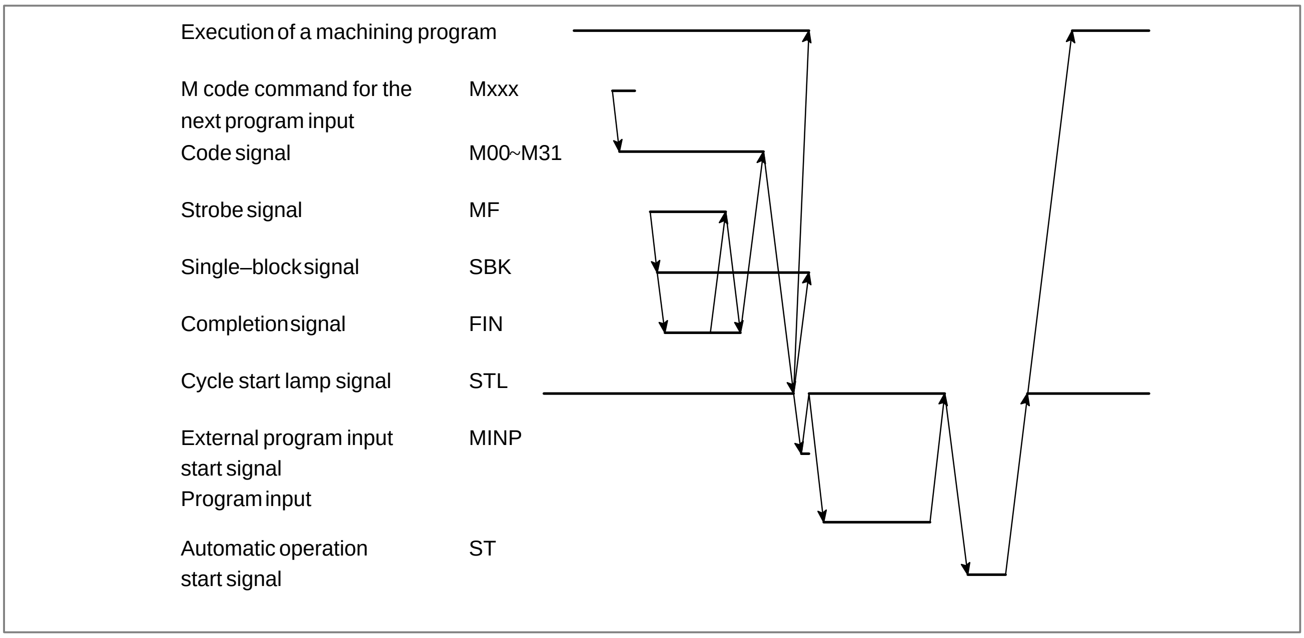
File No. 00 is used for special specification; specifying file No. 00 means that no search operation is to be performed. Therefore, numbers 01 to 31 can be assigned to files.

[Application] This function is applicable to the following case:
When a program to be used for machining is too large to be loaded into CNC memory, the program is divided into several segments. These segments are loaded into memory and executed, one by one.

In this case, the general operation flow is as shown below.



The timing chart for data reading is shown below.



CAUTION
The M code used for input of the next program must not be buffered.

NOTE
While a program is being input, the automatic operation mode signal STL is set to 1. Upon termination of program input, STL is set to 0.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058								MINP

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3201	MIP							

[Data type] Bit type

MIP Specifies whether to load a program into memory according to the external program input start signal (MINP).

0 : Does not load a program into memory.
1 : Loads a program into memory.

Note

NOTE

1 A program can be input according to the external program input start signal only when the program has only one program number.
To read programs having multiple program numbers, reset the CNC each time the CNC reads one program. After reset, search for a desired program by using the workpiece number search signals, then input the program according to the external program input start signal.

2 When program input is activated by the external program input activation signal, all programs are deleted from the CNC, at the beginning.

3 In background editing, the external program input activation signal is ineffective.

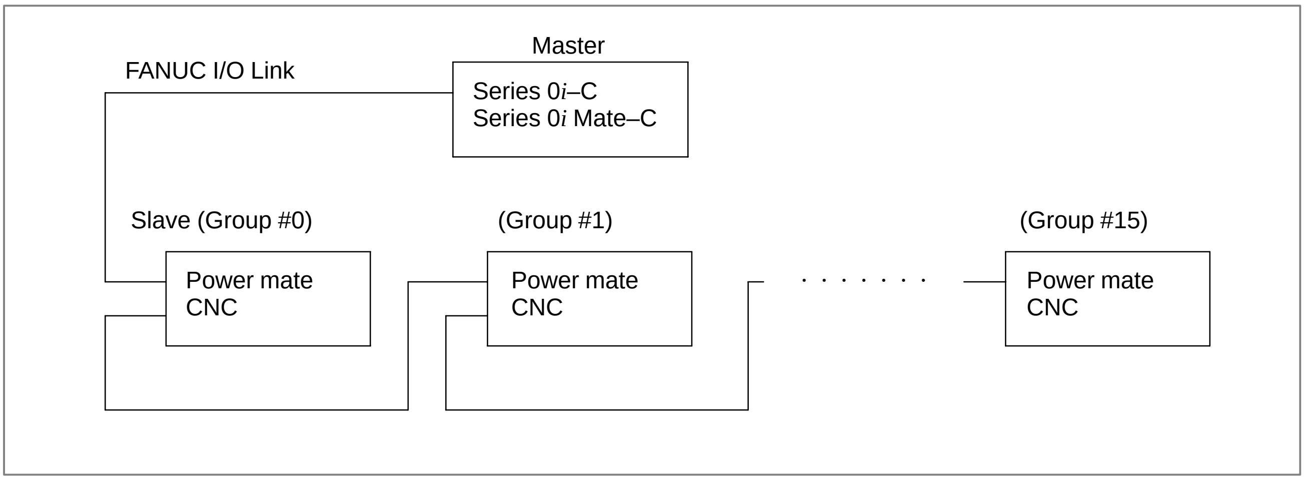
13.5 DATA INPUT/OUTPUT FUNCTIONS BASED ON THE I/O Link

General

Power mate CNC programs, parameters, macro variables, and diagnostic (PMC) data are input/output through the FANUC I/O Link.

With the FANUC I/O Link, slaves from group 0 to group 15 can be connected, thus enabling data input/output to and from a maximum of 16 power mate CNCs.

When data input/output function B based on the I/O Link is used, the external I/O device control function is associated with an I/O Link to enable the specification of an input/output group number and program number from the PMC. The external I/O device control function operates in the background. This means that when no other background operation is being performed, data can be input/output, regardless of the CNC mode and the screen selected.



The programs, parameters, macro variables, and diagnostic (PMC) data of a slave power mate CNC are stored in tape format within the part program storage length; these data items are stored as master program data in a master program memory area.

Data input/output can be performed between the master and the slave of a selected group. When the ordinary data input/output function based on the I/O Link is used, a group is selected by means of parameter setting. When data input/output function B based on the I/O Link is used, a group is selected by issuing the DI signal. Data input/output cannot be performed between the master and more than one group at a time.

For details of the Power Mate signals, refer to the “FANUC Power Mate Connection/Maintenance Manual.”

For details of the FANUC I/O Link, refer to the ”FANUC PMC Programming Manual.”
For details of the external I/O device control function, see Section 13.3.

- **Basic data input/output procedure**

(1) Program input/output

(a) Program input

· When the data input/output function based on the I/O Link is used

1) Specify a number between 20 to 35 as the I/O channel on the setting screen to specify a group number.
2) Specify EDIT mode.
3) Display the program screen with function key PROG .
4) To change the program numbers, enter a new program number as follows:
   (i) Select address O.
   (ii) Key in the new program number.
5) Using soft keys [(OPRT)], continuous–menu key ▷, [READ], and [EXEC], read the program.

· When data input/output function B based on the I/O Link is used

1) To read programs without changing the program numbers, specify –9999. To read programs by changing the program numbers, specify a desired number with the PMC function instruction (WINDW). A program number thus specified serves as the first program number when multiple programs are received. The second and any additional programs are numbered sequentially starting with the first number.
2) Specify a desired group number with group number specification signals SRLNI0 to SRLNI3.
3) Select an I/O Link by setting I/O Link specification signal IOLS to 1.
4) When external punch start signal EXRD is set to 1, programs are read from the power mate CNC.

> **NOTE**
> The user cannot read a program having an arbitrary program number registered in a power mate CNC. All programs are always read from a Power Mate.

(b) Program output

· When the data input/output function based on the I/O Link is used

1) Set a number between 20 and 35 as the I/O channel on the setting screen used to specify a group number.
2) Set EDIT mode.
3) Display the program screen by pressing the function key PROG .
4) Select address O.

5) Key in a program number.
6) Using soft keys [(OPRT)], continuous–menu key ▷, [PUNCH], and [EXEC], output the program corresponding to the keyed–in program number.

· When data input/output function B based on the I/O Link is used
  1) Using the PMC function instruction (WINDW), set the program number of the program to be output. Specify –9999 to output all programs.
  2) Specify a desired group number with group number specification signals SRLNI0 to SRLNI3.
  3) Select an I/O Link by setting the I/O Link specification signal IOLS to 1.
  4) When external punch start signal EXWT is set to 1, the program is output to the power mate CNC.

(2) Parameter input/output
A power mate CNC parameter in tape format is stored as a master–side NC program in a program memory area. The following program number is assigned to a power mate CNC parameter of group n:
(Value set in parameter No. 8760) + n × 10

(a) Parameter input

· When the data input/output function based on the I/O Link is used
In step 4) of program input (described in (1).(a)), key in a program number for a power mate CNC parameter. The other steps are the same as those for program input.

· When data input/output function B based on the I/O Link is used
In step 1) of program input (described in (1).(a)), specify a program number for the power mate CNC parameters. The remaining steps are the same as those for program input.

(b) Parameter output

· When the data input/output function based on the I/O Link is used
In step 5) of program output (described in (1).(b)), key in a program number for a power mate CNC parameter. The other steps are the same as those for program output.

· When data input/output function B based on the I/O Link is used
In step 1) of program output (described in (1).(b)), specify a program number for a power mate CNC parameter. The remaining steps are the same as those for program output.

(3) Macro variable input/output
In the same way as for parameters, a power mate CNC macro variable in tape format is stored as a master side NC program in a program memory area. The following program number is assigned to a power mate CNC macro variable of group n:
(Value set for parameter No. 8760) + n × 10 + 1

(a) Macro variable input

- When the data input/output function based on the I/O Link is used
  With power mate CNC DI signals EDG00 to EDG15, specify a start number for the macro variables to be read. With EDN00 to EDN15, specify the number of macro variables to be read.
  In step 4) of program input (described in (1).(a)), specify a program number for the Power Mate macro variables. The other steps are the same as those for program input.
- When data input/output function B based on the I/O Link is used
  With power mate CNC DI signals EDG00 to EDG15, specify a start number for the macro variables to be read. With EDN00 to EDN15, specify the number of macro variables to be read.
  In step 1) of program input (described in (1).(a)), specify a program number for power mate CNC macro variables. The remaining steps are the same as those for program input.

(b) Macro variable output

- When the data input/output function based on the I/O Link is used
  In step 5) of program output (described in (1).(b)), specify a program number for a power mate CNC macro variable. The other steps are the same as those for program output.
- When data input/output function B based on the I/O Link is used
  In step 1) of program output (described in (1).(b)), specify a program number for a power mate CNC macro variable. The remaining steps are the same as those for program output.

NOTE
The setting parameter or bit 1 (ISO) of parameter No. 0000 must be set to 1 to enable the use of the ISO code for macro variable output. When a macro variable is output in EIA code, the power mate CNC issues alarm 001, while the Series 16*i*/18*i*/21*i* issues alarm 86.

(4) Diagnostic (PMC) data input/output
In the same way as the parameters and macro variables, a power mate CNC diagnostic data item in tape format is stored as a master–side NC program in a program memory area. The following program number is assigned to a Power Mate diagnostic data item of group n:
<u>(Value set in parameter No. 8760) + n × 10 + 2</u>

(a) Diagnostic (PMC) data input

- When the data input/output function based on the I/O Link is used
  With power mate CNC DI signals EDG00 to EDG15, specify a start number for the diagnostic data items to be read. With EDN00 to EDN15, specify the number of diagnostic data items to be read.
  In step 4) of program input (described in (1).(a)), specify a program number for the power mate CNC diagnostic data items. The other steps are the same as those for program input.
- When data input/output function B based on the I/O Link is used
  With power mate CNC DI signals EDG00 to EDG15, specify a start number for the diagnostic data items to be read. With EDN00 to EDN15, set the number of diagnostic data items to be read.
  In step 1) of program input (described in (1).(a)), specify a program number for power mate CNC diagnostic data items. The remaining steps are the same as those for program input.

(b) Diagnostic (PMC) data output

- When the data input/output function based on the I/O Link is used
  In step 5) of program output (described in (1).(b)), specify a program number for a power mate CNC diagnostic data item. The other steps are the same as those for program output.
- When data input/output function B based on the I/O Link is used
  In step 1) of program output (described in (1).(b)), specify a program number for a power mate CNC diagnostic data item. The remaining steps are the same as those for program output.

NOTE
The addresses of power mate CNC DI signals EDG00 to EDG15 and EDN00 to EDN15 differ between Model–D and the other models, as indicated in the table below. For macro variable and diagnostic data input, ensure that these signals are processed correctly. If an invalid value or number is set, the power mate CNC's external I/O device control function is not started.

Signal name	Power Mate A/B/C/E	Power Mate D, F, H
EDG00 to EDG07 EDG08 to EDG15	G100.0 to G100.7 G101.0 to G101.7	G252.0 to G252.7 G253.0 to G253.7
EDN00 to EDN07 EDN08 to EDN15	G102.0 to G102.7 G103.0 to G103.7	G254.0 to G254.7 G255.0 to G255.7

- **Stopping input/output**

There are two methods of forcibly terminating input/output.

(1) Termination by a reset
Input/output can be terminated by a reset. In this case, however, slave read/write stop signal ESTPIO is not output. Therefore, the operation of the power mate CNC is not terminated even if the Power Mate is performing input/output. To terminate power mate CNC operation, create a ladder program so that ESTPIO is set to 1 upon the occurrence of a reset.

(2) Termination using external read/punch stop signal EXSTP (applicable only when data input/output function B based on the I/O Link is used)
Input/output can be terminated by setting external read/punch stop signal EXSTP to 1. In this case, slave read/write stop signal ESTPIO is output, and all processing is terminated once the termination of power mate CNC input/output has been confirmed. See the timing chart for the case where the Series 0*i*/0*i* Mate issues an alarm.

- **Specifying the PMC functions**

The PMC function instruction (WINDW) is used to set the program numbers used with data input/output function B based on the I/O Link.

[Data] This function is a window function for specifying the program numbers used to perform program input/output with the I/O device control function via the I/O Link.

[Input data structure]

Top address +0	+2	+4	+6	+8	+10
Function code	**Completion code**	**Data length**	**Data number**	**Data attribute**	**Data**
194	–	2	0	0	Program number

(‘–’ : Need not be set.)

Specify 0 in the data number and data attribute fields.
Specify 2 in the data length field.
Specify a desired two–byte program number (1 to 9999, –9999) in the data field.

[Output data structure]

Top address +0	+2	+4	+6	+8	+10
Function code	**Completion code**	**Data length**	**Data number**	**Data attribute**	**Data**
194	?	Input data	Input data	Input data	Input data

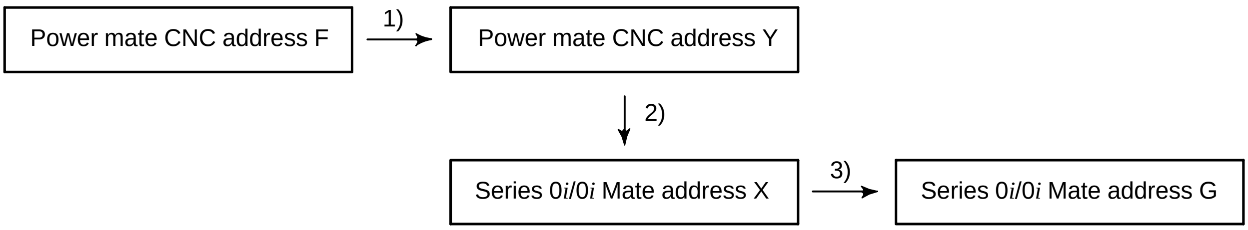
[Completion codes]

0 : Normal termination
5 : Data other than 1 to 9999 or –9999 was specified.

The data length, data number, and data attribute fields are not checked.
For details of the PMC function instructions, refer to the “FANUC PMC Programming Manual.”

- **Power Mate state signals (input)**

When the data input/output function based on the I/O Link is used, the state signals for a specified power mate CNC must be reported to the Series 0*i*/0*i* Mate. These signals must be posted to the Series 0*i*/0*i* Mate via the following path:

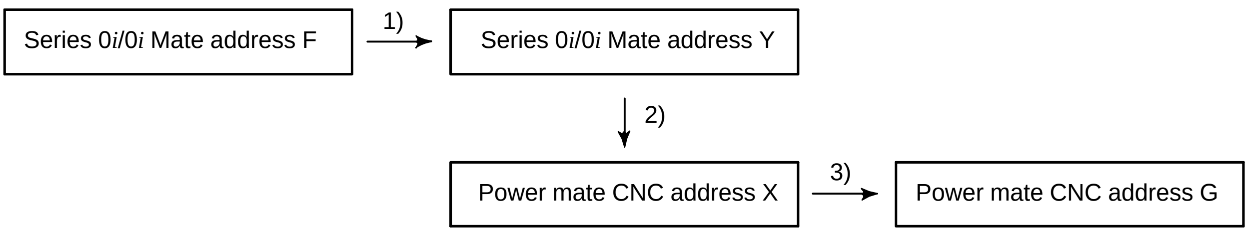


1) Depends on the ladder program of the power mate CNC.
2) Depends on the I/O Link.
3) Depends on the ladder program of the Series 0*i*/0*i* Mate.

For an explanation of the functions of the Power Mate state signals, see item Signal.

- **Power Mate control signals (output)**

When the data input/output function based on the I/O Link is used, power mate CNC control signals must be output from the Series 0*i*/0*i* Mate to control the external I/O device control function of a specified Power Mate. These signals must be posted to the power mate CNC via the following path:



1) Depends on the ladder program of the Series 0*i*/0*i* Mate.
2) Depends on the I/O Link.
3) Depends on the ladder program of the power mate CNC.

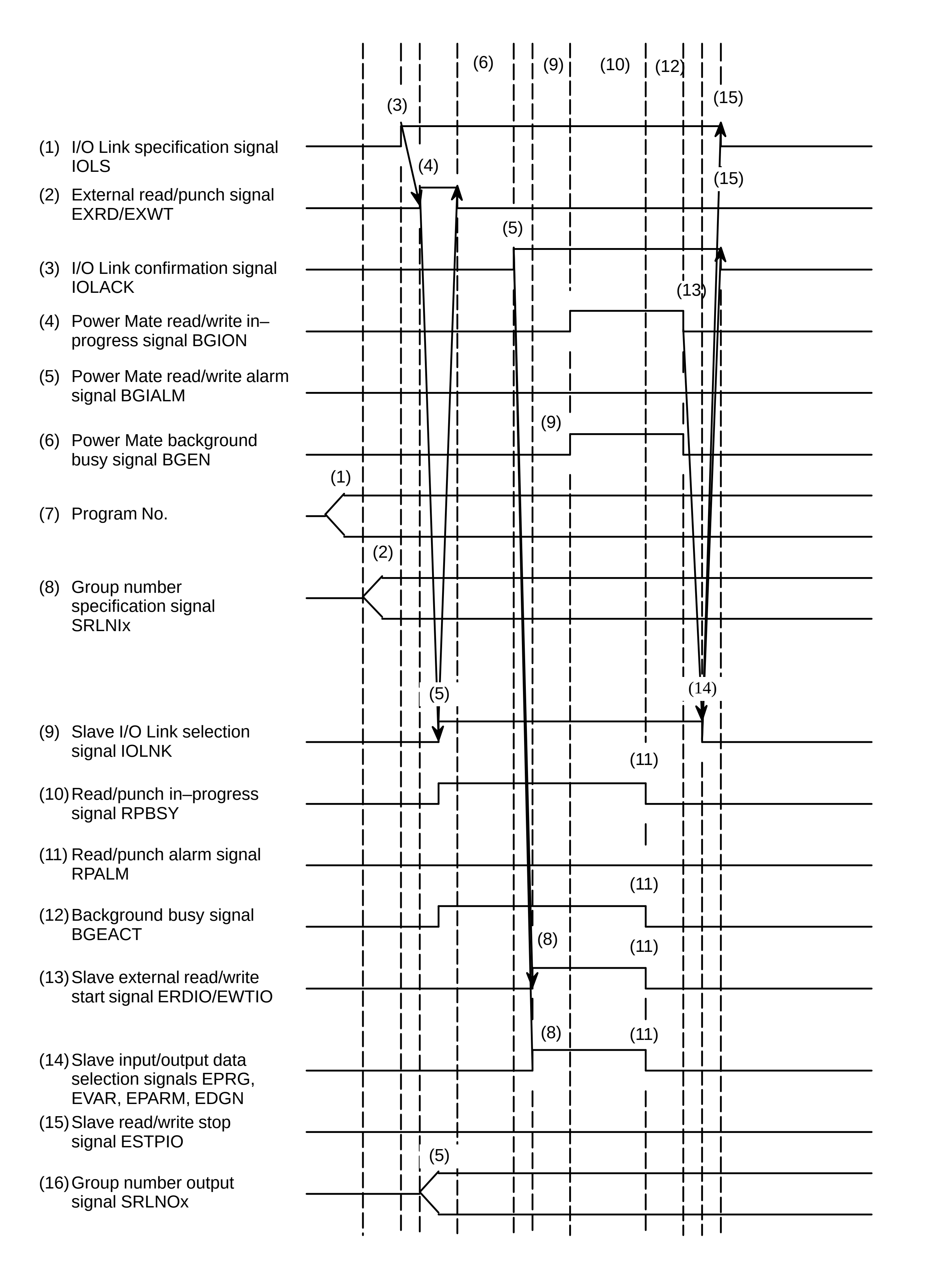
For an explanation of the functions of the Power Mate control signals, see item Signal.

- **DI/DO signal timing charts**

The DI/DO signal timing charts applicable when data input/output function B based on the I/O Link is used are shown below. When the ordinary data input/output function based on the I/O Link is used, 1) through 4) in the figures are subject to MDI–based input/output.

(1) When ordinary input/output is performed

1) Specify a program number with the PMC function instruction (WINDW).
2) Specify the group number of the power mate CNC to be used for input/output with group number specification signals SRLNI0 to SRLNI3.
3) Set I/O Link specification signal IOLS to 1.
4) For input, set external read start signal EXRD to 1.
For output, set external punch start signal EXWT to 1.
5) Processing by the Series 0*i*/0*i* Mate outputs slave I/O Link selection signal IOLNK and group number output signals SRLNO0 to SRLNO3.
6) Using a ladder program, make the necessary preparations for data input/output using the I/O Link.
7) Upon the completion of step 6), set the I/O Link confirmation signal IOLACK to 1. The Series 0*i*/0*i* Mate waits until IOLACK is set to 1.
8) The Series 0*i*/0*i* Mate outputs slave external read/write signal ERDIO/EWTIO, and slave input/output data selection signals EPRG, EVAR, EPARM, and EDGN.
9) The Series 0*i*/0*i* Mate waits for the external I/O device control function of the power mate CNC to start. (The Series 0*i*/0*i* Mate waits for the external I/O device control function of the power mate CNC to start.)
10) The Series 0*i*/0*i* Mate performs data input/output through the I/O Link.
11) Upon the completion of data input/output, the Series 0*i*/0*i* Mate sets the slave external read/write start signal and slave input/output data selection signals to 0.
12) The Series 0*i*/0*i* Mate waits for the completion of the external I/O device control function of the power mate CNC. The Series 0*i*/0*i* Mate waits for termination or completion of the external I/O device control function of the power mate CNC.
13) Upon the completion of the operation of the external I/O device control function of the power mate CNC, the Power Mate read/write in–progress signal BGION is set to 0.
14) The Series 0*i*/0*i* Mate sets the slave I/O Link selection signal IOLNK to 0.
15) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(1) I/O Link specification signal IOLS
(2) External read/punch signal EXRD/EXWT
(3) I/O Link confirmation signal IOLACK
(4) Power Mate read/write in–progress signal BGION
(5) Power Mate read/write alarm signal BGIALM
(6) Power Mate background busy signal BGEN
(7) Program No.
(8) Group number specification signal SRLNIx
(9) Slave I/O Link selection signal IOLNK
(10) Read/punch in–progress signal RPBSY
(11) Read/punch alarm signal RPALM
(12) Background busy signal BGEACT
(13) Slave external read/write start signal ERDIO/EWTIO
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(15) Slave read/write stop signal ESTPIO
(16) Group number output signal SRLNOx
(1)
(2)
(3)
(4)
(5)
(6)
(8)
(9)
(10)
(11)
(12)
(13)
(14)
(15)

(2) When an alarm is issued by the Series 0*i*/0*i* Mate (including the case where processing is stopped by external read/punch signal EXSTP)

Steps 1) to 10) are the same as those for ordinary input/output.

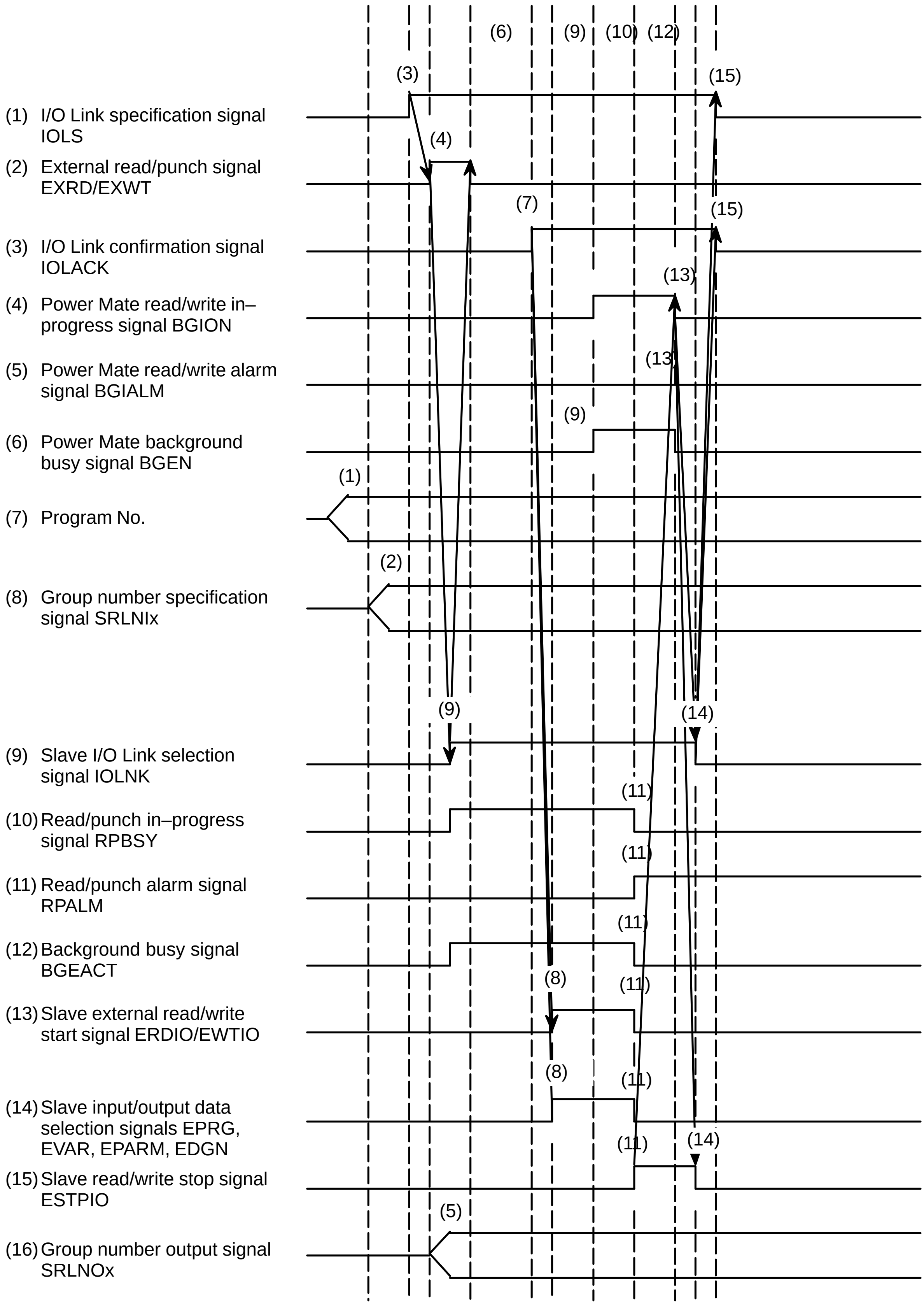
11) When the Series 0*i*/0*i* Mate issues an alarm, or when external read/punch stop signal EXSTP is set to 1, slave read/write stop signal ESTPIO is set to 1.

12) The Series 0*i*/0*i* Mate waits for the external I/O device control function of the power mate CNC to terminate. The Series 0*i*/0*i* Mate merely waits until the external I/O device control function of the power mate CNC terminates.

13) When the external I/O device control function of the power mate CNC terminates, Power Mate read/write in–progress signal BGION is set to 0.

14) The Series 0*i*/0*i* Mate sets slave I/O Link selection signal IOLNK and slave read/write stop signal ESTPIO to 0.

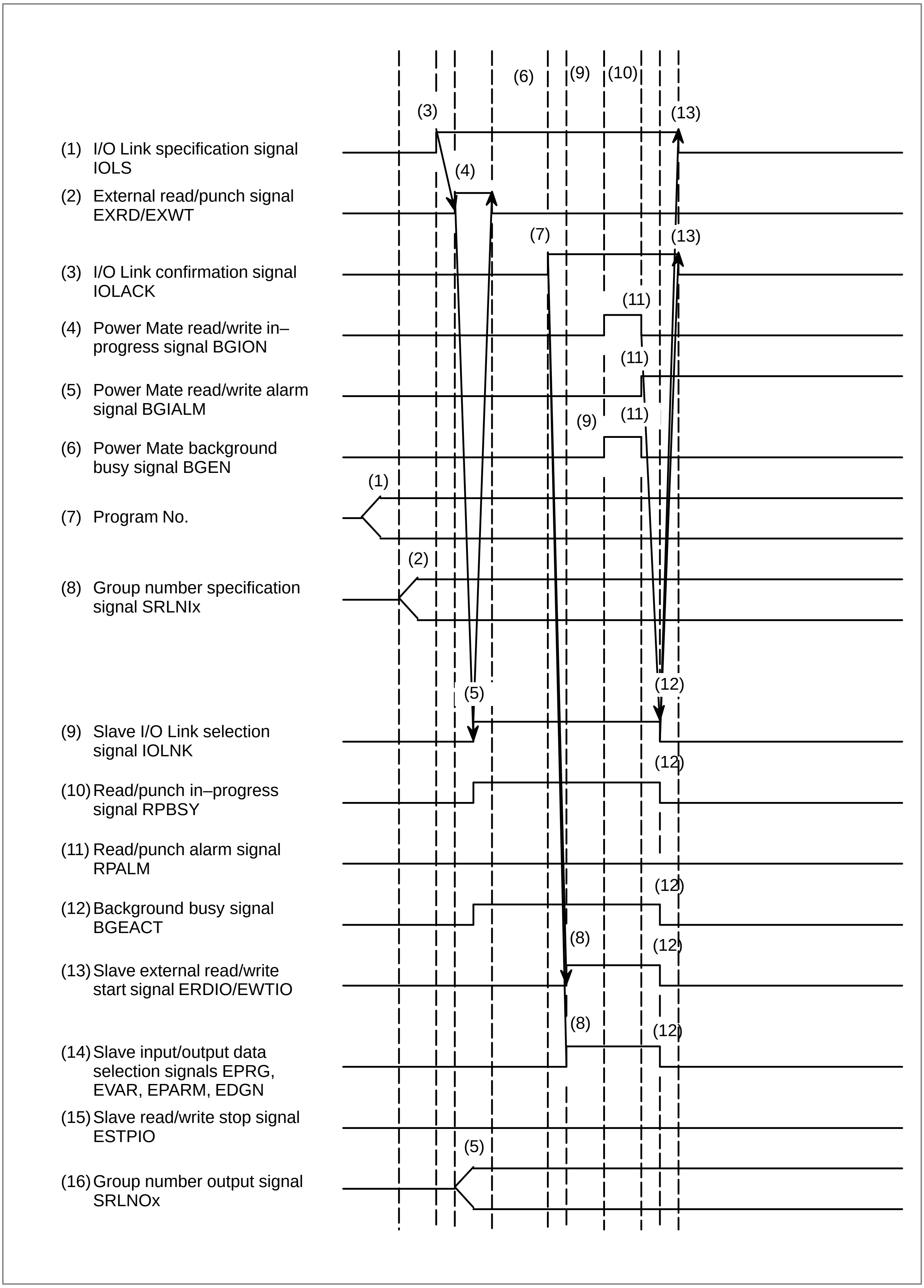
15) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(6)
(9)
(10)
(12)
(3)
(15)
(1) I/O Link specification signal IOLS
(4)
(2) External read/punch signal EXRD/EXWT
(7)
(15)
(3) I/O Link confirmation signal IOLACK
(13)
(4) Power Mate read/write in–progress signal BGION
(13)
(5) Power Mate read/write alarm signal BGIALM
(9)
(6) Power Mate background busy signal BGEN
(1)
(7) Program No.
(2)
(8) Group number specification signal SRLNIx
(9)
(14)
(9) Slave I/O Link selection signal IOLNK
(11)
(10) Read/punch in–progress signal RPBSY
(11)
(11) Read/punch alarm signal RPALM
(11)
(12) Background busy signal BGEACT
(8)
(11)
(13) Slave external read/write start signal ERDIO/EWTIO
(8)
(11)
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(11)
(14)
(15) Slave read/write stop signal ESTPIO
(5)
(16) Group number output signal SRLNOx

(3) When an alarm is issued by the power mate CNC

Steps 1) to 10) are the same as those for ordinary input/output.

11) When the power mate CNC issues an alarm, Power Mate read/write alarm signal BGIALM is set to 1, and Power Mate read/write in–progress signal BGION is set to 0.
12) The Series 0*i*/0*i* Mate sets slave I/O Link selection signal IOLNK and slave read/write stop signal ESTPIO to 0.
13) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(1) I/O Link specification signal IOLS
(2) External read/punch signal EXRD/EXWT
(3) I/O Link confirmation signal IOLACK
(4) Power Mate read/write in–progress signal BGION
(5) Power Mate read/write alarm signal BGIALM
(6) Power Mate background busy signal BGEN
(7) Program No.
(8) Group number specification signal SRLNIx
(9) Slave I/O Link selection signal IOLNK
(10) Read/punch in–progress signal RPBSY
(11) Read/punch alarm signal RPALM
(12) Background busy signal BGEACT
(13) Slave external read/write start signal ERDIO/EWTIO
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(15) Slave read/write stop signal ESTPIO
(16) Group number output signal SRLNOx

- **Troubleshooting**

The data input/output function based on the FANUC I/O Link is implemented by various elements such as ladder programs, I/O Link assignment, Series 0*i*/0*i* Mate parameters, and power mate CNC parameters. So, problems may occur when the function is started.

The table below lists the symptoms of the problems that may occur, together with their causes and corresponding corrective actions.

Symptom	Cause and corrective action
Data input/output is started, but the Series 0*i*/0*i* Mate and power mate CNC return no response.	The background is already active (the background busy signal BGEACT is set to 1). In this case, the function cannot be used. Examples: 1) In MDI mode, the program screen is selected. 2) Background editing has already started.
	Communication via the RS–232–C interface is in progress. Multiple communication operations cannot be performed at the same time, such that any subsequent attempt to start the function is ignored.
	The selected slave is not a power mate CNC. Data input/output function B based on the I/O Link is not started if the slave specified with a group number set with group number specification signals SRLNI0 to SRLNI3 is other than a power mate CNC.
	A ladder program error may prevent the function from being started. Check the ladder program for errors such as mis–specified addresses.
When an attempt is made to output data to a power mate CNC: Series 0*i*/0*i* Mate: OUTPUT blinks continuously. Power mate CNC : LSK blinks continuously . (Caution)	Power mate CNC DO signal RPBSY is not correctly passed to Power Mate read/write in–progress signal RPBSY of the Series 0*i*/0*i* Mate.

Symptom	Cause and corrective action
When an attempt is made to output data to a power mate CNC: Series 0*i*/0*i* Mate: OUTPUT blinks continuously. Power mate CNC : No response is returned. (Caution)	I/O Link confirmation signal IOLACK is not set to 1. A ladder program error, I/O Link assignment error may prevent.
When an attempt is made to read data from a power mate CNC: Series 0*i*/0*i* Mate: LSK blinks continuously. Power mate CNC: No response is returned. (Caution)	
When an attempt is made to output parameters to a power mate CNC: Series 0*i*/0*i* Mate: OUTPUT blinks continuously. Power mate CNC: No response is returned. (Caution)	PWE of the power mate CNC is set to 0. To modify the power mate CNC parameters, PWE must be set to 1. (Caution)
When an attempt is made to output diagnostic data to a power mate CNC: Series 0*i*/0*i* Mate: OUTPUT blinks continuously. Power mate CNC : No response is returned. (Caution)	DWE of the power mate CNC is set to 0. To enable the modification of power mate CNC diagnostic data, DWE must be set to 1. (Caution)

Symptom	Cause and corrective action
When an attempt is made to output macro variables to a power mate CNC: Series 0*i*/0*i* Mate: Alarm 86 is issued. Power mate CNC : Alarm 1 is issued.	Bit 1 (ISO) of parameter No. 0000 is set to 1 (EIA code). The EIA code does not include #, such that an ISO code must be set.
When an attempt is made to read macro variables or diagnostic data from a power mate CNC: Series 0*i*/0*i* Mate: LSK blinks continuously. Power mate CNC : No response is returned. (Caution)	A start number and the number of data items to be transferred are not set correctly in power mate CNC DI signals G100 to G103. Examples: 1 A ladder program error or I/O allocation error prevented a start number and the number of data items from being transferred to power mate CNC DI signals G100 to G103. 2 An invalid start number is specified, or the specified number of data items to be transferred is invalid.

CAUTION

If these symptoms are detected, the Series 0*i*/0*i* Mate waits for a condition to be satisfied in its internal processing. While such a state exists, the screen is not updated. So, the states of signals cannot be checked on a real–time basis on a screen such as the PMC diagnostic screen.

Signal

Power Mate read/write in–progress signal BGION <G092#2>

[Classification] Input signal

[Function] This signal indicates that the Power Mate is performing data input/output.

[Operation] This signal is set to 1 when the power mate CNC is performing data input/output.
This signal is a Power Mate state signal. The corresponding power mate CNC side signal is RPBSY <F223#2>.

Power Mate read/write alarm signal BGIALM <G092#3>

[Classification] Input signal

[Function] This signal indicates that an alarm has been issued while the power mate CNC was performing data input/output.

[Operation] This signal is set to 1 upon the issue of an alarm while the power mate CNC is performing data input/output.
This signal is a Power Mate state signal. The corresponding power mate CNC side signal is RPALM <F223#3>.

Power Mate background busy signal BGEN <G092#4>

[Classification] Input signal

[Function] This signal indicates that the power mate CNC is performing background editing.

[Operation] This signal is set to 1 when the power mate CNC is performing background editing.
This signal is a Power Mate state signal. The corresponding power mate CNC side signal is BGEACT <F223#4>.

I/O Link confirmation signal IOLACK <G092#0>

[Classification] Input signal

[Function] This signal indicates that the Power Mate state signals are valid.

[Operation] When this signal is set to 1, the control unit operates as follows:

– All Power Mate state signals become valid.

For data input/output between the Series 0*i*/0*i* Mate and power mate CNC, the Power Mate state signals are used. For this purpose, ladder program processing is required. Upon the completion of ladder program processing, I/O Link confirmation signal IOLACK is set to 1 to make the Power Mate state signals active.

I/O Link specification signal IOLS <G092#1>

[Classification] Input signal

[Function] This signal specifies whether those signals that are shared by the external I/O device control function are to be used with data input/output function B based on the I/O Link.

[Operation] When this signal is set to 1, the control unit operates as follows:

– The signals (EXRD, EXSTP, EXWT, RPBSY, RPALM, and BGEACT) for the external I/O device control function are used with data input/output function B based on the I/O Link.

NOTE
I/O Link specification signal IOLS is not used with the ordinary data input/output function based on the I/O Link.

Group number specification signals SRLNI0 to SRLNI3 <G091#0 to #3>

[Classification] Input signal

[Function] These signals specify the group number of the power mate CNC that acts as a slave when data input/output function B based on the I/O Link is used.

[Operation] The group number of the power mate CNC that acts as a slave is specified with the values of four digits binary code signals.

NOTE
Group number specification signals SRLNI0 to SRLNI3 are not used with the ordinary data input/output function based on the I/O Link.

Slave I/O Link selection signal IOLNK <F177#0>

[Classification] Output signal

[Function] This signal instructs the power mate CNC to perform data input/output based on the I/O Link.

[Output condition] This signal is set to 1 in the following case:

– When data input/output is performed

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a power mate CNC control signal. The corresponding Power Mate side signal is IOLNK <G099#7/G251#0> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave external read start signal ERDIO <F177#1>

[Classification] Output signal

[Function] This signal indicates that the Series 0*i*/0*i* Mate has started data output.

[Output condition] This signal is set to 1 in the following case:

– When data output is started

This signal is set to 0 in the following case:

– When data output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EXRD <G098#1/G058#1> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave read/write stop signal ESTPIO <F177#2>

[Classification] Output signal

[Function] This signal forcibly terminates power mate CNC data input/output.

[Output condition] This signal is set to 1 in the following cases:

– When the Series 0*i*/0*i* Mate issues an alarm

– When data input/output function B based on the I/O Link is used, and external read/punch stop signal EXSTP is set to 1

This signal is set to 0 in the following case:

–When data input/output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EXSTP <G098#2/G058#2> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave external write start signal EWTIO <F177#3>

[Classification] Output signal

[Function] This signal indicates that the Series 0*i*/0*i* Mate has started data input.

[Output condition] This signal is set to 1 in the following case:

– When data input is started

This signal is set to 0 in the following case:

– When data input is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EXWT <G098#3/G058#3> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave program selection signal EPRG <F177#4>

[Classification] Output signal

[Function] This signal notifies the power mate CNC that the input/output data consists of programs.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of programs, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EPRG <G098#4/G251#4> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave macro variable selection signal EVAR <F177#5>

[Classification] Output signal

[Function] This signal notifies the power mate CNC that the input/output data consists of macro variables.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of macro variables, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EVAR <G098#5/G251#5> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave parameter selection signal EPARM <F177#6>

[Classification] Output signal

[Function] This signal notifies the power mate CNC that the input/output data consists of parameters.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of parameters, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EPARM <G098#6/G251#6> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Slave diagnostic data selection signal EDGN <F177#7>

[Classification] Output signal

[Function] This signal notifies the power mate CNC that input/output data consists of diagnostic (PMC) data.

[Output condition] This signal is set to 1 in the following case:

– When input/output data consists of diagnostic (PMC) data, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding power mate CNC side signal is EDGN <G098#7/G251#7> (the second address being for Power Mate–MODEL D, and the first address for all other models).

Group number output signals SRLNO0 to SRLNO3 <F178#0 to #3>

[Classification] Output signal

[Function] These signals indicate the group number of the power mate CNC that is acting as a slave.

[Operation] The group number of the power mate CNC that is acting as a slave is specified with the values of four binary code signals.
These signals become active when the slave I/O Link selection signal IOLNK is set to 1.

The signals listed below are used with data input/output function B based on the I/O Link. For details, see Section 13.3.

External read start signal EXRD <G058#1>

External punch start signal EXWT <G058#3>

External read/punch stop signal EXSTP <G058#2>

Background busy signal BGEACT <F053#4>

Read/punch in–progress signal RPBSY <F053#2>

Read/punch alarm signal RPALM <F053#3>

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G058					EXWT	EXSTP	EXRD	
G091					SRLNI3	SRLNI2	SRLNI1	SRLNI0
G092				BGEN	BGIALM	BGION	IOLS	IOLACK
F053				BGEACT	RPALM	RPBSY		
F177	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLINK
F178					SRLNO3	SRLNO2	SRLNO1	SRLNO0

Parameter

0020	I/O CHANNEL: Selection of an input/output device or selection of input device in the foreground

Setting entry is acceptable.

[Data type] Byte

[Valid data range] 0 to 35

I/O CHANNEL: Selection of the input/output device to be used

The CNC provides the following interfaces for data transfer to and from the host computer and external input/output devices:

- Input/output device interface (RS–232–C serial port 1, 2)
- DNC2 interface

Data input/output can be performed with a personal computer connected via FOCAS1/Ethernet or FOCAS1/HSSB. Data input/output can be performed with the Power Mate CNC via the FANUC I/O Link.

This parameter selects the interface used to transfer data to and from an input/output device.

Setting	Description
0, 1	RS–232–C serial port 1
2	RS–232–C serial port 2
4	Memory card interface
5	Data server interface
6	The DNC operation is performed or M198 is specified by FOCAS1/ Ethernet.
10	DNC2 interface, OSI–Ethernet
15	M198 is specified by FOCAS1/HSSB. (Bit 1 (NWD) of parameter No. 8706) must also be specified.)
20 21 22 to 34 35	Group 0 Group 1 Group 2 to Group 14 Group 15 Data is transferred between the CNC and a Power Mate CNC in group n (n: 0 to 15) via the FANUC I/O Link.

Supplemental remark 1

If the DNC operation is performed with FOCAS1/HSSB, the setting of parameter No. 20 does not matter. The DMMC signal <G042.7> is used.

Supplemental remark 2

If bit 0 (IO4) of parameter No. 110 is set to control the I/O channels separately, the I/O channels can be divided into four types: input and output in the foreground and input and output in the background. If so, parameter No. 20 becomes a parameter for selecting the input device in the foreground.

8760	Program number for data registration (data input/output function using the I/O link)

[Data type] Word

[Valid data range] 0 to 9999

When the data input/output function using the I/O link is used, this parameter sets the program numbers of the programs to be used for registering data (parameters, macro variables, and diagnostic data) from power mate CNCs.

For a power mate CNC in group n, the following program numbers are used:
For parameters: Setting + n $\times$ 10 + 0
For macro variables: Setting + n $\times$ 10 + 1
For diagnostic data: Setting + n $\times$ 10 + 2

Example: When 8000 is set

8000: Parameters of group 0 (I/O channel = 20)
8001: Macro variables of group 0 (I/O channel = 20)
8002: Diagnostic data of group 0 (I/O channel = 20)
8010: Parameters of group 1 (I/O channel = 21)
8011: Macro variables of group 1 (I/O channel = 21)
8012: Diagnostic data of group 1 (I/O channel = 21)
8020: Parameters of group 2 (I/O channel = 22)
8021: Macro variables of group 2 (I/O channel = 22)
8022: Diagnostic data of group 2 (I/O channel = 22)

8150: Parameters of group 15 (I/O channel = 35)
8151: Macro variables of group 15 (I/O channel = 35)
8152: Diagnostic data of group 15 (I/O channel = 35)

NOTE

When 0 is set, the input/output of parameters, macro variables, and diagnostic data cannot be performed, but program input/output processing is performed.

Alarm and message

Number	Message	Description
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, or FANUC I/O Link an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective. Or, the slave is not ready to perform data input/output (or the slave is performing background editing) if this alarm is issued when data input/output using the FANUC I/O Link is started. Alternatively, slave–side input/output is stopped (with alarm output) if this alarm is issued during input/output.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective. Alternatively, for data read using the FANUC I/O Link, the master directs the termination of a read operation, but the slave does not stop output.

13.6 SCREEN HARD COPY FUNCTION

General

When the display control card has a graphic function, screen information displayed on the CNC can be converted to 640–by–480–dot bit–mapped data and output to a memory card. Then, the created bit map data can be displayed on a personal computer.

Operation

The screen hard copy function is activated by setting the hard copy start signal HDREQ (G67#7) to 1 or by holding down the [SHIFT] key for five seconds. The function is stopped by pressing the [CAN] key or by setting the hard copy stop signal HDABT (G67#6) to 1.
During screen hard copy operation, the hard copy in–progress signal (F061#3) is held 1, and the screen is kept in the static state for several tens of seconds (several seconds for a monochrome LCD) until hard copy operation terminates. Upon completion of screen hard copy operation, the hard copy in–progress signal (F061#3) is set to 0.

The number of colors used in created bit–mapped data depends on the display control card, LCD hardware, and display mode of the CNC display screen. The relationship is shown in Table 13.6 (a).

Table 13.6 (a) Number of colors in the BMP data created by screen hard copy operation

LCD hardware	CNC screen display mode	Number of colors displayed in CNC	Number of colors in created BMP data	Remarks
Monochrome LCD	–	2 colors	2 colors	The monochrome tone is not supported.
Color LCD	VGA compatible mode (Parameter No. 3119#7=1)	Character: 16 colors Graphics: 16 colors	When bit 0 of parameter No. 3301 is 0: 256 colors When bit 0 of parameter No. 3301 is 1: 16 colors	Note that in 16–color mode, colors cannot sometimes be displayed normally.
	VGA mode (Parameter No. 3119#7=0)	256 colors	256 colors	

The following names are assigned sequentially to files of bit–mapped data created by the first and subsequent screen hard copy operations since power–up:

’HDCPY000.BMP’ (Data name assigned by the first hard copy operation since power–up)

’HDCPY001.BMP’ (Data name assigned by the second hard copy operation since power–up)

:

:

’HDCPY099.BMP’

After HDCPY099.BMP is output, executing another screen hard copy operation outputs HDCPY000.BMP. Note that, however, when a file having the same name as that of the BMP data to be output by screen hard copy operation is already present on the memory card, the file is overwritten unconditionally.

With the screen hard copy function, a hard copy of a static image screen displayed on the CNC can be created. Hard copy operation is possible in all operation modes.
Hard copy operation for those screens that meet the restriction conditions described on page XX, however, cannot be performed.
Table 13.6 (b) shows the sizes of bit–mapped data created by screen hard copy operation.

Table 13.6 (b) Size of bit–mapped data created by screen hard copy

Number of colors in bit map	File size [in bytes]
Monochrome (2 colors)	38,462
Color (16 colors)	153,718
Color (256 colors)	308,278

CAUTION
1 During screen hard copy operation, key input is disabled for several tens of seconds, and the screen is held in the static state until the hard copy operation terminates. During this period, except the hard copy in–progress signal (F061#3) that is set to 1, nothing is output. So, do not turn off the power, carelessly regarding this status as faulty.
2 Unless the screen is in the static state, it is difficult to take a hard copy normally. So, when taking a screen hard copy, be sure to place the screen in the static state.
3 When hard copy operation is performed after the power is turned off then on again, file name HDCPY000.BMP is assigned. In this case, if the same file name is present on the memory card, the existing data is overwritten unconditionally, so care should be taken.

Restriction

- This function cannot be used in the following case:
  – Construction of connecting with PC via HSSB and not attaching MDI to the CNC side
- Hard copies of system alarm screens cannot be taken.
- When RS–232C is being used, hard copy operation cannot be performed.
- During automatic operation (or manual operation), hard copy operation cannot be performed. Hard copy operation, however, can be performed in the stopped state.

Alarm and message

When bit 2 (HCALM) of parameter No. 3301 is set to 1, the CNC can issue an alarm message if hard copy execution fails. Table 13.6 (c) lists the alarm messages, their meanings, and responses.

Table 13.6 (c) Alarm messages related to the screen hard copy function

Number	Message	Description
5212	SCREEN COPY: PARAMETER ERROR	An obviously illegal parameter setting is found. Check that the I/O channel is set to 4.
5213	SCREEN COPY: COMMUNICATION ERROR	The right to exclusively use the memory card cannot be obtained. The memory card is abnormal or write–protected.
5214	SCREEN COPY: DATA TRANSFER ERROR	Data transfer to the memory card failed. The memory card may have been removed during hard copy operation. Alternatively, the available area on the memory card becomes 0.

Signal

Hard copy request signal HCREQ <G067#7>

[Classification] Input signal

[Function] This signal requests the CNC to execute hard copy operation.

[Operation] When this signal is set to 1, the CNC operates as follows:

- Starts hard copy operation.

Hard copy stop request signal HCABT <G067#6>

[Classification] Input signal

[Function] This signal requests the CNC to stop hard copy operation.

[Operation] When this signal is set to 1, the CNC operates as follows:

- Stops hard copy operation.

Hard copy in–progress signal HCEXE <F061#3>

[Classification] Output signal

[Function] This signal reports that hard copy operation is in progress.

[Output condition] This signal is set to 1 when:

- The hard copy execution state has been entered by setting G067#7 (HDCPY) to 1, by holding down the [SHIFT] key for 4.8 seconds, and so forth.

The signal is set to 0 when:

- The hard copy execution state is released because hard copy operation is completed or canceled.

Hard copy stop request acceptance flag HCAB2 <F061#2>

[Classification] Output signal

[Function] This signal reports that a request to stop hard copy operation is made.

[Output condition] This signal is set to 1 when:

- G063#6 (HCABT) is set to 1, or a hard copy stop command is issued by, for example, pressing the [CAN] key.

The signal is set to 0 when:

- The reset key is pressed.
- Another hard copy operation starts.

Time charts of the input and output signals are shown below. Fig. 13.6 (a) is a time chart of normal termination of hard copy operation, and Fig. 13.6 (b) is a time chart of forced termination of hard copy operation.

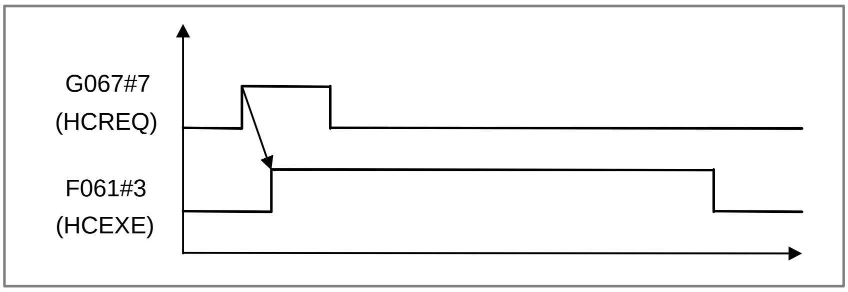


Fig. 13.6 (a) Time chart when screen hard copy terminates normally

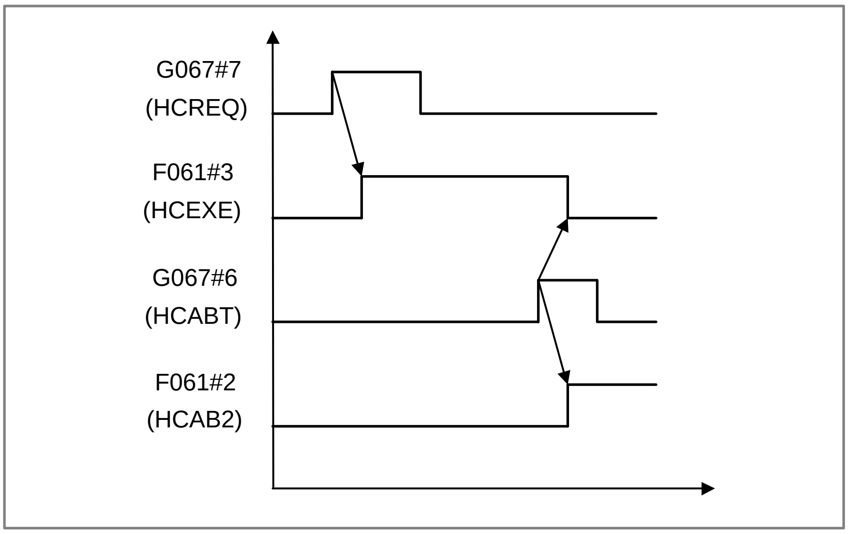


Fig. 13.6 (b) Time chart when screen hard copy is interrupted

CAUTION
1 Even when the hard copy stop request acceptance signal (HCAB2, F063#2) is 1, another hard copy operation can start. In this case, HCAB2 is set to 0 automatically.
2 Even when the hard copy in–progress signal (F061#3) is set from 1 to 0 (hard copy operation is terminated for a cause), the next hard copy operation does not start unless G067#7 (HDCPY) is set to 0 once.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G67	HCREQ	HCABT						

	#7	#6	#5	#4	#3	#2	#1	#0
F61					HCEXE	HCAB2		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3301	HDCPY				HCNEG	HCALM		HDCL1

[Data type] Bit

HDCL1 When the screen display is in VGA compatible mode on a color LCD:

0 : Hard copy is performed with 256–color BMP. (The same colors as those on the screen display can always be obtained, but data is transferred to the memory card slowly.)

1 : Hard copy is performed with 16–color BMP. (Data is transferred to the memory card at high speed, but the colors in the data may differ from those on the screen display.)

HCALM When an alarm is issued during hard copy operation:

0 : No alarm message is displayed.
1 : An alarm message is displayed.

HCNEG When a hard copy is taken with the monochrome VGA or character card:

0 : Characters (graphics) are in black, and the background is in while in the hard copy.
1 : Characters (graphics) are in white, and the background is in black in the hard copy.

HDCPY Screen hard copy operation is:

0 : Disabled.
1 : Enabled.

> **CAUTION**
> In addition to the above parameters, it is necessary to set 4 in parameter No. 20 (I/O channel selection).

Diagnostic screen

The hard copy status is output. When the hard copy start signal (HDCPY, G67#7) is not set to 1, pressing the reset button resets all bits of diagnostic number 35 to 0.

	#7	#6	#5	#4	#3	#2	#1	#0
0035				HCER3	HCER2	HCER1	HCAB3	HCEND

HCEND Normal termination of hard copy operation

HCAB3 Acceptance of hard copy interruption request

HCER1 Hard copy parameter setting error (= P/S5212)

HCER2 Failure to obtain the right to exclusively use the memory card (= P/S5213)

HCER3 Failure to write data to the memory card (= P/S5214)

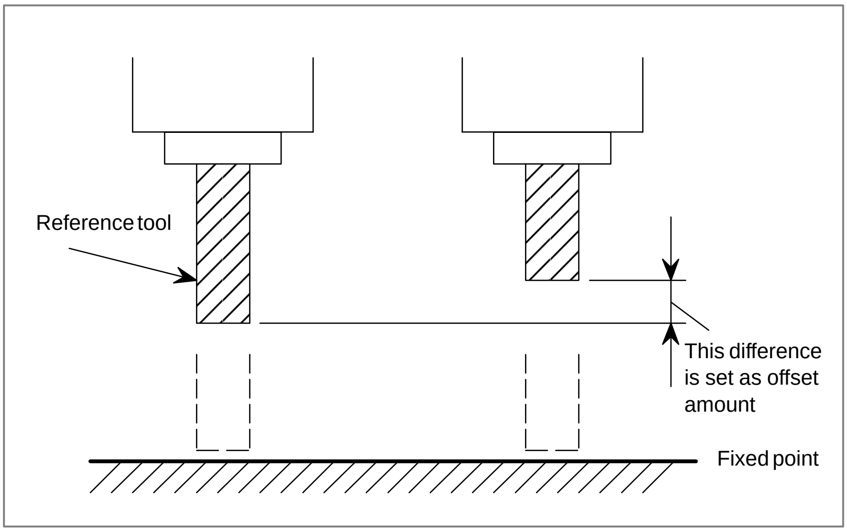
14 MEASUREMENT

14.1 TOOL LENGTH MEASUREMENT (M SERIES)

General

The value displayed as a relative position can be set in the offset memory as an offset value by a soft key.

Switch to the offset value display screen on the CRT. Relative positions are also displayed on this screen. Then select the reference tool and set it at the fixed point on the machine by manual operation. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by manual operation. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts.



Reference Item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.2	Tool Length Measurement
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.2	Tool Length Measurement

14.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES)/ AUTOMATIC TOOL OFFSET (T SERIES)

General

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset.

Signal

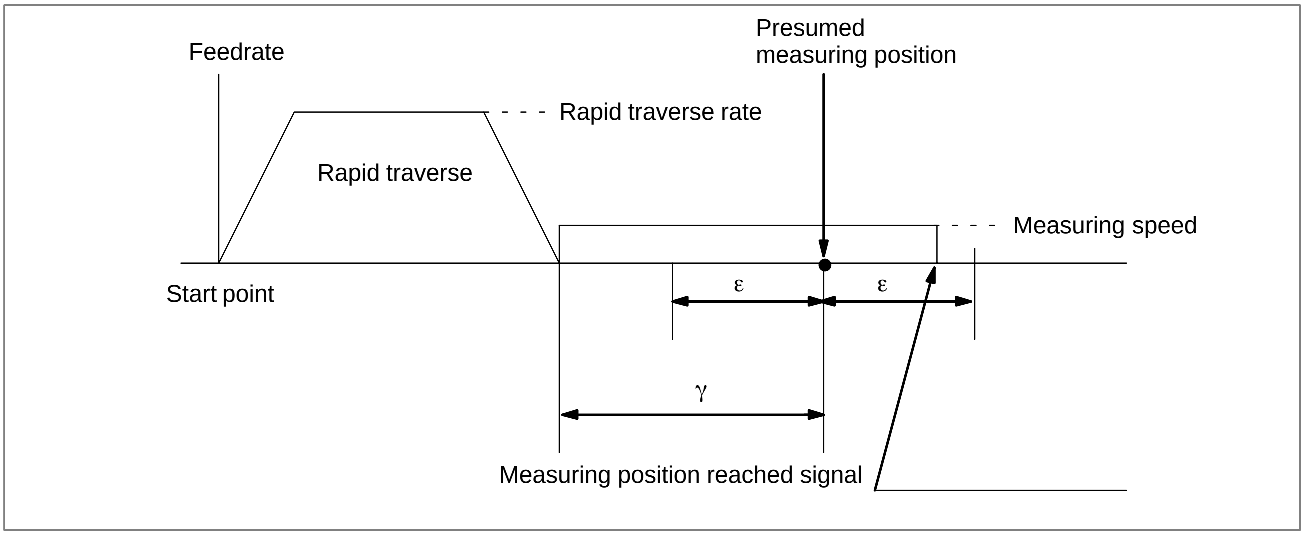
Measuring position reached signals XAE<X004#0>, YAE<X004#1>, ZAE<X004#2>(M series) XAE<X004#0>, ZAE<X004#1>(T series)

[Classification] Input signal

[Function] If the measuring position specified by a program command differs from the measuring position which a tool has reached in practice, that is, the position at the moment the measuring position reached signal has just been turned “1”, the difference in the coordinate value is added to the current tool compensation value to update the compensation value. The tool is first fed to the specified measuring position by rapid traverse in a block where one of the following commands has been specified:

G37 (M series)
G36, G37 (T series)

The tool decelerates and temporarily stops at the distance γ before the measuring position.
The tool then moves to the measuring position at the speed preset by a parameter No. 6241. If the measuring position reached signal corresponding to the G code is turned “1” after the tool has approached within distance ε of the measuring position and before the tool overshoots the measuring position by distance ε, the control unit updates the compensation value and terminates the move command for the block. If the measuring position reached signal is not turned “1” even after the tool has overshot the measuring position by distance ε, the control unit enters an alarm state and terminates the move command for the block without updating the compensation value.



[Operation] When the signal is turned “1”, the control unit works as follows:

- Reads the position of the tool along the axis currently specified and updates the current compensation value based on the difference between the specified measuring position and the read measuring position in the following case: When the measuring position reached signal corresponding to the G code is turned on in a block where G36 (T series) or G37 is specified after the tool is within distance ε of the measuring position specified by a program and before the tool overshoots the measuring position by distance ε. The control unit then stops the tool, and terminates the move command for the block.
- Enters an alarm state and terminates the move command for the block without updating the compensation value in the following case: When the measuring position reached signal corresponding to the command is turned “1” in a block where G36 (T series), G37 is specified after the tool is within distance γ of the measuring position but before the tool is within distance ε of the measuring position.
- The control unit does not monitor the measuring position reached signal for its rising edge but monitors the state of the signal. If the signal remains “1” when the next corresponding automatic tool length measurement (automatic tool offset) is specified, the control unit enters an alarm state when the tool is within distance γ of the measuring position.

> **NOTE**
> 1 The measuring position reached signal requires at least 10 msec.
> 2 The CNC directly inputs the measuring position reached signals from the machine tool; the PMC does not process them.
> 3 If automatic tool offset or automatic tool length measurement is not used, the PMC can use the signal terminals corresponding to the measuring position reached signal as the general-purpose input signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004							ZAE	XAE
						ZAE	YAE	XAE

Parameter

6241	Feedrate during measurement of automatic tool offset
	Feedrate during measurement of tool length automatic measurement

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Metric input	1 mm/min	6 to 15000	6 to 12000
Inch input	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

This parameter sets the feedrate during measurement of automatic tool offset (T series) and tool length automatic compensation (M series).

6251	γ value on X axis during automatic tool offset
	γ value during tool length automatic measurement
6252	γ value on Z axis during automatic tool offset

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the γ value during automatic tool offset (T series) or tool length automatic measurement (M series).

CAUTION
Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

6254	ε value on X axis during automatic tool offset	
	ε value during tool length automatic measurement	
6255	ε value on Z axis during tool automatic offset	

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the ε value during automatic tool offset (T series) or tool length automatic measurement (M series).

CAUTION
Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

Alarm and message

Number	Message	Description
080	G37 ARRIVAL SIGNAL NOT ASSERTED (M series)	In the automatic tool length measurement function (G37), the measurement position reached signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT ASSERTED (T series)	In the automatic tool offset function (G36, G37), the measurement position reached signal (XAE or ZAE) is not turned on within an area specified in parameter 6254, and 6255 (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37 (M series)	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
	OFFSET NUMBER NOT FOUND IN G37 (T series)	Automatic tool offset (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	H–CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool measurement (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
	T–CODE NOT ALLOWED IN G37 (T series)	T code and automatic tool offset (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37 (M series)	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
	ILLEGAL AXIS COMMAND IN G37 (T series)	In automatic tool offset (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.

Note

NOTE

1 Measurement speed, γ, and ε are set as parameters. ε must be positive numbers and satisfy the condition of $\gamma > \varepsilon$.

2 The compensation value is updated by the following formula:

New compensation value =(Current compensation value)+[(Current position of the tool along the specified axis when the measuring position reached signal is “1”) – (specified measuring position)]

The following compensation values are updated:

(1) In a M series, the compensation value corresponding to the tool compensation number selected by an H code. The tool wear compensation value for the H code is changed.

(2) In a T series, the compensation value corresponding to the tool compensation number selected by a T code and to the specified axis (X, Z) in G36, G37.

3 The maximum measuring error is calculated as shown below.

$$ERRmax = Fm \times \frac{1}{60} \times \frac{4}{1000}$$

ERRmax: Maximum measuring error (mm)
Fm : Measuring feedrate (mm/min)
If Fm = 100 mm/min, for example, ERRmax = 0.007 mm

4 After the measuring position reached signal has been detected, the tool moves for a maximum of 20 msec, then stops. Values for calculating the compensation amount, that is the coordinate of the tool where the tool reached the measuring position are not those obtained after stop, but those obtained at the position where the measuring position reached signal was detected.

The overtravel amount for 20 msec is calculated as follows.

$$Qmax = Fm \times \frac{1}{60} \times \frac{1}{1000} (20 + Ts)$$

Qmax: Maximum overtravel amount (mm)
Fm : Measuring feedrate (mm/min)
Ts : Servo time constant [msec] (1/loop gain)

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.14.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.14.5	AUTOMATIC TOOL OFFSET (G36, G37)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.14.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)

14.3 SKIP FUNCTION

14.3.1 Skip Function

General

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5068, as follows:

#5061 Fisrt axis coordinate value
#5062 Second axis coordinate value
#5063 3rd axis coordinate value
:
:

Signal

Skip Signal SKIP<X004#7> SKIPP<G006#6> (T series)

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to "1" in a block containing G31 is stored in a custom macro variable, and the move command of the block is terminated at the same time.

[Operation] When a skip signal turns to "1", the control unit works as described below.

· When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

· The skip signal is monitored not for a rising edge, but for its state. So, if a skip signal continues to be "1", a skip condition is assumed to be satisfied immediately when the next skip cutting is specified.

> **NOTE**
> 1 The skip signal width requires at least 10 msec.
> 2 The CNC directly reads the skip signal SKIP<X004#7> from the machine tool; the PMC is no longer requires to process the signal.
> 3 If the skip function G31 is not used, the PMC can use the signal terminal SKIP<X004#7> corresponding to the skip signal as a general purpose input signal.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP							

	#7	#6	#5	#4	#3	#2	#1	#0
G006		SKIPP						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	GSK
	SKF						SK0	

[Data type] Bit

GSK In skip cutting (G31), the signal SKIPP (bit 6 of G006) is:

0 : Not used as a skip signal.
1 : Used as a skip signal.

SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP <X004#7> and the multistage skip signals <X004> (for the T series only).

0 : When these signals are 1, they are assumed to be input (skip).
1 : When these signals are 0, they are assumed to be input (skip).

SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command

0 : Disabled
1 : Enabled

Alarm and message

Number	Message	Description
035	CAN NOT COMMANDED G31 (T series)	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
036	CAN NOT COMMANDED G31 (M series)	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.

Warning

WARNING

Disable feedrate override, dry run, and automatic acceleration/deceleration (enabled with parameter No. 6200#7 SKF=1) when the feedrate per minute is specified, allowing for reducing an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

Note

NOTE

1 The G31 block is always set to G01 mode. The feedrate is specified by an F code.

2 When the measuring motion is made by utilizing the skip signal, program a constant feedrate; otherwise, if the feedrate changes, the measuring error will be noticeable. With a constant feedrate, the maximum measuring error can be calculated as follows:

$$\text{ERRmax} = \text{Fm} \times \frac{1}{60} \times \frac{4}{1000}$$

ERRmax : Maximum measuring error (mm or inch)
Fm : Measuring feedrate (mm/min or inch/min)

3 Overtravel amount Qmax after skip signal has been turned to “1” is calculated by the following:

$$\text{Qmax} = \text{Fm} \times \frac{1}{60} \times \frac{1}{1000} (20^{*1} + \text{Tc} + \text{Ts})$$

Qmax : Overtravel amount (mm or inch)
Fm : Feedrate (mm/min or inch/min)
Tc : Cutting time constant (ms)
Ts : Servo time constant (ms) (1 loop gain)

*1 : The value becomes 28 when the skip signal SKIPP <G006#6> is used. (Also it changes according to the processing time of ladder program).

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (M series) (B–64124EN)	II.4.18	SKIP FUNCTION(G31)
	OPERATOR’S MANUAL (T series) (B–64114EN)	II.4.11	SKIP FUNCTION(G31)
Series 0*i* Mate–C	OPERATOR’S MANUAL (M series) (B–64144EN)	II.4.7	SKIP FUNCTION(G31)
	OPERATOR’S MANUAL (T series) (B–64134EN)	II.4.9	SKIP FUNCTION(G31)

14.3.2 High–speed Skip Signal

General

The skip function operates based on a high-speed skip signal (HDI0: connected directly to the CNC; not via the PMC) instead of an ordinary skip signal (X004#7). In this case, one signal can be input. (Either can be enabled/disabled, using parameters (bit 4 of parameter No. 6201 and bit 4 of parameter No. 6200).)

Delay and error of skip signal input is 0 to 2 msec at the CNC side (not considering those at the PMC side).

This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.

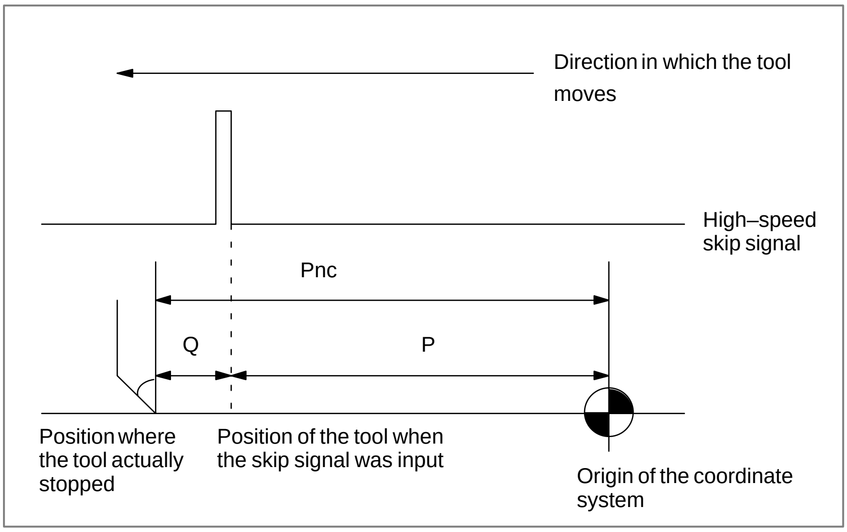
Acceleration / Deceleration and Servo Delay Compensation (Type A/B)

The skip function causes the NC to memorize the “current position” of the tool when a skip signal is input. However, the “current position” includes a delay in the servo system. In other words, the “current position” deviates by the distance corresponding to the servo delay from the position where the tool actually was when the skip signal was input. This deviation can be calculated from the positional error in the servo system and the number of remaining pulses due to feedrate acceleration/deceleration in the NC. Taking this deviation into account eliminates the necessity to include the servo delay in a measurement error.

The deviation of the “current position” can be compensated for by either of the following two types, using parameter SEA (bit 0 of parameter No. 6201) or parameter SEB (bit 1 of parameter No. 6201).

(1) Type A: The deviation is calculated from the cutting time constant and the servo time constant (loop gain).

(2) Type B: The deviation is assumed to be a sum of the number of remaining pulses due to acceleration/deceleration caused when the skip signal is turned on, and the positional error.



Pnc : Position where the tool actually stopped after the skip signal was input [mm/inch]

P : Distance to be measured [mm/inch]

Q : Servo delay [mm/inch]

Under the conditions shown above, the NC calculates the following equation using parameter SEA (bit 0 of parameter No. 6201) or SEB (bit 1 of parameter No. 6201):

$$P = Pnc - Q$$

For type A (SEA bit 0 of parameter No. 6201 is “1”), the deviation is calculated by:

$$Q = Fm \times 1/60 \times (\alpha \times Tc/1000 + Ts/1000)$$

where

Fm : Feedrate [mm/min or inch/min]

Tc : Cutting time constant [msec]

Parameter No. 1622: Exponential acceleration/deceleration

Parameter No. 1628: Linear acceleration/deceleration after interpolation

If parameter SKF (bit 7 of parameter No. 6200) = 0, Tc = 0.

Ts : Servo time constant [msec]

Assuming that the loop gain (parameter No. 1825) is G (unit: 1/s):

Ts = 1000/G

α : = 1 Exponential acceleration/deceleration

= 1/2 Linear acceleration/deceleration after interpolation

NOTE

For type A (parameter SEA (No. 6201 #0)=1), the skip signal must be turned on when the tool moves at constant feedrate.

Signal

High Speed Skip Staus Signal HDO0 <F122#0>

[Classification] Output signal

[Function] This signal informs the PMC of the input status of the high-speed skip signal HDI0.

[Output condition] Each bit is set to 1 when:

- The high-speed skip signal HDI0 is logical 1.

Each bit is set to 0 when:

- The high-speed skip signal HDI0 is logical 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F122								HDO0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Data type] Bit type

HSS 0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SRE When a high-speed skip signal is used:
0 : The signal is considered to be input 1.
1 : The signal is considered to be input 1.

	#7	#6	#5	#4	#3	#2	#1	#0
6201				IGX			SEB	SEA

[Data type] Bit type

SEA When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:
0 : Ignored.
1 : Considered and compensated (type A).

SEB When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:
0 : Ignored.
1 : Considered and compensated (type B).

IGX When the high-speed skip function is used, SKIP (bit 7 of X004), SKIPP (bit 6 of G006), and +MIT1 to –MIT2 (bits 2 to 5 of X004) are:

0 : Enabled as skip signals.
1 : Disabled as skip signals.

> **NOTE**
> SKIPP (bit 6 of G006) and +MIT1 to –MIT2 (bits 2 to 5 of X004) are enabled only when bit 0 (GSK) of parameter No. 6200 is set to 1 and bit 3 (MIT) of parameter No. 6200 is set to 1. Note also that these signals are enabled only for the T series.

	#7	#6	#5	#4	#3	#2	#1	#0
6202								1S1

1S1 For the G31 skip signal:

0 : The skip signal HDI0 is disabled.
1 : The skip signal HDI0 is enabled.

> **NOTE**
> HDI0 is high-speed skip signal.

Reference item

CONNECTION MANUAL (Hardware) (B–64113EN)	5.3	CONNECTING THE HIGH–SPEED SKIP (HDI)

14.3.3 Multi–step Skip

General

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable and cancels the remaining distance that the block was supposed to be moved when a skip signal (8 points) or high-speed skip signal (4 points) is turned on.
Also in a block specifying Q1 to Q4 after G04, this function skips a dwell when the skip signal or high speed skip signal has turned on.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Signal

Skip signal SKIP, SKIP2 to SKIP8 <X004>

[Classification] Input signal

[Function] These signals terminate skip cutting. That is, the position where a skip signal turns to “1” in a command program block containing G31P1 (or G31), G31P2, or G31P3, G31P4 is stored in a custom macro variable, and the move command of the block is terminated at the same time. Furthermore, in a block containing G04, G04Q1, G04Q2, G04Q3 or G04Q4, the dwell command of the block is terminated.

In either case, until all other commands (such as miscellaneous functions) of the block are completed, machining never proceeds to the next block.

Which of the eight skip signals is applicable to blocks containing the G codes can be determined by parameter (no. 6202 to 6206). The eight skip signals can correspond to the G codes on a one-to-one basis. One skip signal can also be made applicable to multiple G codes. Conversely, multiple skip signals can be made applicable to one G code.

[Operation] When a skip signal turns to “1”, the control unit functions as described below.

· When a block contains a G code from (G31, G31P1 to P4) for skip cutting, and the skip signal is made applicable by parameter setting to the command, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the axis was supposed to be moved in that block.

· When a block contains a G04, or G04Q1 to Q4 code for dwell, and the skip signal is made applicable by parameter setting to the command, the control unit stops dwell operation, and cancels any remaining dwell time.

· The skip signal is not monitored for a rising edge, but for its state. So, if a skip signal continues to be “1”, a skip condition is assumed to be satisfied immediately when the next skip cutting or dwell operation is specified.

Signal address

		#7	#6	#5	#4	#3	#2	#1	#0	
X004		SKIP	ESKIP / SKIP6	–MIT2 / SKIP5	+MIT2 / SKIP4	–MIT1 / SKIP3	+MIT1 / SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
		SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	ZAE / SKIP2	YAE / SKIP8	XAE / SKIP7	(M series)

WARNING

1. SKIP2 to SKIP6 are at the same addresses as skip signal ESKIP (axis control by PMC) and axial manual feed interlock signals +MIT1, –MIT1, +MIT2, and –MIT2 (direct input B for tool compensation measurements). Be careful when using both. (T series)
2. SKIP2 and SKIP6 to SKIP8 are at the same addresses as skip signal ESKIP (axis control by PMC) and measurement position arrival signal XAE, YAE, and ZAE (tool length automatic measurement). Be careful when using both. (M series)

CAUTION

If the automatic tool compensation is used, SKIP5 to SKIP8 cannot be used. (T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200			SLS	HSS				

[Data type] Bit type

HSS 0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SLS 0 : The multi–step skip function does not use high-speed skip signals while skip signals are input.
1 : The multi–step skip function uses high-speed skip signals while skip signals are input.

NOTE

Skip signals (SKIP and SKIP2 to SKIP8) do not depend on the setting of this parameter. They are always enabled. And, it is possible to set disable with parameter IGX bit 4 of No. 6201.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Data type] Bit type

1S1–1S8, 2S1–2S8, 3S1–3S8, 4S1–4S8, DS1–DS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi–step skip function.

The following table shows the correspondence between the bits, input signals, and commands.

The settings of the bits have the following meanings:

0 : The skip signal corresponding to the bit is disabled.
1 : The skip signal corresponding to the bit is enabled.

Multi–step skip function

Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P2 G04Q2	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5	1S5	2S5	3S5	4S5	DS5
SKIP6	1S6	2S6	3S6	4S6	DS6
SKIP7	1S7	2S7	3S7	4S7	DS7
SKIP8	1S8	2S8	3S8	4S8	DS8

NOTE
HDI0 to HDI3 are high-speed skip signals.

Note

NOTE

The skip cutting commands G31 P1, G31 P2, G31 P3, and G31 P4 are all identical, except that they correspond to different skip signals. The tool moves along the specified axis performing linear interpolation until the SKIP signal is set to "1" or the end point of the specified movement is reached. The feedrate is specified in the program. G31 is the same as G31 P1.

Dwell commands G04, G04 Q1, G04 Q2, G04 Q3, and G04 Q4 are also identical, except that they correspond to different skip signals.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	II.4.10	Multi–step Skip (G31)
	OPERATOR'S MANUAL (T series) (B–64114EN)	II.4.11	Multi–step Skip (G31)
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	II.4.10	Multi–step Skip (G31)
	OPERATOR'S MANUAL (T series) (B–64134EN)	II.4.10	Multi–step Skip (G31)

14.3.4 Torque Limit Skip

General

Specifying a move command after G31 P99 (or G31 P98) with a motor torque limit set (for example, specifying a torque limit on the PMC window) allows the same cutting feed as that specified with G01 to be performed.

While the tool is moved with a motor torque limit set during cutting feed, skip is performed when a signal indicating that the motor torque limit has been reached is input as a result of an operation such as pushing something against the motor.

- **Basic operations**

When the motor torque limit is reached or the SKIP signal <X0004#7> is input during the execution of G31 P99, the execution of the next block starts without executing the remaining portion of the move command.

When the motor torque limit is reached during the execution of G31 P98, the execution of the next block starts without executing the remaining portion of the move command. (The SKIP signal does not affect the execution of G31 P98.)

When no torque limit is specified before executing G31 P99 or P98, the move command is executed without performing the skip operation.

For G31 P99 and P98, the coordinate, indicating the position to which the tool is to be positioned after skip, is stored in the system variable of the custom macro.

Alarm No. 244 occurs if errors have accumulated to an amount (32767) that cannot be corrected in one distribution before the torque–limit–reached signal is input during the execution of G31 P99 or P98.

Signal

Torque limit reached signals TRQL1 to TRQL4 <F114>

[Classification] Output signal

[Function] Indicates that the torque limit has been reached.

[Output condition] Set to “1” when:

· The torque limit has been reached for the corresponding axis.

Set to “0” when:

· The torque limit has not been reached for the corresponding axis

Numbers 1 to 4 indicate the corresponding axis numbers.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F114					TRQL4	TRQL3	TRQL2	TRQL1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6201					TSA	TSE		

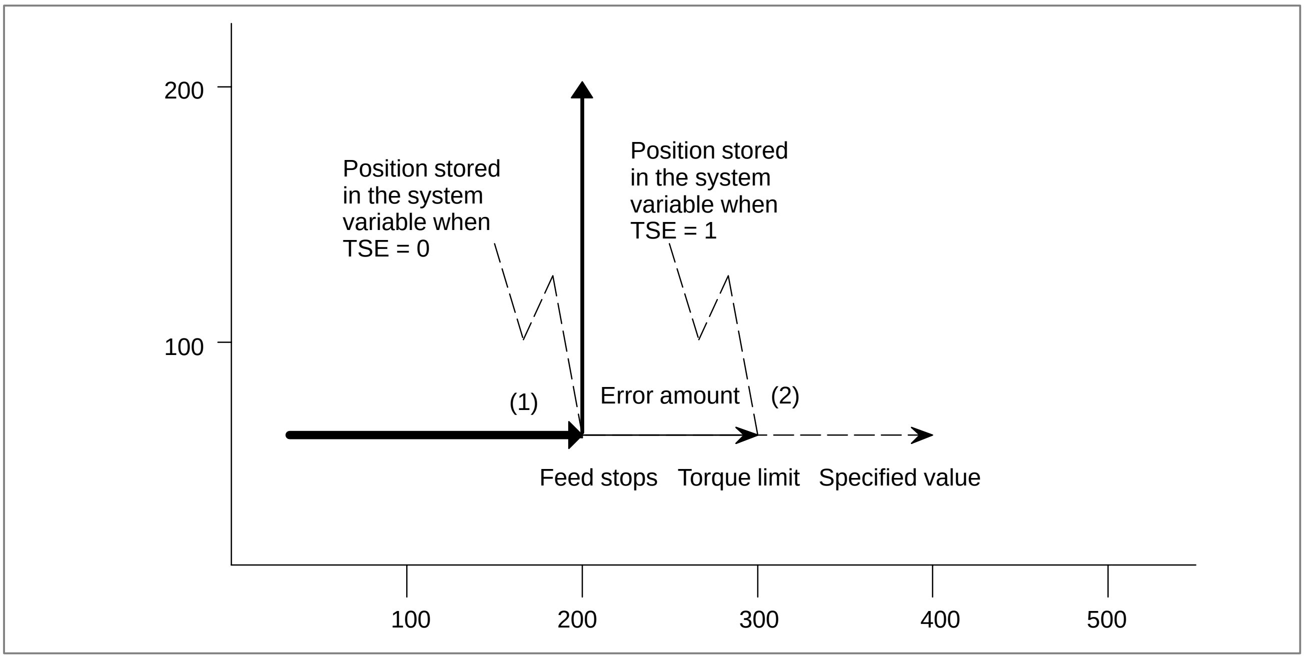
[Data type] Bit type

TSE When a skip operation is performed by the G31 P99 or P98 command used to specify torque limit skip:

0 : Corrects servo errors. (1)
1 : Does not correct servo errors. (2)

TSA Selects the axes to be monitored for whether the torque limit has been reached during the execution of the G31 P99 or 98 command used to specify torque limit skip:

0 : Monitors all axes.
1 : Monitors the axes specified by G31 P99 or P98.



Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED	In the block including the command for the skip function (G31 P99/P98), to be executed under the control of the torque limit reach signal, no axis move command is specified, or two or more axes are specified. In a single block, specify one axis only.
244	P/S ALARM	When the skip function to be executed under the control of the torque limit reach signal is enabled, an error value (32767) that exceeds the maximum return value that can be handled with a single distribution is detected before input of the torque limit reach signal. Retry the processing after changing the axis feedrate, torque limit, or other conditions.

Caution

CAUTION

1 Specify a torque limit before G31 P99/P98. If G31 P99/P98 is executed with no torque limit specified, the move command is executed without a skip operation.
2 When G31 P99 is specified, the SKIP signal causes a skip operation. However, avoid using the high–speed skip operation.
3 Before specifying G31 P99/P98, cancel tool–tip radius compensation with G40.
4 Set the SKF bit (bit 7 of parameter No. 6200) to 0 to disable the dry run, override, and automatic acceleration/deceleration functions for the G31 skip command.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	II.4.12	TORQUE LIMIT SKIP
Series 0*i* Mate–C	OPERATOR’S MANUAL (T series) (B–64134EN)	II.4.11	TORQUE LIMIT SKIP

14.4 ENTERING COMPENSATION VALUES

14.4.1 Input of Offset Value Measured A (T series)

General

This is a function of setting an offset value by key-inputting a workpiece diameter manually cut and measured from the MDI keyboard.

First the workpiece is cut in the longitudinal or cross direction manually. When the position record signal is turned “1” (prepare a button on the machine operator’s panel) on completion of the cutting, the workpiece coordinate value of X axis and Z axis at that time is recorded in the CNC. Then, withdraw the tool, stop the spindle, and measure the diameter if the cutting was on the longitudinal direction or measure the distance from the standard face if the cutting was on the facing. (The reference face is made as Z = 0.) When the measured value is entered on the offset value display screen, NC inputs the difference between the input measured value and the coordinate value recorded in NC, as the offset value of the offset number.

If you release the tool without moving the tool in the axis along which an offset value is entered but moves the tool along the other axis, an offset value can be set without using the position record signal.

The workpiece coordinate system can be shifted using the technique of directly inputting the measured value for offset. This technique is used when the coordinate system planned in the program does not match with the coordinate system actually set.

The procedures are the same as those for direct input for offset, except a difference of using the standard tool on the work shift screen.

Signal

Position record signal PRC <G040#6>

[Classification] Input signal

[Function] This signal is prepared for the function of input of offset value measured A. It is used to store in the control unit the data on the positions of the tool for tentative cutting. After measuring a dimension of the workpiece, input the measured value by the specified manual operation. The difference is then stored as the specified tool compensation value.

[Operation] The control unit stores the current position along X and Z axes when the signal turns to “1”.

> **NOTE**
> To use this signal, set parameter PRC (No.5005#2) to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G040		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

[Data type] Bit type

PRC Direct input of tool offset value and workpiece coordinate-system shift value

0 : Not use a PRC signal
1 : Uses a PRC signal

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.2	Direct Input of Tool Offset Measured
Series 0*i* Mate–C	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.2	Direct Input of Tool Offset Measured

14.4.2 Input of Tool Offset Value Measured B (T series)

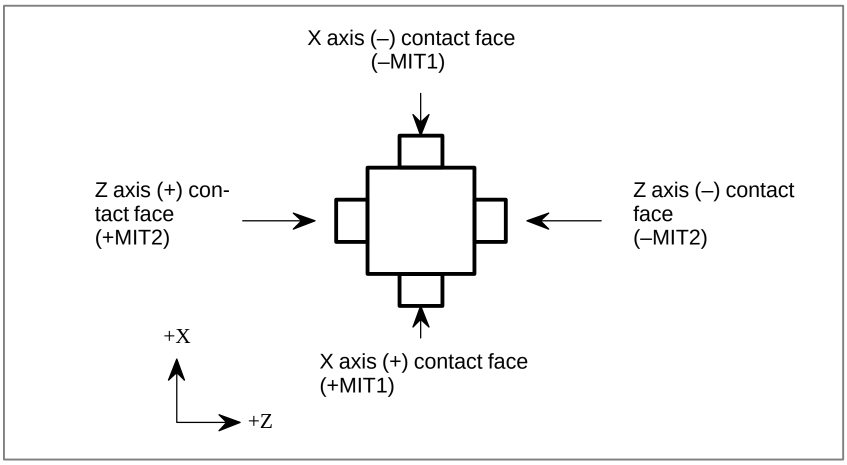
General

When the touch sensor is provided, the tool offset value can be automatically settable in the tool offset memory, by moving the tool to make contact with the touch sensor during manual operation. The workpiece coordinate system shift amount can also be automatically set.

- **Touch sensor**

Either of the following two cases may be selected depending on parameter setting.

1) If TS1 (bit 3 of parameter No. 5004) is 0
   The touch sensor has two direction–specific contact faces for each axis, thus outputting four signals (+MIT1, +MIT2, –MIT1, and –MIT2) when contact is detected.
2) If TS1 (bit 3 of parameter No. 5004) is 1
   A touch sensor based on one contact input outputs one signal (+MIT1) when the one–contact input detects contact. So, which of the two directions of each axis is involved is automatically determined, and feeding in the corresponding axis direction is disabled.



Touch sensor

- **Setting tool offset value**

Determine a specific point on the machine tool as the measuring reference position. In advance, set the distance from this point to the measuring position (contact face of the touch sensor) as a reference value, using parameter No. 5015 to 5018. Select the tool whose offset value is to be measured, and bring it to touch the sensor, receiving a contact detection signal (tool compensation value write signal). The mechanical coordinate value is the distance from the tool nose position of the measuring tool at the mechanical reference (home) position to the measuring position; set the difference between this value and the reference value (parameter setting) into the tool offset value memory as the tool geometry offset value. The corresponding tool wear offset value becomes 0.

(Tool offset value to be set)

= (Mechanical coordinate value when tool compensation value write signal has become "1") – (Reference value (parameter value) corresponding to the tool compensation value write signal)

The tool offset value to be set differs according to the method of determining the measuring reference position.

If touch sensor contact detection is based on a one–contact input

If touch sensor contact detection is based on a one–contact input (the TS1 parameter (bit 3 of parameter No. 5004) is 1), when a contact detected signal (tool compensation amount write signal +MIT1) is received from the touch sensor, which of the two directions of each axis is involved is automatically determined according to several pulses stored as a result of the axis movement that was made before the signal reception. So the number of interpolation cycles related to the stored pulses must previously be set in parameter No. 5021.

Once which of the two directions of each axis is involved is automatically determined, the corresponding axis direction is subjected to axis interlock to stop feeding, and the obtained tool compensation amount is stored in tool compensation memory.

If the directions of stored pulses are not unified, or no pulse has been stored, for example, because the servo power has been shut off (servo off) or no axis movement occurred, a P/S alarm (No. 5195) is issued.

A P/S alarm is issued also if the tool moves along two axes (X–axis and Z–axis) simultaneously; move the tool along one axis at a time.

If a P/S alarm is issued, no tool compensation amount will be set up, resulting in the four directions (two axes) being subjected to interlock.

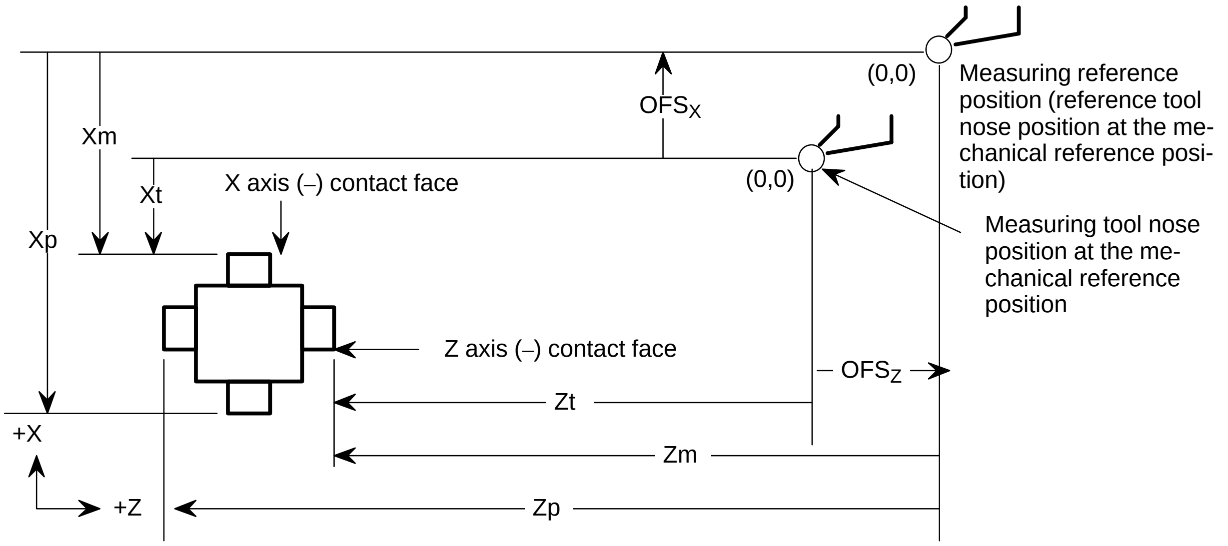
NOTE

1 Pulse storage for automatic decision is carried out in the manual mode while the GOSQM <G039#7> (tool compensation amount write mode select) signal is 1. Stored pulses are lost if:
   a. The manual mode is exited,
   b. The GOSQM <G039#7> (tool compensation amount write mode select) signal becomes 0,
   c. A contact detected signal is received from the touch sensor, resulting in a tool compensation amount being set or a P/S alarm (No. 5195) being issued,
   d. The servo power becomes off (resulting in the stored pulses for the related axis being lost), or
   e. Axis movement occurs (resulting in the stored pulses for the other axis being lost).

2 Axis interlock that has occurred for the axis direction identified by parameter–based automatic decision and two–axis, four–direction interlock that has occurred because of a P/S alarm being issued are canceled when the manual mode is exited or the GOSQM <G039#7> (tool compensation amount write mode select) signal becomes 0.
   A reset does not cancel interlock.

Example 1

The difference between the reference tool nose tip position and the measuring tool nose tip position can be set as the tool offset value. Define the reference tool nose tip position at the mechanical reference position (machine zero position) as the measuring reference position, then set the distances Xp, Zp, Xm, Zm, from the measuring reference position to the contact faces of the sensor in parameters.



XP : Distance from the measuring reference position to X-axis (+) contact face (parameter no. 5015)

Xm : Distance from the measuring reference position to X-axis (–) contact face (parameter no. 5016)

Zp : Distance from the measuring reference position to Z-axis (+) contact face (parameter no. 5017)

Zm : Distance from the measuring reference position to Z-axis (–) contact face (parameter no. 5018)

Xt : X-axis direction moving distance of the measuring tool up to the contact face of sensor (X-axis machine coordinate value)

Zt : Z-axis direction moving distance of the measuring tool up to the contact face of sensor (Z-axis machine coordinate value)

(when Xt and Zt touch the X-axis (–) contact face and Z-axis (–) contact face in the above figure)

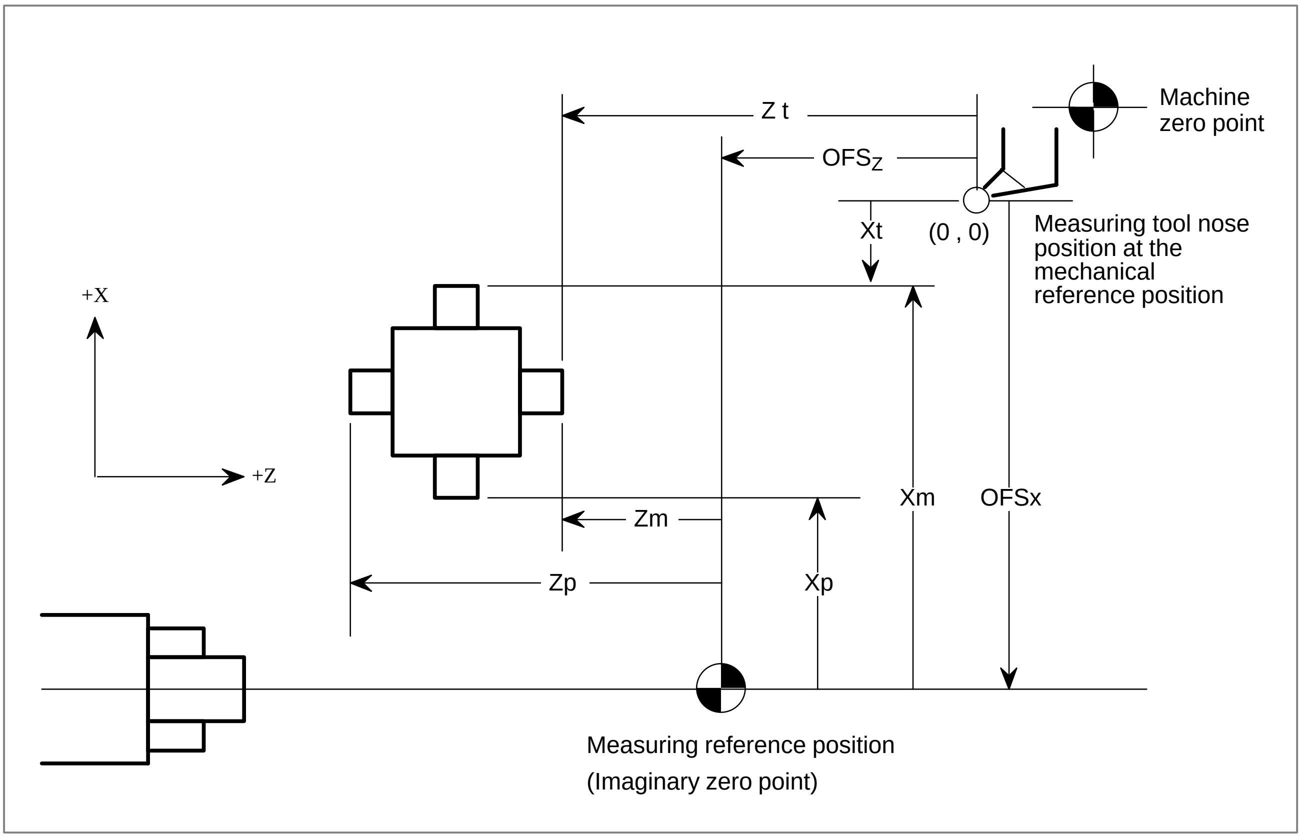
OFSx : Tool offset value to be set (X-axis): OFSx = Xt - Xm

OFSz : Tool offset value to be set (Z-axis): OFSz = Zt - Zm

When the reference tool nose tip position is set as the measuring reference position

Example 2

The measuring reference point may be an imaginary point (imaginary zero point), as shown in the figure below. The difference between the imaginary zero point and the measuring tool nose tip position at the mechanical reference point can be set as the tool offset value of the measuring tool, by setting the distances from the imaginary zero point to the respective contact faces in parameters.



When the imaginary zero position is set as the measuring reference position

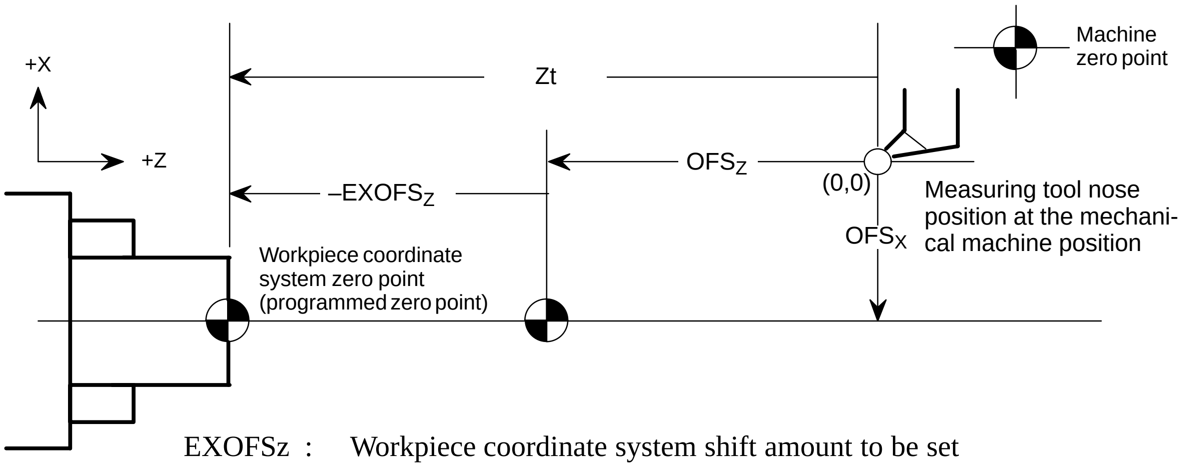
- **Setting the workpiece coordinate system shift amount**

The workpiece coordinate system shift amount for the Z-axis can be set as follows: Bring the tool to touch the workpiece end face. Subtract the tool geometry offset value of the tool (the value shifted in the coordinate system by the tool geometry offset) from the machine coordinate value (the distance from the measuring tool nose tip position at the mechanical reference position (machine zero point) to the workpiece end face).The result is set as the workpiece coordinate system shift value.

(Z axis workpiece coordinate system shift amount to be set (EXOFSz))

= (Z axis tool geometry offset value of the corresponding tool (OFSz)) – (Z axis machine coordinate value(Zt))

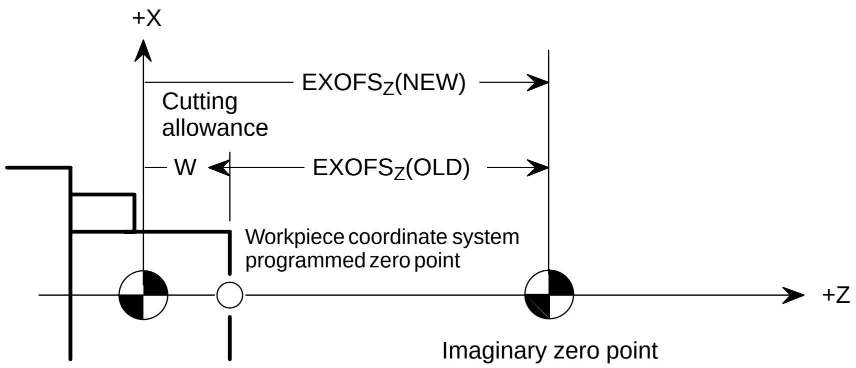
Using the above methods, the workpiece coordinate system is set with the workpiece end face (the contact point of the sensor) specified as the programmed zero point of the workpiece coordinate system of the Z-axis.



EXOFSz	:	Workpiece coordinate system shift amount to be set
OFSz	:	Tool geometry offset value
Zt	:	Mechanical coordinate value (Distance to the workpiece end face)

Setting of workpiece coordinate system shift amount

To deviate the programmed zero point of the workpiece coordinate system from the workpiece end face, such as by adding a cutting allowance, use the incremental input of the workpiece coordinate system shift amount in MDI operation. By setting the distance from the programmed zero point to the workpiece end face with a sign, the numeric value input is added to the preset amount.



$EXOFS_Z$(OLD)	:	Workpiece coordinate system shift amount being set by the function of input of tool offset value measured B
W	:	Cutting allowance (incremental input value)
$EXOFS_Z$(NEW)	:	Workpiece coordinate system shift amount after adding the cutting allowance.

Setting of cutting allowance

Basic Procedure to Set Tool Offset Value

(1) Execute manual reference position return.
By executing manual reference position return, a machine coordinate system is established.
The tool offset value is computed on the machine coordinate system.

(2) Select manual handle mode or manual continuous feed mode and set the tool compensation value write mode select signal GOQSM to “1”.
The screen is automatically changed to the tool offset screen (geometry), and the “OFST” indicator starts blinking in the status indication area at the bottom of the screen, which indicates that the tool compensation value writing mode is ready.

(3) Select a tool to be measured.

(4) When the cursor does not coincide with the tool offset number desired to be set, move the cursor to the desired offset number using the page key and cursor key.
The cursor can also be coincided with the tool offset number desired to be set automatically by the tool offset number input signals (when parameter QNI(No.5005#5)=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.

(5) Near the tool to the sensor by manual operation.

(6) Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the tool compensation value writing signals to input to be CNC.
The tool compensation value writing signal is set to “1”, and the :
The following tool compensation amount write signals are set up according to the setting of the TS1 parameter (bit 3 of No. 5004).
When the parameter is 0: +MIT1, –MIT1, +MIT2, –MIT2
When the parameter is 1: +MIT1 only

i) The axis is interlocked in this direction and its feed is stopped.

ii) The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.

(7) For both X-axis and Z-axis, their offset values are set by operations **(5)** and **(6)**.

(8) Repeat operations **(3)** to **(7)** for all necessary tools.

(9) Set the tool compensation value writing mode signal GOQSM to “0”.
The writing mode is canceled and the blinking “OFST” indicator light goes off.

Basic Procedure to Set Workpiece Coordinate Shift Value

(1) Set the tool geometry offset values of each tool in advance.

(2) Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shift amount is computed based on the machine coordinate system of the tool.

(3) Set the workpiece coordinate system shift amount writing mode select signal WOQSM to “1”.
The CRT display automatically switches to the workpiece shifting screen, the “WFST” indicator starts blinking at the status indicator area in the bottom of the screen, which indicates that the workpiece coordinate system shift amount writing mode is ready.

(4) Select a tool to be measured.

(5) Check tool offset numbers.
The tool offset number corresponding to the tool required for measurement, shall be set in the parameter (No.5020) in advance.
The tool offset number can also be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).

(6) Manually approach the tool to an end face of the workpiece.

(7) Place the tool edge to the end face (sensor) of the workpiece using manual handle feed.
When the tool edge contacts the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET.
The workpiece coordinate system shift amount on the Z–axis is automatically set.

(8) Release the tool.

(9) Set the workpiece coordinate system shift amount write mode select signal WOQSM to “0”.
The writing mode is canceled and the blinking “WSFT” indicator light goes off.

Signal

Tool offset write mode select signal GOQSM <G039#7>

[Classification] Input signal

[Function] Selects the mode for writing tool compensation.

[Operation] When this signal is turned “1” in a manual operation mode, the mode for writing tool compensation is selected. The control unit then automatically switches the screen displayed on the CRT to the tool geometry compensation screen and blinks the “OFST” status display at the bottom of the screen to notify that the mode has been changed to the mode for writing tool compensation.

Tool offset write signal +MIT1, +MIT2 <X004#2, #4> –MIT1, –MIT2 <X004#3, #5>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to "1", the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

[Operation] When these signals are turned "1" during tool offset write mode, the control unit operates as follows:

- Inhibits tools from being fed along the corresponding axis during manual operation.
  If the TS1 parameter (bit 3 of 5004) is 0
  +MIT1 : Inhibits the tool from being manually fed in the positive direction along the X-axis.
  –MIT1 : Inhibits the tool from being manually fed in the negative direction along the X-axis.
  +MIT2 : Inhibits the tool from being manually fed in the positive direction along the Z-axis.
  –MIT2 : Inhibits the tool from being manually fed in the negative direction along the Z-axis.
  If the TS1 parameter (bit 3 of 5004) is 1
  +MIT1 : Automatic decision causes the related two directions of each axis to be subjected to interlock.
  –MIT1 : Not used
  +MIT2 : Not used
  –MIT2 : Not used
- When signal GOQSM for selecting the mode for writing tool compensation is turned "1", the manual feed interlock signal also automatically calculates the tool geometry compensation for the tool compensation number pointed to by the cursor and sets the result in tool compensation memory.

NOTE
1 This signal is used as the manual feed interlock signal in each axis direction.
2 This signal is valid only when the GSC parameter (bit 0 of No. 5009) is 0.

Tool offset write signal
+MIT1, +MIT2
<G132#0, #1>
–MIT1, –MIT2
<G134#0, #1>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to "1", the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

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

- Inhibits tools from being fed along the corresponding axis during manual operation.

  +MIT1 : Inhibits the tool from being manually fed in the positive direction along the X-axis.

  –MIT1 : Inhibits the tool from being manually fed in the negative direction along the X-axis.

  +MIT2 : Inhibits the tool from being manually fed in the positive direction along the Z-axis.

  –MIT2 : Inhibits the tool from being manually fed in the negative direction along the Z-axis.

  If the TS1 parameter (bit 3 of 5004) is 1

  +MIT1 : Automatic decision causes the related two directions of each axis to be subjected to interlock.

  –MIT1 : Not used

  +MIT2 : Not used

  –MIT2 : Not used

- When signal GOQSM for selecting the mode for writing tool compensation is turned "1", the manual feed interlock signal also automatically calculates the tool geometry compensation for the tool compensation number pointed to by the cursor and sets the result in tool compensation memory.

> **NOTE**
> This signal is valid only when the bit 0 (GSC) of parameter No. 5009 is 1.

Tool offset number select signals OFN0 to OFN5 <G039#0 to #5>

[Classification] Input signal

[Function] Selects the tool offset number.

[Operation] When the mode for writing tool compensation is selected, the cursor is automatically positioned on the tool geometry compensation number selected by these signals.
A tool offset number is specifed by a 7–bit binary number. Numbers 0 to 63 correspond to the compensation number 1 to 64.

> **NOTE**
> This signal is available only when parameter QNI (No. 5005#5) =1.

Workpiece coordinate system shift value write mode select signal WOQSM <G039#6>

[Classification] Input signal

[Function] Selects the mode for writing the shift amount for the workpiece coordinate system.

[Operation] When this signal is turned to “1” in a manual operation mode, the mode for writing the shift amount for the workpiece coordinate system is selected. The control unit then automatically switches the screen displayed to the WORK SHIFT screen and blinks the “OFST” status display at the bottom of the screen to notify that the mode has been changed to the mode for writing the shift amount for the workpiece coordinate system. However, this is not performed when the mode for writing tool compensation values is selected.

Workpiece coordinate system shift value write signal WOSET <G040#7>

[Classification] Input signal

[Function] Automatically calculates and sets the shift amount for the workpiece coordinate system.

[Operation] When this signal turns to “1” in the mode for writing the shift amount for the workpiece coordinate system, it triggers the automatic calculation and setting of the shift amount for the workpiece coordinate system.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004			–MIT2	+MIT2	–MIT1	+MIT1		
			SKIP5	SKIP4	SKIP3	SKIP2		

WARNING
Since the same addresses are used for both +MIT1, –MIT1, +MIT2, –MIT2 and skip signals SKIP2 to SKIP5 (multi–step skip), be careful when using these two signal types.

	#7	#6	#5	#4	#3	#2	#1	#0
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040	WOSET							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT			

[Data type] Bit

DIT Interlock for each axis direction
0 : Enabled
1 : Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
5004					TS1			

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

[Data type] Bit

TS1 When the tool offset measurement value direct input B function is used, touch sensor contact detection is based on:
0 : Four–contact input.
1 : One–contact input.

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI					

[Data type] Bit

QNI The function to input offset value measured B
0 : Does not automatically select the tool offset number
1 : Automatically selects a tool offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5009								GSC

[Data type] Bit

GSC Specifies from which the offset write input signal is to be input for the tool compensation amount measurement value direct input B function, as follows:

0 : To be input from the machine.
1 : To be input from the PMC.

> **NOTE**
> 1 After setting this parameter, turn the power off then on again so that the setting will take effect.
> 2 If axis direction–specific interlock is enabled (bit 3 or parameter No. 3003 (DIT) is 0), its input is also switched from the machine to the PMC.

5015	Distance (X1P) between reference position and X axis + contact surface (touch sensor 1 side)
5016	Distance (X1M) between reference position and X axis – contact surface (touch sensor 1 side)
5017	Distance (Z1P) between reference position and Z axis + contact surface (touch sensor 1 side)
5018	Distance (Z1M) between reference position and Z axis – contact surface (touch sensor 1 side)
5056	Distance (X2P) between reference position and X axis + contact surface (touch sensor 2 side)
5057	Distance (X2M) between reference position and X axis – contact surface (touch sensor 2 side)
5058	Distance (Z2P) between reference position and Z axis + contact surface (touch sensor 2 side)
5059	Distance (Z2M) between reference position and Z axis – contact surface (touch sensor 2 side)

[Data type] Two–word

[Unit of data]

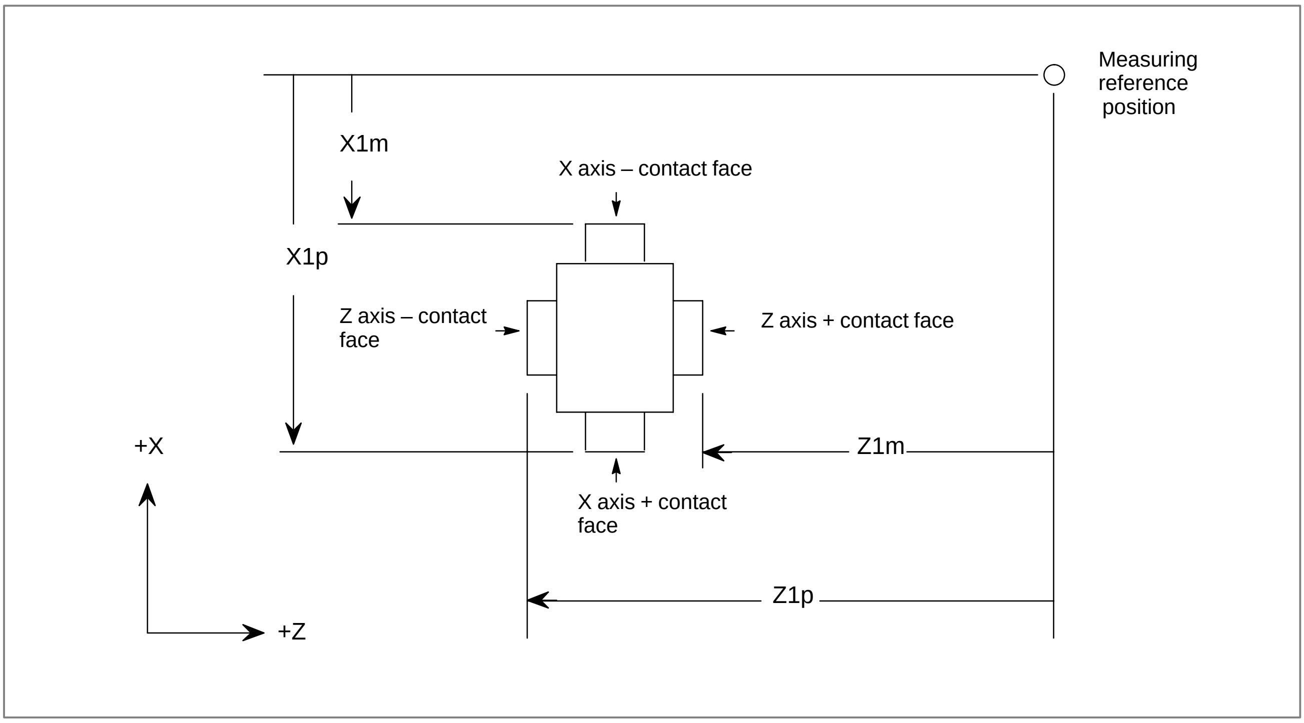
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

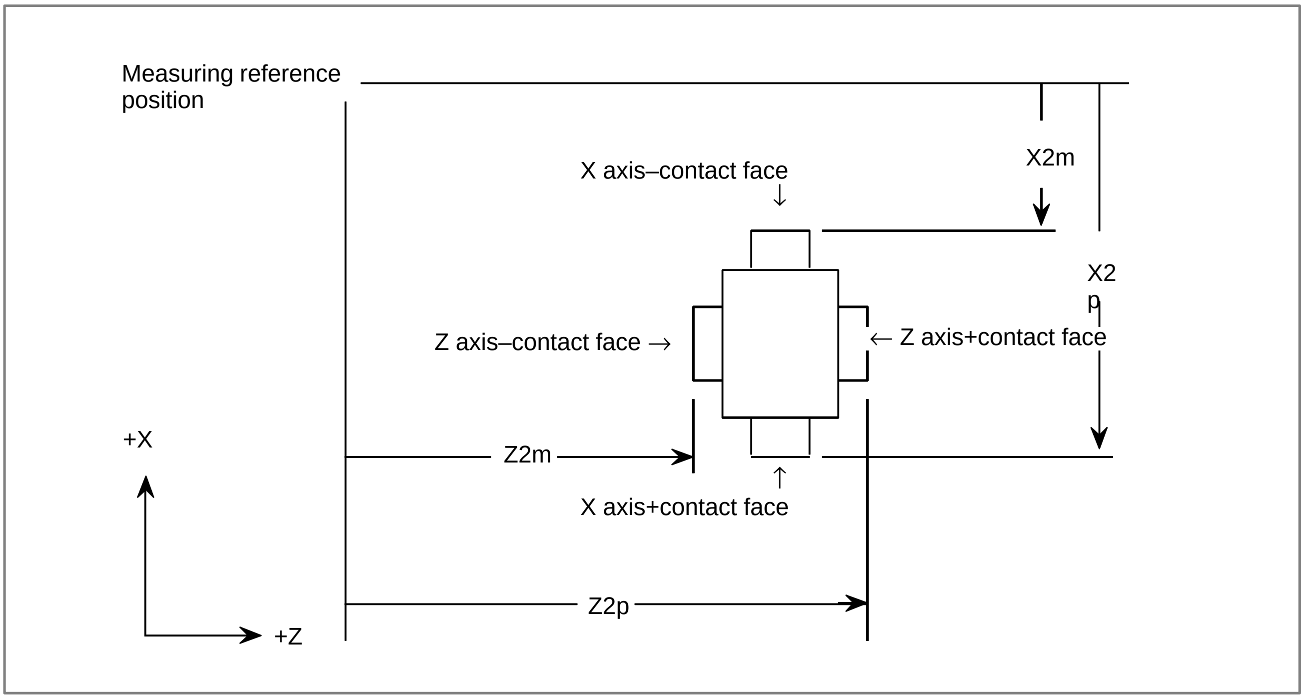
[Valid data range] –99999999 to 99999999

These parameters are related to the function to input tool offset value measured B.

They set the distance (with sign) between the measurement reference position and sensor contact surface. For an axis under diameter programming, set it using a diameter value.

NOTE
Parameter Nos. 5056 to 5069 are valid if the 2NR parameter (bit 0 of parameter No. 5051) is 1.





5020	Tool offset number used for the input of tool offset value measured B

[Data type] Byte

[Valid data range] 0 to the number of tools to be compensated.

Set tool offset number used for the input of tool offset value measured B function (i.e. when workpiece coordinate system shift value is set). (The tool offset number corresponding to the measured tool shall be set in advance.) This parameter is valid when the tool offset number is not selected automatically (QNI, #5 of parameter 5005, is zero).

5021	Number of pulse interpolation cycles memorized prior to contacting the touch sensor

[Data type] Byte

[Unit of data] Interpolation cycle

[Valid data range] 0 to 8

This parameter sets the number of pulse interpolation cycles to be memorized until the operator manually touches the tool with a one–contact input touch sensor when the tool offset measurement value direct input B function is used.

If 0 is set for this parameter, the specification of 8 (maximum allowable value) is assumed.

NOTE
This parameter is enabled when the TS1 parameter (bit 3 of parameter No.5004) is set to 1.

Alarm and message

Number	Message	Description
5195	DIRECTION CAN NOT BE JUDGED	For a one–contact input touch sensor used with the tool compensation amount measurement value direct input B function, stored pulse directions are not unified. • The machine is at a stop in the offset write mode. • The servo power is off. • Pulse directions are diverse. Alternatively, the tool is moving along two axes (X–axis and Z–axis) simultaneously.

Reference item

Series 0*i*–C	OPERATOR’S MANUAL (T series) (B–64114EN)	III.11.4.3	Direct Input of Tool Offset Measured B
Series 0*i* Mate–C	OPERATOR’S MANUAL (T series) (B–64134EN)	III.11.4.3	Direct Input of Tool Offset Measured B

14.4.3 Input of Measured Workpiece Origin Offsets

General

By directly entering the measured deviation of the actual coordinate system from a programmed work coordinate system, the workpiece zero point offset at the cursor is automatically set so that a commanded value matches the actual measurement.

Reference item

Series 0*i*–C	OPERATOR'S MANUAL (M series) (B–64124EN)	III.11.4.7	Input of Measured Workpiece Origin Offsets
	OPERATOR'S MANUAL (T series) (B–64114EN)	III.11.4.11	Input of Measured Workpiece Origin Offsets
Series 0*i* Mate–C	OPERATOR'S MANUAL (M series) (B–64144EN)	III.11.4.7	Input of Measured Workpiece Origin Offsets
	OPERATOR'S MANUAL (T series) (B–64134EN)	III.11.4.10	Input of Measured Workpiece Origin Offsets

15 PMC CONTROL FUNCTION

15.1 PMC AXIS CONTROL

15.1.1 PMC Axis Control

General

The PMC can directly control any given axis, independent of the CNC. In other words, moving the tool along axes that are not controlled by the CNC is possible by entering commands, such as those specifying move distance and feedrate, from the PMC. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

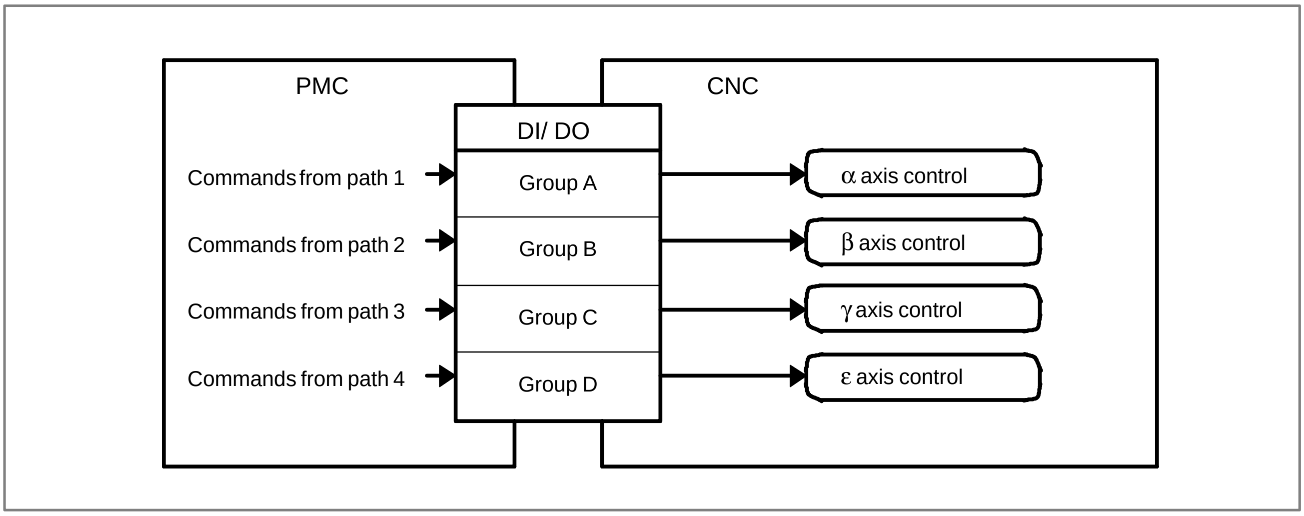
Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

The PMC can directly control the following operations:

(1) Rapid traverse with move distance specified
(2) Cutting feed – feed per minute, with move distance specified
(3) Cutting feed – feed per revolution, with move distance specified
(4) Skip – feed per minute, with move distance specified
(5) Dwell
(6) Continuous feed
(7) Reference position return
(8) 1st reference position return
(9) 2nd reference position return
(10) 3rd reference position return
(11) 4th reference position return
(12) External pulse synchronization – Main spindle
(13) External pulse synchronization – first manual handle
(14) External pulse synchronization – second manual handle
(15) External pulse synchronization – third manual handle (for M series only)
(16) Feedrate control
(17) Auxiliary function, Miscellaneous function 2, Miscellaneous function 3
(18) Selection of the machine coordinate system
(19) Torque control

The PMC is provided with four paths to control these operations using input and output signals.

By issuing commands through these four paths, the PMC can simultaneously control multiple separate axes. Use parameter No. 8010 to determine which path controls which axis. Commands may be issued through one path to two or more axes, thus allowing the PMC to control multiple axes using one path.



In the following description, input/output signals from the four paths are called group A (path 1), group B (path 2), group C (path 3), and group D (path 4), respectively.

The name of an input/output signal used for PMC axis control always contains a lowercase “g”, as in EBUFg. However, there is no such signal as EBUFg. The actual signal names represented by EBUFg are EBUFA, EBUFB, EBUFC, and EBUFD, which respectively correspond to signals of group A (path 1), group B (path 2), group C (path 3), and group D (path 4).

Basic procedure

(1) In parameter No. 8010, specify which DI/DO signal group (A, B, C, or D) is to be used for PMC axis control on a per–axis basis.

When using the same group for simultaneously controlling two or more axes, check that the settings of the parameters related to feedrate (rapid traverse rate, acceleration/deceleration time constant, diameter/radius, linear axis/rotation axis, etc.) are identical for each axis to be controlled.

(2) To enable direct PMC axis control, set each control axis selection signal (EAX1 to EAX4), that corresponds to an axis to be controlled, to 1.

(3) Determine the operation.

The axis control command signals (EC0g to EC6g) specify the type of operation. The axis control feedrate signals (EIF0g to EIF15g) specify the feedrate. The axis control data signals (EID0g to EID31g) specify the moving distance and other data.

These signals, together with block stop prohibition signal EMSBKg (described later), determine one complete operation, which is tantamount to one block executed during CNC–controlled automatic operation. These signals may be collectively called the axis control block data signals.

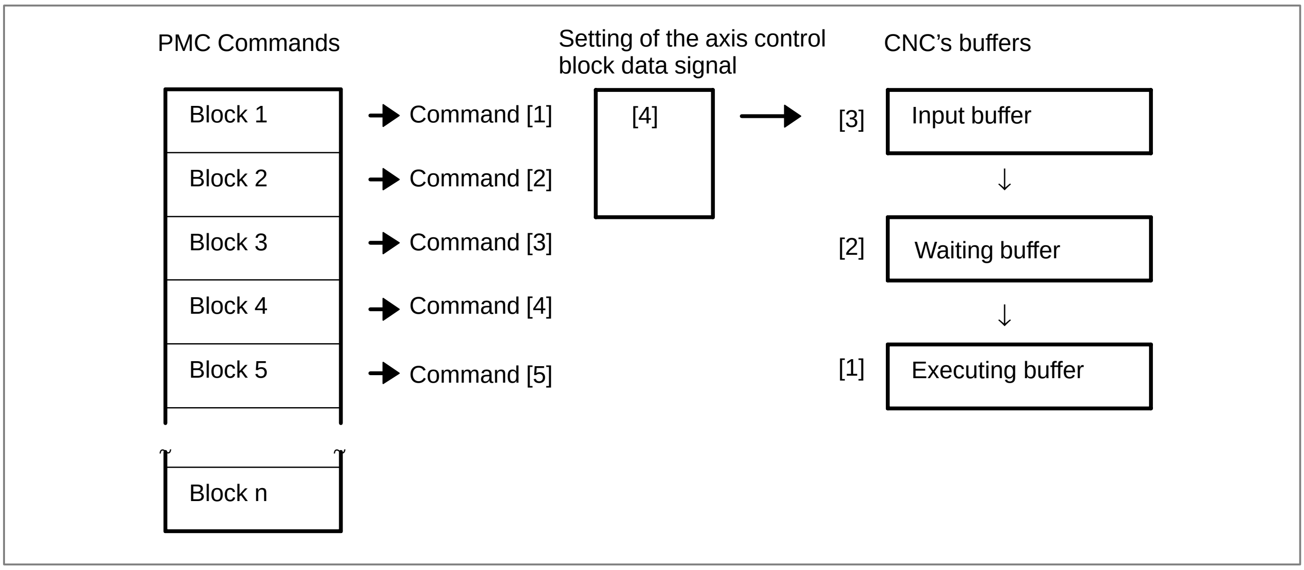
⊙ List of Signals Determining Data, Tantamount to One Block for PMC Axis Control

Generic name	Signal name	Symbol	Data type
Axis control block data signals	Block stop prohibition signal	EMSBKg	Bit
	Axis control command signal	EC0g to EC6g	Byte
	Axis control feedrate signal	EIF0g to EIF15g	Word
	Axis control data signal	EID0g to EID31g	Two–word

(4) When the data governing a complete operation (one block) is determined, reverse the logical state of the axis control command read signal EBUFg (i.e., from “0” to “1” or vice versa). Note that, for this to occur, axis control command read completion signal EBSYg must be in the same logical state as EBUFg.

- The CNC is capable of storing axis control functions from the PMC in its buffer so that multiple operations can be performed in series, under the control of the PMC. This allows the CNC to accept a new command block from the PMC during the execution of another block if the buffer has free space.

  The following figure illustrates an example in which command [1] is being executed, commands [2] and [3] are stored in the buffers, and command [4] has been issued (the axis control block data signal is set).



When the execution of command [1] is completed:

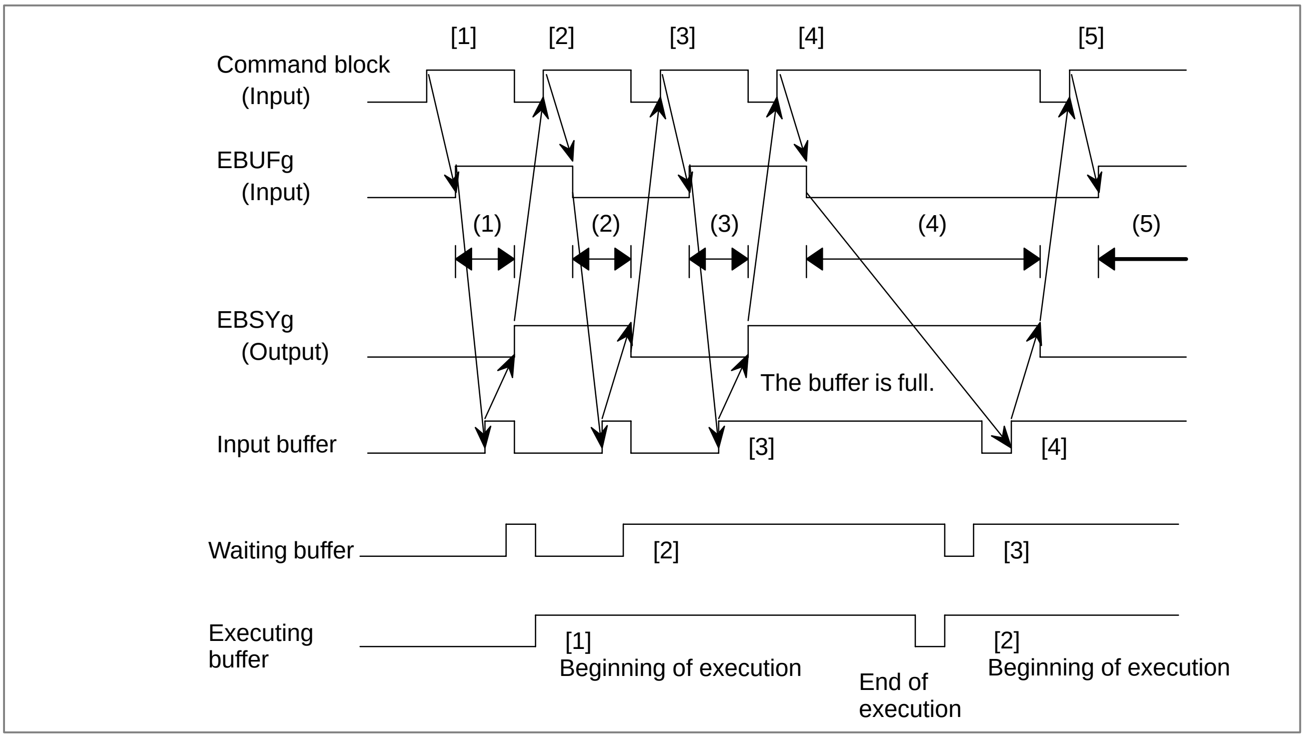
· command [2] is transferred from the waiting buffer to the executing buffer;

· command [3] is transferred from the input buffer to the waiting buffer; and

· command [4] is transferred to the input buffer as the command block (axis control block data signal).

After the reception of command [4] by the input buffer, the PMC can issue command [5] to the CNC (the axis control block data signal is set).

The timing chart for the command operation is shown below.



(1), (2), (3), (4), (5) : A new block cannot be issued during these intervals (while EBUFg and EBSYg are in different logical states).

- The status of the CNC buffer can be determined by the exclusive OR of the axis control command read signal EBUFg, input from the PMC, and axis control command read completion signal EBSYg, output from the CNC.

EBUFg ‖ EBSYg	Exclusive OR (XOR)	CNC buffer status
0 ‖ 0 1 ‖ 1	0	The previous block has already been read into the CNC buffer. The PMC can issue the next block.
0 ‖ 1 1 ‖ 0	1	The previous block has not yet been read completely. It is just being read or waiting for the CNC buffer to become available. Do not issue the next block, nor reverse the logical state of EBUFg. Reversing the EBUFg state invalidates any block that has been already issued.

(5) Repeat steps (3) and (4) until all the blocks have been issued.

When the final block has been issued, set control axis selection signals EAX1 to EAX4 to “0”. Before setting these signals to “0”, however, check that the blocks stored in the CNC’s input, waiting, and executing buffers have all been executed. Setting the signals to “0” while a block is being executed, or while a block remains in any of these buffers, results in the issue of a P/S alarm. This alarm suspends the current block execution and invalidates the blocks stored in the input and waiting buffers.

To ensure no block is being executed, or that there are no blocks remaining in the input or waiting buffer, check that control axis selection status signal *EAXSL is set to “0”.

For those axes that are always subject to PMC control, such as those controlling turrets, pallets, and ATCs, ensure that the EAX1 to EAX4 signals are always set to “1”. There is no need to set these signals to “0” after issuing commands from the PMC to the CNC. When all command blocks have been executed (there are no blocks remaining to be executed), the CNC automatically stops execution.

(6) When control axis selection signals EAX1 to EAX4 are set from “1” to “0”, control is returned to the CNC.

Signal

Signal list

No.	Symbol	Signal name
1	EAX1 to EAX4	Control axis selection signals
2	EC0g to EC6g	Axis control command signals
3	EIF0g to EIF15g	Axis control feedrate signals
4	EID0g to EID31g	Axis control data signals
5	EBUFg	Axis control command read signal
6	EBSYg	Axis control command read completion signal
7	ECLRg	Reset signal
8	ESTPg	Axis control temporary stop signal
9	ESBKg	Block stop signal
10	EMSBKg	Block stop disable signal
11	EM11g to EM48g	Auxiliary function code signals
12	EMFg	Auxiliary function strobe signal
13	EMF2g	Miscellaneous function 2 strobe signal
14	EMF3g	Miscellaneous function 3 strobe signal
15	EFINg	Auxiliary function completion signal
16	ESOFg	Servo–off signal

No.	Symbol	Signal name
17	EMBUFg	Buffering disable signal
18	*EAXSL	Control axis selection status signal
19	EINPg	In–position signal
20	ECKZg	Following zero checking signal
21	EIALg	Alarm signal
22	EGENg	Axis moving signal
23	EDENg	Auxiliary function executing signal
24	EOTNg	Negative–direction overtravel signal
25	EOTPg	Positive–direction overtravel signal
26	*FV0E to *FV7E	Feedrate override signals
27	OVCE	Override cancellation signal
28	ROV1E, ROV2E	Rapid traverse override signals
29	DRNE	Dry run signal
30	RTE	Manual rapid traverse selection signal
31	EOV0	Override 0% signal
32	ESKIP	Skip signal
33	EADEN1 to EADEN4	Distribution completion signals
34	EABUFg	Buffer full signal
35	EACNT1 to EACNT4	Controlling signals
36	*+ED1 to *+ED4 *–ED1 to *–ED4	External deceleration signal
37	ELCKZg	Accumulated zero check signal
38	TRQMx	Torque control mode signal

Signal Detail

1 Control axis selection signals EAX1 to EAX4

[Classification] Input signal

[Function] When the signal is set to "1", the corresponding axis becomes subject to PMC control.

When the signal is set to "0", PMC control becomes invalid. Changing the setting of the control axis selection signal is possible only when control axis selection status signal *EAXSL is set to "0". Changing the setting when *EAXSL is set to "1" results in the issue of a P/S alarm (No. 139). Alarm signal EIALg is set to "1".

When NCC, bit 5 of parameter No. 8001, is set to "0", a command issued from the CNC is executed while the control axis selection signal is set to "1" and signal *EAXSL is set to "0". When the parameter is set to "1", the same attempt results in the issue of a P/S alarm (No. 139). Note that the command is invalidated when the tool is moving along the axis in manual continuous feed mode.
If the control axis selection signal is set to "1" while the CNC is currently executing a command, a P/S alarm is generated. In manual continuous feed mode, setting this signal to "1" suspends the execution of the command. While *EAXSL is set to "0", the status of alarm signal EIALg does not change to 1 when the control axis selection signal is set to 1 and a P/S alarm (No. 139) is generated. In this case, the axis can be controlled from the PMC, even when the CNC is in the alarm status.

NOTE
After setting control axis selection signals EAX1 to EAX4 to 1, it takes at least 8 msec before the PMC can issue commands to the CNC.

2 Axis control command signals EC0g to EC6g

[Classification] Input signal

[Function] Specifies the following operations through each path.

Axis control command (hexadecimal code)	Operation
00h	Rapid traverse (linear acceleration/deceleration)
	Performs the same operation as G00, used by the CNC.
01h	Cutting feed – feed per minute (exponential acceleration/deceleration or linear acceleration/deceleration after interpolation)
	Performs the same operation as G94 G01, used by the CNC.

Axis control command (hexadecimal code)	Operation
02h	Cutting feed – feed per revolution (exponential acceleration/ deceleration or linear acceleration/deceleration after interpolation)
	Performs the same operation as G95 G01, used by the CNC.
03h	Skip – feed per minute
	Performs the same operation as G31 G01, used by the CNC.
04h	Dwell
	Performs the same operation as G04, used by the CNC.
05h	Reference position return
	Moves the tool in the direction of reference position return specified by ZMIx, bit 5 of parameter No. 1006, in rapid traverse mode, then performs the same operation as manual reference position return, done by the CNC.
06h	Continuous feed (exponential acceleration/deceleration)
	Moves the tool in the specified direction in jog feed mode. Performs the same operation as that of JOG feed, done by the CNC.
07h	1st reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G28 of the CNC.
08h	2nd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P2 of the CNC.
09h	3rd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P3 of the CNC.
0Ah	4th reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P4 of the CNC.
0Bh	External pulse synchronization – main spindle
	Synchronizes with the main spindle.
0Dh	External pulse synchronization – 1st manual handle
	Synchronizes with the 1st manual handle.

Axis control command (hexadecimal code)	Operation
0Eh	External pulse synchronization – 2nd manual handle
	Synchronizes with the second manual handle.
0Fh	External pulse synchronization – 3rd manual handle
	Synchronizes with the 3rd manual handle.
10h	Speed command (linear acceleration/deceleration)
	Performs continuous feed at the specified speed.
11h	Torque control
	Continuous feed under torque control
12h	Auxiliary function
	Performs the same function as the miscellaneous function (M function), used by the CNC.
14h	Miscellaneous function 2
	Similar to the miscellaneous function of the CNC
15h	Miscellaneous function 3
	Similar to the miscellaneous function of the CNC
20h	Machine coordinate system selection
	Performs the same operation as G53, used by the CNC.

Rapid traverse rate

When using the rapid traverse command (EC0g to EC6g: 00h), the feedrate can be specified in either the same parameter as that used by the CNC (No. 1420) or the PMC's axis interface feedrate signals EIF0g to EIF15g. This can be set with RPD, bit 0 of parameter No. 8002.

Reference position return without dogs

The reference position return command (EC0g to EC6g: 05h) enables the following operation: When DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs for each axis, is valid and the tool has not been returned to the reference position since the power was turned on, move each axis in the direction specified by the continuous feed command (EC0g to EC6g: 06h) (position the tool to a point near the reference position) and issue the reference position return command (EC0g to EC6g: 05h). This returns the tool to the reference position (positions the tool to the grid nearest the current position) without using the deceleration signal for reference position return.

Note that, when positioning the tool to a point near the reference position, the tool must be moved in the direction of reference position return at such a speed that the servo position error exceeds the value of parameter No. 1836.

The direction of the grid relative to the proximate position depends on ZMIx, bit 5 of parameter No. 1006.

After the reference position has been established, reference position return can be performed at high speed by issuing the reference position return command (EC0g to EC6g: 05h), irrespective of the reference position return direction specified by ZMIx, bit 5 of parameter No. 1006.

First reference position return for an incomplete reference position return without dogs

When using the 1st reference position return command (EC0g to EC6g: 07h), if DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs on a per–axis basis is valid and the tool has not been returned to the reference position since the power was turned on, issuing the 1st reference position return command (EC0g to EC6g: 07h) results in the issue of a P/S alarm (No. 090).

1st to 4th reference position return

When using the 1st to 4th reference position return commands (EC0g to EC6g: 07h to 0Ah), the feedrate can be specified using RPD, bit 0 of parameter No. 8002, in the same manner as when using the rapid traverse command (EC0g to EC6g: 00h).

Note that, in the case of the 1st reference position return, if the tool has not been manually returned to the reference position after the power was turned on, the feedrate specified by parameter No. 1424 applies.

External pulse synchronization

When using the external pulse synchronization commands (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the tool moves backwards if the external pulse has a negative value. When a manual handle interrupt is executed for the axis to which the external pulse is being applied, the moving distance is the sum of the external pulse and the interrupt pulse.

When a serial spindle is used, synchronization with the main spindle (EC0g to EC6g: 0Bh) is not possible.

Display of remaining distance

When using the continuous feed command (EC0g to EC6g: 06h) and the external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the displayed remaining distance is always “0”.

Speed command

When using the speed command (EC0g to EC6g: 10h), specify the axis to be controlled as a rotation axis in ROTX, bit 0 of parameter No. 1006.

While position control is being executed for the continuous feed command (EC0g to EC6g: 06h), the speed command (EC0g to EC6g: 10h) exerts speed control over the servo motor, thus allowing the speed to be dynamically changed during continuous feed. This makes this command suitable for driving a rotation tool with a servo motor.

A linear acceleration/deceleration time constant can be set for each axis, using parameter No. 8028.

Note that, while jog feed is being executed by the speed command, no coordinate system values are changed. This will result in the loss of the tool position. Therefore, after continuous feed has been completed, always return the tool to the reference position before executing a move command.

Machine coordinate system selection

The machine coordinate system selection command (EC0g to EC6g: 20h) performs absolute positioning to move the tool in rapid traverse to a specified position on the machine coordinate system. This command is used to move the tool to a position specifically defined for the machine, such as the tool change position.

For a rotation axis, short cut rotation can also be specified. When using this command for a T series, cancel the tool offset and the tool nose radius compensation. For a M series, cancel cutter compensation, tool length compensation, and tool offset.

The machine coordinate system must be set before attempting to use this command. After turning on the power, return the tool to the reference position either manually or by using G28. When an absolute position detector is provided, returning the tool to the reference position is not necessary because the tool position will be stored in memory.

Torque control command

When torque control is selected (EC0g to EC6g: 11h), the PMC controlled axis can be subjected to torque control instead of position control. In the torque control mode, the servo motor outputs the torque specified by the NC.

(1) Switching from position control to torque control

a. Setting a torque control axis

Set the axis to be subjected to torque control in the TRQMx bit (bit 7 of parameter No. 2007). Also set the torque control axis in parameter No. 2105 of the torque constant. The motor–specific standard value is automatically set in this parameter when the power is turned on with the DGPR bit (bit 1 of parameter No. 2000) set to 0.

b. Position management in torque control mode

In torque control mode, choosing whether to perform follow–up can be selected by setting the TQF bit (bit 4 of parameter No. 1803).

When follow–up is not performed, an integrated travel value (error count) exceeding the value of parameter No. 1885 causes servo alarm 423 to be issued. When torque control is switched to position control, follow–up is always performed, even if follow–up suppression is selected.

The CNC manages the position even in torque control mode. After torque control is switched to position control, no reference position return is necessary.

c. Traveling direction and speed in torque control mode

In torque control mode, the torque specified by the axis control data signal is output. While there is no target of torque generation or while the output torque falls below the specified torque, the traveling direction is determined by the plus or minus sign added to the torque data, as in position control. When the traveling speed exceeds the specified speed, servo alarm 422 is issued.

d. Timing of switching to torque control

A mode is switched to torque control mode after the position error enters the effective range (in–position state).

(2) Switching from torque control to position control (canceling torque control mode)

Torque control mode is canceled when any of the following conditions is satisfied:

1) The reset signal ECLRg is brought to “1”.

2) A servo alarm is issued.

3) An OT alarm is issued on the torque control axis.

4) An emergency stop occurs.

5) The servo motor is turned off by the servo off signal ESOFg.

a. Timing for canceling torque control

When torque control by the PMC controlled axis is canceled, follow–up may or may not be performed, depending on the setting in the TRF bit (bit 4 of parameter No. 1803). If follow–up is suppressed, the torque control mode signal TRQMx immediately goes “0” when a condition to cancel torque control is detected. Then, position control mode is selected. The execution terminates when the position error enters the effective range. If follow–up is performed, the torque control mode signal TRQMx goes “0” when a condition to cancel torque control is detected. Then, follow–up starts. When the position error falls below or equals the cancel limit value specified in parameter 1886, position control mode is selected. The execution terminates when the position error enters the effective range.

b. Command after torque control mode is canceled

When torque control mode is canceled, normal position control mode is selected. In torque control mode, the position is managed, and the machine coordinates are not displaced. The workpiece coordinates, however, are shifted from the machine coordinates. The shift must be canceled by specifying the command for setting the workpiece coordinate system or similar function.

CAUTION

1 If the torque control axis may be moved in torque control mode, the follow–up parameter TQF (bit 4 of parameter No. 1803) must be set to “1”.

2 If torque control mode is canceled while the torque control axis is moving, the return to position control mode causes a mechanical impact. A faster movement causes a greater impact. To cancel torque control mode, decelerate or stop the movement in advance.

3 When specifying the torque control command after a manual reference position return is completed, set the feed direction selection signal to “0” in advance. Alternatively, select a mode other than manual reference position return mode in advance.

The following table shows the correspondence between the axis control commands and their data:

Command block		
Operation	**Axis control code signal EC0g to EC6g**	**Command data**
Rapid traverse	00h	Total moving distance EID0g to EID31g Rapid traverse rate EIF0g to EIF15g The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.
Cutting feed – feed per minute	01h	Total moving distance EID0g to EID31g Feedrate EIF0g to EIF15g
Skip – feed per minute	03h	
Cutting feed – feed per revolution	02h	Total moving distance EID0g to EID31g Feed per rotation EIF0g to EIF15g
Dwell	04h	Dwell time EID0 to EID31g
Reference position return	05h	None
Jog feed	06h	Feed direction EID31g Jog feedrate EIF0g to EIF15g
1st reference position return	07h	Rapid traverse rate EIF0g to EIF15g The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.
2nd reference position return	08h	
3rd reference position return	09h	
4th reference position return	0Ah	
External pulse synchronization – main spindle	0Bh	Pulse weight EIF0g to EIF15g
External pulse synchronization – manual handle	0Dh	
	0Eh	
	0Fh <For M series only>	
Speed command	10h	Continuous feed EIF0g to EIF15g

Operation	Axis control code signal EC0g to EC6g	Command data
Torque control	11h	Maximum feedrate, EIF0g to EIF15g Torque data, EID0g to EID31g
Auxiliary function	12h	Auxiliary function code EID0g to EID15g
Miscellaneous function 2	14h	Miscellaneous function code, EID0g to EID15g
Miscellaneous function 3	15h	Miscellaneous function code, EID0g to EID15g
Machine coordinate system setting	20h	Machine coordinate system setting (absolute value) EID0g to EID31g Rapid traverse rate EIF0g to EIF15g The rapid traverse rate setting is effective when bit 0 (RPD) of parameter No. 8002 is set to "1".

3 Axis control feedrate signals EIF0g to EIF15g

[Classification] Input signal

[Function]
(1) Rapid traverse (EC0g to EC6g: 00h)
(2) 1st reference position return (EC0g to EC6g: 07h)
(3) 2nd reference position return (EC0g to EC6g: 08h)
(4) 3rd reference position return (EC0g to EC6g: 09h)
(5) 4th reference position return (EC0g to EC6g: 0Ah)
(6) Machine coordinate system selection (EC0g to EC6g: 20h)

For these commands, signals EIF0g to EIF15g are used to specify the rapid traverse rate, in binary format, when bit 0 (RPD) of parameter No. 8002 is set to "1". For 1st reference position return, however, the rapid traverse rate specified with parameter No. 1424 is used if manual reference position return has not been performed after the power was first turned on.

[Unit of data]

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1		mm/min
	Inch machine	0.1		inch/min
Rotation axis		1		deg/min

[Valid data range] Specify data within the range given in the following table.

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	30 to 15000	30 to 12000	mm/min
	Inch machine	30 to 6000	30 to 4800	inch/min
Rotation axis		30 to 15000	30 to 12000	deg/min

(7) Cutting feed – feed per minute (EC0g to EC6g: 01h)

(8) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, the signals are used to specify, in binary format, the feedrate along an axis. The specified feedrate can be magnified by ten by the setting of bit 3 (F10) of parameter No. 8002.

[Unit of data] When bit 3 (F10) of parameter No. 8002 is set to 0

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1	0.1	mm/min
	Inch machine	0.01	0.001	inch/min
Rotation axis		1	0.1	deg/min

When bit 3 (F10) of parameter No. 8002 is set to 1

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	10	1	mm/min
	Inch machine	0.1	0.01	inch/min
Rotation axis		10	1	deg/min

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1 to 100000	0.1 to 12000.0	mm/min
	Inch machine	0.01 to 4000.00	0.001 to 480.000	inch/min
Rotation axis		1 to 100000	0.1 to 12000.0	deg/min

CAUTION
When “0” is specified, the CNC continues to perform buffering without moving the tool. In such a case, release the buffering by issuing reset signal ECLRg.
Cutting feedrate clamp is disabled.

(9) Cutting feed – feed per rotation (EC0g to EC6g: 02h)

For this command, the signals are used to specify the amount by which the tool is moved for every rotation of the spindle.

<For T series>

[Unit of data] The data increment depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.0001	0.000001	0.0001
0	0			
0	1	0.001	0.00001	0.001
1	0	0.01	0.0001	0.01

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.0001 to 500.0000		mm/rev
	Inch input	0.000001 to 9.999999		inch/rev
Rotation axis		0.0001 to 500.0000		deg/rev

<For M series>

[Unit of data] The data unit depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.01	0.0001	0.01
0	0			
0	1	0.1	0.001	0.1
1	0	1	0.01	1

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.01 to 500.00		mm/rev
	Inch input	0.0001 to 9.9999		inch/rev
Rotation axis		0.01 to 500.00		deg/rev

WARNING

1. The value of parameter No. 8022 is used as the upper limit for clamping the feedrate.
2. Override for the feedrate is effective. Dry run is invalid.

> **CAUTION**
> The specified feedrate can be magnified by 1, 10, or 100 by setting bits 6 (FR1) and 7 (FR2) of parameter No. 8002 accordingly.

(10) External pulse synchronization – main spindle
(EC0g to EC6g: 0Bh)

(11) External pulse synchronization – first manual handle
(EC0g to EC6g: 0Dh)

(12) External pulse synchronization – second manual handle
(EC0g to EC6g: 0Eh)

(13) External pulse synchronization – third manual handle
(EC0g to EC6g: 0Fh)

For these commands, the signals are used to specify the weight of the external pulses. A weight range of ±1/256 to ±127 can be set by using signals EIF0g to EIF7g for the figures after the decimal point. When a negative weight is specified, the tool is moved in the reverse direction. When a new pulse weight is specified while the tool is moving in synchronization with external pulses, inverting signal EBUFg causes the tool to move with the new pulse weight.

As commands for (10) to (13) are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

> **CAUTION**
> The pulse weight is clamped according to the value set for parameter No. 1424 (parameter for the manual rapid traverse rate for each axis).

> **NOTE**
> When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

(14) Continuous feed (EC0g to EC6g: 06h)

Set the feedrate as the same as for cutting feed – feed per minute (EC0g to EC6g: 01h). The feedrate can be changed during continuous feed.

Specify the feedrate with signals EIF0g to EIF15g, and invert the axis control command read signal EBUFg during continuous feed, then the tool moves at the new feedrate.

As commands for jog feed are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

The specified feedrate can be magnified by 10 by setting bit 3 (F10) of parameter No. 8002.

- Maximum feedrate (with override of 254%)

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magnified by 1	166458 mm/min	1664.58 inch/min	16645 mm/min	166.45 inch/min
Magnified by 10	1664589 mm/min	16645.89 inch/min	166458 mm/min	1664.58 inch/min
Magnified by 200 (Note 1)	1966050 mm/min	1966605.00 inch/min	196605 mm/min	19660.50 inch/min

CAUTION
The maximum feedrate depends on whether override is applied or canceled. The following table lists the maximum feedrate when override is canceled.

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magnified by 1	65535 mm/min	655.35 inch/min	6553 mm/min	65.53 inch/min
Magnified by 10	655350 mm/min	6553.50 inch/min	65535 mm/min	655.35 inch/min
Magnified by 200	13107000 mm/min	131070.00 inch/min	1310700 mm/min	13107.00 inch/min

NOTE
The actual speed may not be displayed correctly, depending on the feedrate.

(15) Speed command (EC0g to EC6g: 10h)

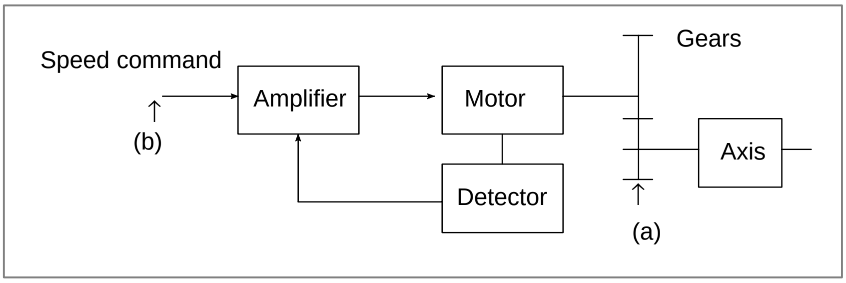
For this command, the signals are used to specify, in binary format, the servo motor speed.

Specify a positive value for rotation in the forward direction. Specify a negative value (twos complement) for rotation in the reverse direction.

When a new servo motor speed is specified, inverting the axis control command read signal EBUFg accelerates or decelerates the servo motor until it attains the new speed.

Data range	**Unit**
–32768 to +32767	min^{-1}

NOTE
The servo motor speed may contain a slight error, as follows:



(a) The speed command for PMC axis control requires the specification of the servo motor speed, not the feedrate along an axis. To specify a feedrate along the axis when gears are used to link the servo motor and axis, the feedrate must be converted to a rotation speed of the servo motor speed. As the motor speed must be specified with an integer, the converted speed is subject to a round–off error.

(b) The minimum increment for specifying the motor speed is calculated by the following formula and rounded to the nearest integer:

$$\mathrm{Fmin} = \frac{P \times 2}{15} \times \frac{1}{1000}$$

Fmin : Minimum increment for the motor speed

P : Number of pulses per rotation of the detector for velocity feedback

Specify the speed command using the value calculated by the following formula:

$$F = \frac{N \times P \times 2}{15} \times \frac{1}{1000}$$

F : Speed command value (integer)

N : Servo motor speed (min^{-1})

P : Number of detector pulses issued per rotation for velocity feedback

NOTE

In speed command mode, the speed after acceleration/deceleration is specified to the servo control unit. The loop gain for position control is invalid.

(16) When a torque control command (EC0g to EC6g: 11h) is specified

Specify the maximum speed in torque control mode, in min^{-1}.

A servo alarm (No. 422) is issued if there is no object for which a torque is to be generated or if the feedrate exceeds the specified value while torque control is being applied.

The maximum feedrate in torque control mode can be changed in torque control mode by setting the new feedrate data in the signal then inverting the logic of axis control command read signal EBUFg.

Valid data range	Units
1 to 32767	min^{-1}

NOTE
The data units will be cm/min when a linear motor is being used.

4 Axis control data signals EID0g to EID31g

[Classification] Input signal

[Function]

[Unit of data]

	IS–B	IS–C	Unit
Metric input Degree input	0.001	0.0001	mm deg
Inch input	0.0001	0.00001	inch

[Valid data range]

(1) Rapid traverse (EC0g to EC6g: 00h)
(2) Cutting feed – feed per minute (EC0g to EC6g: 01h)
(3) Cutting feed – feed per rotation (EC0g to EC6g: 02h)
(4) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, signals EID0g to EID31g are used to specify, in binary format, the incremental moving distance, according to the input increment used for the axis.

	IS–B	IS–C	Unit
Metric input Degree input	±99999.999	±9999.9999	mm deg
Inch input	±9999.9999	±999.99999	inch

When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

(5) Dwell (EC0g to EC6g: 04h)

For this command, the signals are used to specify, in binary format, the dwell time.

Data range	Unit
1 to 9999999	ms

When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

NOTE

When the increment system IS–C is used, the least input increment for the dwell time can be set to 0.1 ms, according to the setting of bit 1 (DWE) of parameter No. 8002.

(6) Continuous feed (EC0g to EC6g: 06h)

For this command, signal EID31g is used to specify the direction of continuous feed, as follows:

0: Positive direction
1: Negative direction

Signals EID0g to EID30g are undefined.

(7) Auxiliary functions (EC0g to EC6g: 12h)
Auxiliary functions 2 (EC0g to EC6g: 14h)
Auxiliary functions 3 (EC0g to EC6g: 15h)

For this command, the signals are used to specify, in binary format, an auxiliary function code to be sent to the PMC. The auxiliary function code can be specified using either one or two bytes, depending on the setting of bit 6 (AUX) of parameter No. 8001, in signals EID0g to EID15g.

(8) Machine coordinate system selection (EC0g to EC6g: 20h)

For this command, the signals are used to specify, in binary format, an absolute machine coordinate, according to the increment system used by the axis.

Example: For absolute value “10000”

Input increment	inch	1.0000	mm	10.000
Output increment	mm	25.400	inch	0.3937

The direction of rotation can be specified about a rotation axis with a parameter. To enable the roll–over function, set bit 0 (ROAx) of parameter No. 1008 to 1. Then, select whether the tool is to be rotated in the direction corresponding to the sign of the specified value, or in whichever direction minimizes the distance to the end point, using bit 1 (RABx) of parameter No. 1008. The moving distance per rotation must be set in parameter No. 1260.

(9) When a torque control command (EC0g to EC6g: 11h) is specified

Specify the torque data.

Specify a positive value when the torque is to be applied in the positive direction. Specify a negative value when the torque is to be applied in the negative direction.

The specified torque data can be changed in torque control mode by setting the new data in the signal then inverting the logic of axis control command read signal EBUFg.

Valid data range	Units
–99999999 to +99999999	0.00001 Nm

> **NOTE**
> The data units will be 0.001 N when a linear motor is being used.

5 Axis control command read signal EBUFg

[Classification] Input signal

[Function] Directs the CNC to read a block of command data for PMC axis control. See “Basic procedure” for details of the operation performed when this signal is set from “0” to “1” or from “1” to “0”.

6 Axis control command read completion signal EBSYg

[Classification] Output signal

[Function] Notifies the system that the CNC has read a block of command data for PMC axis control and has stored the block in the input buffer. See “Basic procedure” for details of the output conditions and the procedure.

7 Reset signal ECLRg

[Classification] Input signal

[Function] Resets the corresponding PMC–controlled axis.
When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation.

Simultaneously, all buffered commands are canceled. Any control command is ignored while this signal is set to “1”.

The continuous feed command (EC0g to EC6g: 06h) and external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh) can be terminated by setting reset signal ECLRg to “1”. When these commands are terminated, the servo motor decelerates and stops, the axis moving signal EGENg is set to “0”, and the control axis selection status signal *EAXSL is set to “0”. Confirm that the control axis selection status signal *EAXSL has been set to “0” before issuing the next command. Do not set reset signal ECLRg to “0” until the control axis selection status signal *EAXSL has been set to “0”.

The speed command (EC0g to EC6g: 10h) can also be terminated by setting the reset signal ECLRg to “1”. When this command is terminated, the servo motor decelerates and stops, and the axis moving signal EGENg is set to “0”. Confirm that the axis moving signal EGENg has been set to “0” before issuing the next command. Do not attempt to set the reset signal ECLRg to “0” until the axis moving signal EGENg has been set to “0”.

8 Axis control temporary stop signal ESTPg

[Classification] Input signal

[Function] When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is in dwell: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation when auxiliary function completion signal EFINg is input.

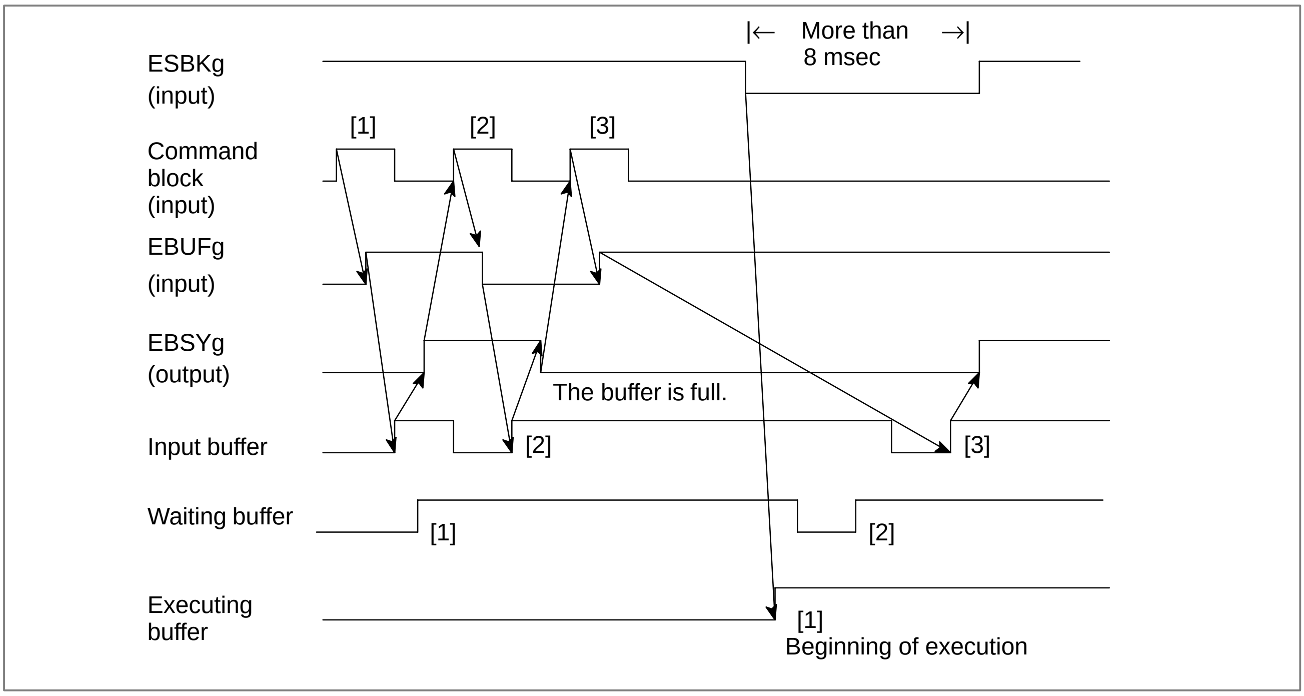
The stopped operation can be restarted by setting this signal to “0”.

9 Block stop signal ESBKg
10 Block stop disable signal EMSBKg

[Classification] Input signal

[Function] When block stop signal ESBKg is set to “1” during the execution of a command issued from the PMC, axis control is stopped after the block being executed is completed. When this signal is set to “0”, the buffered command is executed. Block stop signal ESBKg is disabled when block stop disable signal EMSBKg is set to “1” for the block.

The timing chart for the command operation is shown below.



11 Auxiliary function code signals EM11g to EM48g

[Classification] Output signal

12 Auxiliary function strobe signal EMFg

[Classification] Output signal

13 Miscellaneous function 2 strobe signal EMF2g

[Classification] Output signal

14 Miscellaneous function 3 strobe signal EMF3g

[Classification] Output signal

15 Auxiliary function completion signal EFINg

[Classification] Input signal

[Function] When an auxiliary function command (EC0g to EC6g: 12h) is issued by the PMC, the auxiliary function code is specified in a byte (using signals EID0g to EID7g) or two bytes (using signals EID0g to EID15g), depending on the setting of bit 6 (AUX) of parameter No. 8001.

The CNC sends the auxiliary function code specified in signals EID0g to EID7g and EID8g to EID15g to auxiliary function code signals EM11g to EM28g and EM31g to EM48g and awaits the auxiliary function completion signal EFINg. When the auxiliary function completion signal EFINg is returned, the CNC starts executing the next block.
The timings for sending the auxiliary function code signals and auxiliary function strobe signal, as well as for receiving the auxiliary function completion signal, are the same as those for the miscellaneous functions (M functions) under the control of the CNC. See “Auxiliary function executing signal” for details.

16 Servo–off signal ESOFg

[Classification] Input signal

[Function] When this signal is set to “1”, the servo motor for the corresponding PMC–controlled axis is turned off (servo–off state).

When this signal is set to “0”, the servo motor is turned on.

When a torque control command (EC0g to EC6g: 11h) is specified, entering the servo–off state cancels torque control mode, but the torque control state remains set. In such a case, set reset signal ECLRg to “1”.

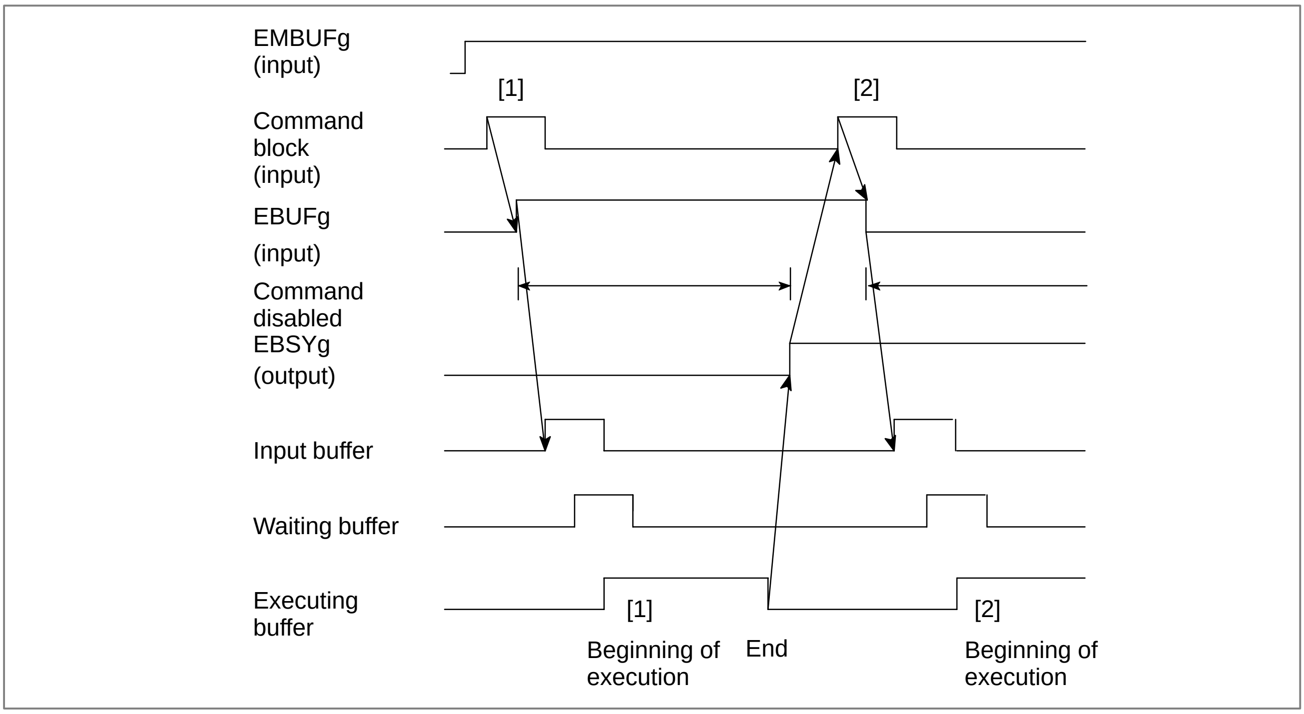
17 Buffering disable signal EMBUFg

[Classification] Input signal

[Function] When this signal is set to “1”, commands from the PMC are not read while the executing, waiting, or input buffer contains a block. If this signal is set to “1” when any of these buffers contain a block, that block is executed but subsequent commands are read only when the buffers are all empty.

To discriminate the buffering disabled condition, the CNC outputs the axis control command read completion signal (EBSYg) only when a command is read when all buffers are empty.

The timing chart for the command operation is shown below.



Buffering is disabled, regardless of the buffering disable signal EMBUFg, for the following commands:

(1) Skip–feed per minute (EC0g to EC6g: 03h)

(2) Reference position return (EC0g to EC6g: 05h)

(3) 1st reference position return (EC0g to EC6g: 07h)

(4) 2nd reference position return (EC0g to EC6g: 08h)

(5) 3rd reference position return (EC0g to EC6g: 09h)

(6) 4th reference position return (EC0g to EC6g: 0Ah)

(7) Machine coordinate system selection (EC0g to EC6g: 20h)

The following commands, for which the reset signal ECLRg is used for termination, operate as though buffering had been disabled. That is, the subsequently specified block is not executed but canceled:

(1) Continuous feed (EC0g to EC6g: 06h)

(2) External pulse synchronization – main spindle (EC0g to EC6g: 0Bh)

(3) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)

(4) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(5) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

(6) Speed command (EC0g to EC6g: 10h)

18 Control axis selection status signal *EAXSL

[Classification] Output signal

[Function] When this signal is set to "0", control axis selection signals EAX1 to EAX4 can be changed.

This signal is set to "1" in the following cases:

(1) When the tool is moving along a PMC–controlled axis

(2) When a block is being read into a buffer

(3) When the servo–off signal ESOFg is set to "1"

When this signal is set to "1", control axis selection signals EAX1 to EAX4 cannot be changed. Any attempt to change these signals results in the output of P/S alarm No. 139.

If an attempt to change signals EAX1 to EAX4 is made when servo–off signal ESOFg is "1", P/S alarm No. 139 occurs and cannot be released simply by setting reset signal ECLRg to "1". In such a case, restore signals EAX1 to EAX4 or set servo–off signal ESOFg to "0" before setting reset signal ECLRg to "1".

When a command is issued for any of the four paths with PMC axis control, signal *EAXSL is set to "1" to disable axis selection. Thus, changing signals EAX1 to EAX4 results in the output of P/S alarm No. 139. For paths for which commands are not issued, however, axis selection is enabled if bit 5 (DSL) of parameter No. 8004 is set accordingly.

19 In–position signal EINPg

[Classification] Output signal

[Function] This signal is set to "1" when the corresponding PMC–controlled axis is in the in–position state.

When the tool is decelerated, in–position check is performed to disable the next command until the tool enters the in–position area. The in–position check, however, can be skipped using bit 6 (NCI) of parameter No. 8004 to reduce the cycle time.

NOTE
When the axis is fed at a very low speed, the in –position signal might turn to "1".

20 Following zero checking signal ECKZg

[Classification] Output signal

[Function] This signal is set to “1” when following zero check or in–position check is being performed for the corresponding PMC–controlled axis.

21 Alarm signal EIALg

[Classification] Output signal

[Function] This signal is set to “1” when a servo alarm, overtravel alarm, or P/S alarm No. 130 or 139 occurs for the corresponding PMC–controlled axis. This signal is set to “0” when the reset signal ECLRg is set to “1” after the alarm is released, as described below.

- Servo alarm

  Eliminate the cause of the alarm, then reset the CNC.
- Overtravel alarm

  Move the tool into the area within the stored stroke limit, then reset the CNC.

  The following commands can be used to move the tool into the area within the stored stroke limit during an overtravel alarm:

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) Cutting feed–feed per minute (EC0g to EC6g: 01h)

(3) Cutting feed–feed per rotation (EC0g to EC6g: 02h)

(4) Continuous feed (EC0g to EC6g: 06h)

(5) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)

(6) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(7) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

- P/S alarm (130 or 139)

  Reset the CNC. See “Alarms and messages” for details.

Reset signal ECLRg cannot be used to reset the CNC in the above cases. Use the reset button on the setting panel, external reset signal ERS, or emergency stop signal *ESP.

22 Axis moving signal EGENg

[Classification] Output signal

[Function] This signal is set to “1” when the tool is moving along the corresponding PMC–controlled axis according to commands such as rapid traverse (EC0g to EC6g: 00h) and cutting feed (EC0g to EC6g: 01h).

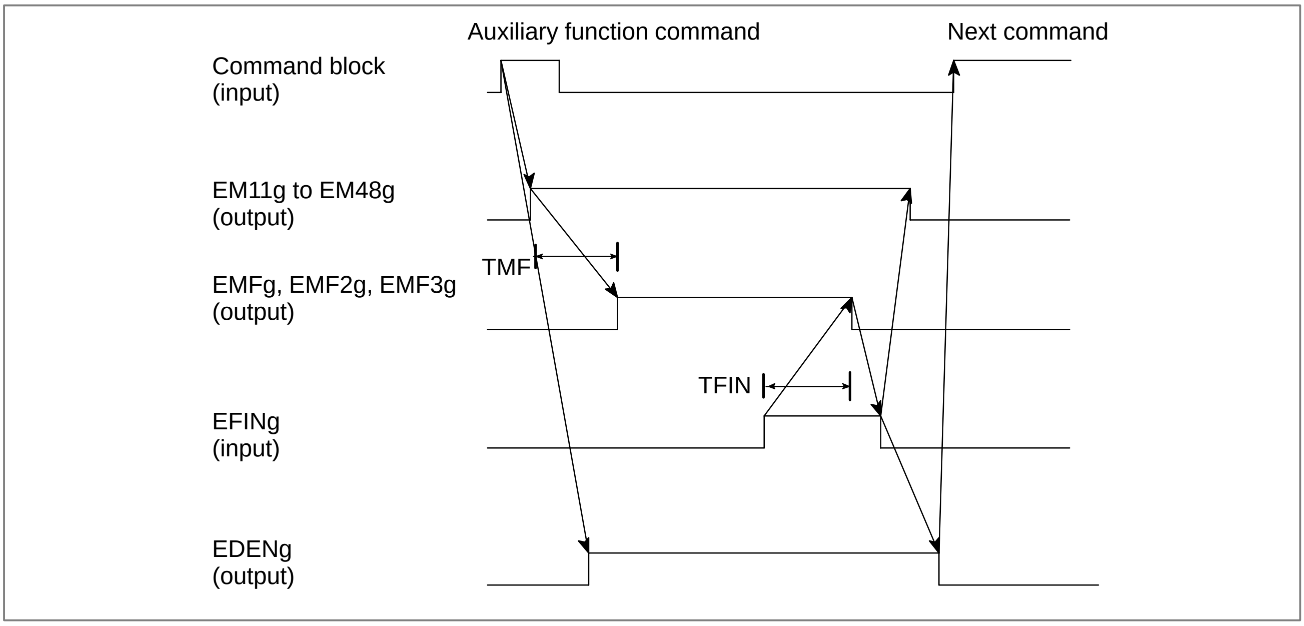
> **NOTE**
> This signal is set to “0” when distribution for the axis is completed (the signal is set to “0” during deceleration).

23 Auxiliary function executing signal EDENg

[Classification] Output signal

[Function] When an auxiliary function (EC0g to EC6g: 12h) is specified by the PMC, this signal is set to “1” during the period from when auxiliary function codes EID0g to EID15g are sent to auxiliary function code signals EM11g to EM48g until the auxiliary function completion signal EFINg is returned.

The timing chart for the command operation is shown below.



TMF and TFIN are set with parameters No. 3010 and 3011.

24 Negative–direction overtravel signal EOTNg
25 Positive–direction overtravel signal EOTPg

[Classification] Output signal

[Function] These signals are set to “1” when an overtravel alarm is detected. When the stroke limit in the negative direction is exceeded, signal EOTNg is set to “1”. When the stroke limit in the positive direction is exceeded, signal EOTPg is set to “1”. Simultaneously, alarm signal EIALg is set to “1”.

These signals are set to “0” when the overtravel alarm is released and reset signal ECLRg is set to “1”. See “Alarm signal EIALg” for details of how to release an overtravel alarm.

26 Feedrate override signals *FV0E to *FV7E

[Classification] Input signal

[Function] Like the CNC’s feedrate override signals *FV0 to *FV7, these signals can be used to select the override for the cutting feedrate, in steps of 1% from 0 to 254%, independently of the CNC using bit 2 (OVE) of parameter No. 8001.

These signals form an eight–bit binary code and correspond to the override value as follows:

$$\text{Override value} = \sum_{i=0}^{7} |2^{i} \times Vi|\%$$

Vi = 0 when signal *FViE is 1
Vi = 1 when signal *FViE is 0

That is, each signal has the following significance:

*FV7E = 128%, *FV3E = 8% ,
*FV6E = 64%, *FV2E = 4% ,
*FV5E = 32%, *FV1E = 2% ,
*FV4E = 16%, *FV0E = 1%

When all signals are set to “0”, the override is regarded as being 0%, as well as when all signals are “1”.

27 Override cancellation signal OVCE

[Classification] Input signal

[Function] When override is enabled, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001, setting this signal to “1” fixes the cutting feed override to 100%. This signal does not affect the rapid traverse override.

28 Rapid traverse override signals ROV1E and ROV2E

[Classification] Input signal

[Function] These signals can be used to select the override for the rapid traverse rate, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001.

Rapid traverse override signals		Override value
ROV2E	ROV1E	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the minimum feedrate specified with parameter No. 1421.

29 Dry run signal DRNE
30 Manual rapid traverse selection signal RTE

[Classification] Input signal

[Function] These signals can be used to perform dry run or manual rapid traverse, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001. When the dry run signal DRNE is set to “1”, the specified rapid traverse rate and cutting feedrate are ignored and the tool moves at the dry run speed (set in parameter No. 1410) multiplied by the specified override. Bit 3 (RDE) of parameter No. 8001 can be used to specify whether to enable or disable dry run for rapid traverse.

When the manual rapid traverse selection signal RTE is set to “1” during dry run, the tool moves at the rapid traverse rate for rapid traverse and at the maximum jog feedrate for cutting feed. When the signal is set to “0”, the tool moves at the jog feedrate. When the dry run signal DRNE is set to “0”, the specified rapid traverse rate or cutting feedrate is restored.

Feedrate of Dry run

Manual rapid traverse select signal	Command from PMC	
	Rapid traverse	Feed
1	Rapid traverse rate	Maximum cutting feedrate
0	Dry run feed rate × JV(*)	Dry run feed rate × JV

* Can also be set to the rapid traverse rate with bit 3 (RDE) of parameter No. 8001.

31 Override 0% signal EOV0

[Classification] Output signal

[Function] This signal is set to "1" when the feedrate override is 0%.

32 Skip signal ESKIP

[Classification] Input signal

[Function] When this signal is set to "1" during execution of the skip cutting command, the block being executed is immediately stopped and the next block is executed. Bit 7 (SKE) of parameter No. 8001 can be used to select whether to use signal SKIP, which is the common skip signal for the PMC and CNC, or PMC–specific skip signal ESKIP.

33 Distribution completion signals EADEN1 to EADEN4

[Classification] Output signal

[Function] These signals are set to "0" when the tool is moving with a command from the PMC. The signals are set to "1" when the tool is not moving, except when it is stopped by an axis control temporary stop signal ESTPg during the execution of a move command.

34 Buffer full signal EABUFg

[Classification] Output signal

[Function] This signal is set to "1" when the input buffer contains a command block.

35 Controlling signals EACNT1 to EACNT4

[Classification] Output signal

[Function] When the control axis selection status signal *EAXSL is set to "1", signal EACNTn corresponding to the axis being controlled is set to "1".

36 External deceleration signals *+ED1 to *+ED4, *–ED1 to *–ED4

[Classification] Input signal

[Function] These signals are also used by the CNC. The signals are provided for each direction of the individual controlled axes. The plus or minus sign in the signal name indicates the direction, and the number at the end corresponds to the axis number of the controlled axis.

± EDx

1 … Feed along the first axis is decelerated.
:
4 …Feed along the fourth axis is decelerated.

\+ … Feed in the positive direction is decelerated.
:
– … Feed in the negative direction is decelerated.

When any of the following axis control command is specified while the EDC bit (bit 0 of parameter 8005) is held at 1, the external deceleration function becomes effective:

(1) Rapid traverse	(EC0g to EC6g: 00h)
(2) Cutting feed per minute	(EC0g to EC6g: 01h)
(3) Reference position return	(EC0g to EC6g: 05h)
(4) Continuous feed	(EC0g to EC6g: 06h)
(5) First reference position return	(EC0g to EC6g: 07h)
(6) Second reference position return	(EC0g to EC6g: 08h)
(7) Third reference position return	(EC0g to EC6g: 09h)
(8) Fourth reference position return	(EC0g to EC6g: 0Ah)
(9) Machine coordinate system selection	(EC0g to EC6g: 20h)

While the signal is held at “0”, the feedrate in the corresponding direction of axial movement can be reduced to the speed specified in parameter 1427 (external deceleration speed).

When the feedrate is lower than the external deceleration speed, the specified feedrate is valid.

The feedrate on other axes for which the signal is not held at “0” is not affected.

37 Accumulated zero check signal ELCKZg

[Classification] Input signal

[Function] Setting this signal to 1 causes an accumulated zero check between blocks to be made at a subsequent cutting feed command.

(1) Cutting feed per minute (EC0g to EC6g: 01h)

(2) Cutting feed per rotation (EC0g to EC6g: 02h)

This enables the chopping function.

38 Torque control mode signal TRQMx

[Classification] Output signal

[Function] This signal indicates which axis is placed in torque control mode under PMC axis control.

Signal address

MT→CNC

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP	ESKIP						

PMC→CNC

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
G118					*+ED4	*+ED3	*+ED2	*+ED1
G120					*–ED4	*–ED3	*–ED2	*–ED1
G136					EAX4	EAX3	EAX2	EAX1
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E

For group A

	#7	#6	#5	#4	#3	#2	#1	#0
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA	ELCKZA	EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

For group B

	#7	#6	#5	#4	#3	#2	#1	#0
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB	ELCKZB	EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

For group C

	#7	#6	#5	#4	#3	#2	#1	#0
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC	ELCKZC	EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

For group D

	#7	#6	#5	#4	#3	#2	#1	#0
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD	ELCKZD	EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

CNC→PMC

ADDRESS

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F112					EADEN4	EADEN3	EADEN2	EADEN1
F129	*EAXSL		EOV0					

For group A

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131					EMF3A	EMF2A	EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A

For group B

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134					EMF3B	EMF2B	EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B

For group C

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137					EMF3C	EMF2C	EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C

For group D

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140					EMF3D	EMF2D	EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D

Group	ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
Group A	F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
Group B	F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
Group C	F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
Group D	F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D

	#7	#6	#5	#4	#3	#2	#1	#0
F182					EACNT4	EACNT3	EACNT2	EACNT1
F190					TRQM4	TRQM3	TRQM2	TRQM1

Parameter

1427	External deceleration speed of each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

	#7	#6	#5	#4	#3	#2	#1	#0
1803				TQF				

[Data type] Bit

TQF In torque control mode,

0 : Follow–up is not performed.
1 : Follow–up is performed.

1885	Permissible value of integrated travel distance under torque control

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the maximum permissible value for the integrated travel distance in torque control mode. If the integrated travel distance exceeds the value set in this parameter, servo alarm No. 423 is issued.

NOTE
This parameter is valid when TQF (bit 4 of parameter No. 1803) is set to 0.

1886	Torque control cancel limit

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the cancel limit value used to cancel torque control mode.

When the position error falls below or is equal to the value set in this parameter, torque control mode is canceled and position control becomes effective.

> **NOTE**
> This parameter is valid when TQF (bit 4 of parameter 1803) is held 0.

	#7	#6	#5	#4	#3	#2	#1	#0
2000							DGPRx	

[Data type] Bit axis

DGPRx At power–ON, the torque constant (parameter No. 2105):

0 : Is automatically set to the standard value specific to the motor.
1 : Is not automatically set to the standard value specific to the motor.

> **NOTE**
> This parameter is intended to initialize the servo parameters. It causes parameters other than those for torque constants to be set automatically. Be careful when using this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
2007	TRQx							

[Data type] Bit axis

TRQx Torque control:

0 : Is not performed for the axis.
1 : Is performed for the axis.

2105	Torque constant

[Data type] Word axis

[Unit of data] 0.00001 Nm/(torque command)

[Valid data range] 1 to 32767

A torque constant is set for each motor torque characteristic.

> **NOTE**
> When a linear motor is used, the units of data are 0.001 N/(torque command).

	#7	#6	#5	#4	#3	#2	#1	#0
3105							PCF	

[Data type] Bit

PCF Movement along PMC–controlled axes is included in the actual speed display

0 : Is included
1 : Is not included

> **NOTE**
> This parameter is valid when the same axis is controlled alternately by the CNC and PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8001	SKE	AUX	NCC		RDE	OVE		MLE

[Data type] Bit

MLE Machine lock signal MLK is valid for PMC–controlled axes

0 : Is valid
1 : Is invalid

> **NOTE**
> Each–axis machine lock signals MLK1 to MLK4 are always valid, regardless of the setting of this parameter.

OVE Signals related to dry run and override used in PMC axis control

0: Same signals as those used for the CNC
(1) Feedrate override signals *FV0 to *FV7
(2) Override cancellation signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT

1: Signals specific to the PMC
(1) Feedrate override signals *FV0E to *FV7E
(2) Override cancellation signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

RDE Dry run is valid for rapid traverse in PMC axis control

0 : Is invalid
1 : Is valid

NCC When a travel command is issued for a PMC–controlled axis (selected by a controlled–axis selection signal) according to the program:

0 : P/S alarm 139 is issued while the PMC controls the axis with an axis control command. While the PMC does not control the axis, a CNC command is enabled.
1 : P/S alarm 139 is issued unconditionally.

AUX The number of bytes for the code of an auxiliary function (12H) command to be output is

0 : 1 byte type (0 to 255)
1 : 2 bytes type (0 to 65535)

SKE Skip signal during axis control by the PMC

0 : Uses the same signal SKIP (X004#7) as CNC.
1 : Uses dedicated axis control signal ESKIP (X004#6) used by the PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10	SUE	DWE	RPD

[Data type] Bit

RPD Rapid traverse rate for PMC–controlled axes

0 : Feedrate specified with parameter No. 1420
1 : Feedrate specified with the feedrate data in an axis control command

DWE Minimum time which can be specified in a dwell command in PMC axis control when the increment system is IS–C

0 : 1 ms
1 : 0.1 ms

SUE Whether acceleration/deceleration is performed for an axis that is synchronized with external pulses, for external pulse synchronization commands in PMC axis control

0 : Is performed (exponential acceleration/deceleration)
1 : Is not performed

F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control

F10	Metric input	Inch input
0	1 mm/min	0.01 inch/min
1	10 mm/min	0.1 inch/min

PF1, PF2 Set the the feedrate unit of feed per minute in PMC axis control

PF2	PF1	Feedrate unit
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

FR1, FR2 Set the feedrate unit for feed per rotation for an axis controlled by the PMC.

FR2	FR1	Metric input	Inch input
0	0	0.0001 mm/rev	0.000001 inch/rev
1	1		
0	1	0.001 mm/rev	0.00001 inch/rev
1	0	0.01 mm/rev	0.0001 inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

NOTE
When this parameter is set, the power must be turned off then back on again to make the setting effective.

[Data type] Bit type

PIM If a linear axis is controlled solely by the PMC, the commands for that axis are:

0 : Affected by inch/metric input.

1 : Not affected by inch/metric input.

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL	G8R	G8C	JFM	NMT	CMV
		NCI	DSL	G8R	G8C	JFM	NMT	CMV

[Data type] Bit type

CMV If the PMC issues a command for an axis after the completion of a movement along that axis according to a command issued by the CNC, but before receiving the signal indicating that the miscellaneous function is completed in the same block,

0 : P/S alarm No. 130 occurs.

1 : The axis is handled as a PMC axis and the specified movement is executed.

NMT If the CNC issues a command that does not result in any movement along a PMC–controlled axis while another command, specified for the axis, is being processed,

0 : P/S alarm No. 130 occurs.

1 : No alarm occurs.

JFM Specifies the units used to specify the feedrate for continuous feed (06H) for a PMC–controlled axis.

Increment system	JFM	Metric input	Inch input	Rotation axis
IS–B	0	1 mm/min	0.01 inch/min	0.00023 min^{-1}
	1	200 mm/min	2.00 inch/min	0.046 min^{-1}
IS–C	0	0.1 mm/min	0.001 inch/min	0.000023 min^{-1}
	1	20 mm/min	0.200 inch/min	0.0046 min^{-1}

G8C If a cutting feed command is specified for a PMC–controlled axis, look–ahead control is:

0 : Disabled.

1 : Enabled.

> **NOTE**
> The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

G8R If a rapid traverse command is specified for a PMC–controlled axis, look–ahead control is:

0 : Disabled.
1 : Enabled.

> **NOTE**
> The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

DSL If an axis exchange is attempted while the selection of a PMC–controlled axis is inhibited,

0 : The attempt fails and a P/S alarm No. 139 occurs.
1 : The axis exchange is executed for the axes that belong to an unspecified path.

NCI During deceleration along an axis controlled by the PMC, the in–position check is:

0 : Performed.
1 : Not performed.

NDI When diameter programming is selected for a PMC controlled axis, under PMC axis control:

0 : Radius programming is used to specify the travel distance and feedrate.
1 : Diameter programming is used to specify the travel distance and feedrate.

> **NOTE**
> While the CDI bit (bit 1 of parameter No. 8005) is held to 0, the NDI bit is valid for an axis of diameter programming (the DIAx bit (bit 3 of parameter No. 1006) is set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8005	MFD						CDI	EDC

[Data type] Bit

EDC Under PMC axis control, the external deceleration signal is:

0 : Ineffective.
1 : Effective.

CDI When diameter programming is selected for a PMC controlled axis, under PMC axis control:

0 : Radius programming is used to specify the travel distance and feedrate.
1 : Diameter programming is used to specify the travel distance, and radius programming is used to specify the feedrate.

> **NOTE**
> 1 This parameter is valid when the DIAx bit (bit 3 of parameter 1006) is held to 1.
> 2 When the CDI bit is set to 1, the NDI bit (bit 7 of parameter 8004) is invalid.

MFD : The miscellaneous function individual output of the PMC axis control function is:

0 : Not effective.
1 : Effective.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Data type] Byte axis

[Valid data range] 1 to 4

Specify the DI/DO group to be used to specify a command for each PMC–controlled axis.

Value	Description
1	DI/DO group A (G142 to G153) is used.
2	DI/DO group B (G154 to G165) is used.
3	DI/DO group C (G166 to G177) is used.
4	DI/DO group D (G178 to G189) is used.

> **NOTE**
> If another value is specified, the axis is not PMC–controlled.

8022	Maximum feedrate for feed per rotation along a PMC–controlled axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Specify the maximum feedrate for feed per rotation along a PMC–controlled axis

> **NOTE**
> The maximum feedrate set to first axis is valid for all axes. The data after the second axis is invalid.

8028	Linear acceleration/deceleration time constant for jog feed specified by the speed command for each PMC–controlled axis

[Data type] Word axis

[Unit of data] msec/1000 min^{-1}

[Valid data range] 0 to 32767

Specify, for each PMC–controlled axis, the time needed to increase or decrease the speed of the servo motor by 1000 min^{-1}, that is, the time constant of linear acceleration/deceleration for jog feed according to the speed specified for that axis.

NOTE
If 0 is specified, the system does not control the acceleration/deceleration.

Alarm and message

A servo alarm or overtravel alarm for a PMC–controlled axis is detected in the same way as an alarm for a CNC–controlled axis.

If an alarm occurs, the alarm is handled by using the normal procedure, alarm signal EIALg is set to “1” to inform the PMC of the alarm.

(If an overtravel alarm occurs, either negative overtravel signal EOTNg or positive overtravel signal EOTPg is also set to “1”.)

If the PMC issues a command for a CNC–controlled axis, a P/S alarm No. 130 occurs.

Commands issued by the PMC are effective if the axis is in feed hold or single block stop mode. The command results in the issue of an alarm if cutting feed is executed with an override of 0%, or if interlock is enabled.

If the CNC issues a command for a PMC–controlled axis, a P/S alarm No. 130 occurs.

If the PMC issues a movement command for an axis in the plane of polar coordinate interpolation in polar coordinate interpolation mode (G12.1), a P/S alarm No. 130 occurs.

(1) P/S Alarm

Number	Message	Description
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	A currently selected PMC controlled axis was selected again. Or, the CNC issued a control command for an axis that has been set as a PMC–controlled axis for which no command has been specified. Or, an axis under control of CNC was selected by the PMC.

(2) Servo alarm

Number	Message	Description
417	SERVO ALARM: n AXIS EXCESS ERR	An invalid parameter is specified for torque control. The torque constant parameter is set to 0.
422	SERVO ALARM: n AXIS EXCESS ERR	The maximum speed permitted under torque control has been exceeded.
423	SERVO ALARM: n AXIS EXCESS ERR	The maximum permissible value of integrated travel distance set in the parameter under torque control has been exceeded.

Warning

WARNING

1 The mode selection, CNC reset, and other CNC statuses have no effect.

2 Feed hold, single block stop, reset, or interlock of one or all axes, performed by the CNC, does not affect a PMC–controlled axis. Similar control is possible by using the equivalent signals issued from the PMC.

3 The mirror image functions (setting, parameter, input signal) are disabled.

Caution

CAUTION

1 Emergency stop or machine lock is enabled. Machine lock can be disabled if the MLE bit (bit 0 of parameter No. 8001) is set to “1”. However, machine lock for an individual axis is always enabled.
2 In consecutive cutting feed blocks, a new block starts its operation without waiting for the following zero of the servo acceleration/deceleration. For all other cases, a new block starts its operation after the following zero of the servo acceleration/deceleration is confirmed.
3 Commands for a linear axis that is controlled solely by the PMC (not used as an axis controlled by the CNC) are not affected by inch/metric input setting of the PIM bit (bit 0 of parameter No. 8003). The current position display is also not affected by inch/metric input.
4 For a PMC–controlled axis, manual absolute mode is always set. If the PMC starts control of an axis after manual intervention (manual continuous feed, manual handle feed, etc.) is performed during automatic operation while manual absolute mode is not set (*ABSM is set to 1), manual absolute mode is set.
5 Under PMC axis control, all commands are handled as axis commands. Even for the miscellaneous functions, the position check is effective.
6 When the CNC executes the command to set the workpiece coordinate system setting (G54 to G59) during an axial movement by the PMC, a valid coordinate system cannot be set.

Note

NOTE

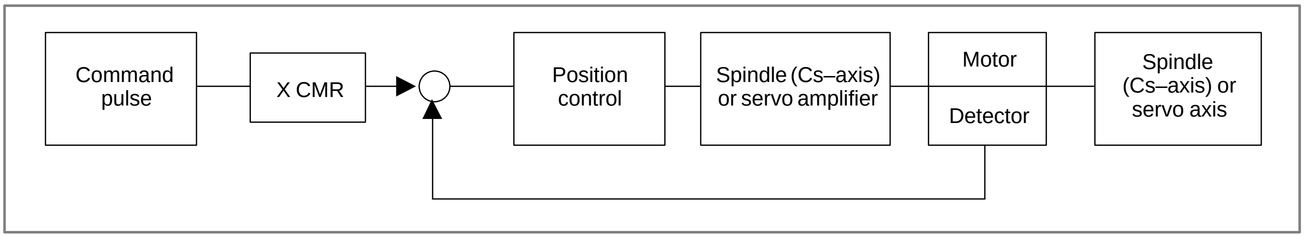
1 The actual speed excluding the effect of movement along a PMC–controlled axis can be displayed if the PCF bit (bit 1 of parameter No. 3105) is set to “1”.
2 If an absolute pulse coder is used, a specified reference position is retained in memory, even after the power is turned off.
3 If the index table indexing function of the M series is used, the PMC cannot control the fourth axis.
4 The individual output of the miscellaneous function is provided by adding a signal for individual output. The timing diagram of controlling and specifying the miscellaneous function is not changed. The normal specifications of the miscellaneous functions for PMC axis control function are applied.

15.1.2 Constant Velocity Command Position Control

Overview

This function enables control of the Cs–axis by executing a constant velocity command for PCM–axis control using position loop control. In other words, the function can operate the PMC–axis at a constant velocity by outputting command pulses equivalent to the rotational speed to the servo position control below, instead of outputting rotational speed data to the amplifier.

Block diagram



Specifications

Bit 4 of parameter No. 8005 specifies either position loop control or velocity loop control is used to execute a constant velocity command for PMC–axis control.

For a constant velocity command, acceleration or deceleration is performed only by its dedicated parameters and normal acceleration or deceleration is not performed even in position control (bit 4 of parameter No. 8005 is 1).
In position control (bit 4 of parameter No. 8005 is 1), an override is enabled.

Set the amount of travel per rotation, expressed in least command increments, in parameter No. 8040 for each axis and then use this to convert from a velocity command to a position command.

The following expression is obtained by assuming that P is output pulses (least command increment).

$$P = \frac{\text{Velocity command} \times \text{Parameter No. 8040} \times \text{ITP time} \times \text{Override}}{60 \times 1000 \times 100}$$

Notes

This function is included in the PMC–axis control function.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8005				PMCPOS				

Parameter input

[Data type] Bit

PMCPOS A constant velocity command for PMC–axis control is performed as:

0 : A velocity command.
1 : A position command.

8040	Amount of travel per rotation expressed in least command increments when a position command is executed as a constant velocity command for PMC–axis control

Parameter input

[Data type] 2 word axis

[Unit of data]

Setting unit	IS–B	IS–C
Millimeter machines	0.001	0.0001
Inch machines	0.0001	0.00001
Rotation axis	0.001	0.0001

[Valid data range] 1 to 999999999

When a constant velocity command for PMC–axis control is performed as a position command (bit 4 of parameter No. 8005 is 1), set the amount of travel per rotation expressed in least command increments.

15.2 EXTERNAL DATA INPUT

General

The following signals are used to send data from the PMC to the CNC.

Signal name	Signal code
Data signal for external data input (input)	ED0 to ED15
Address signal for external data input (input)	EA0 to EA6
Read signal for external data input (input)	ESTB
Read completion signal for external data input (output)	EREND
Search completion signal for external data input (output)	ESEND

The basic external data input procedure is described below:

(1) The PMC sets the address signals, EA0 to EA6 that indicate the data type and data signals ED0 to ED15.

(2) The PMC sets the read signal ESTB to “1”.

(3) When the ESTB signal is set to “1”, the control unit reads the address.

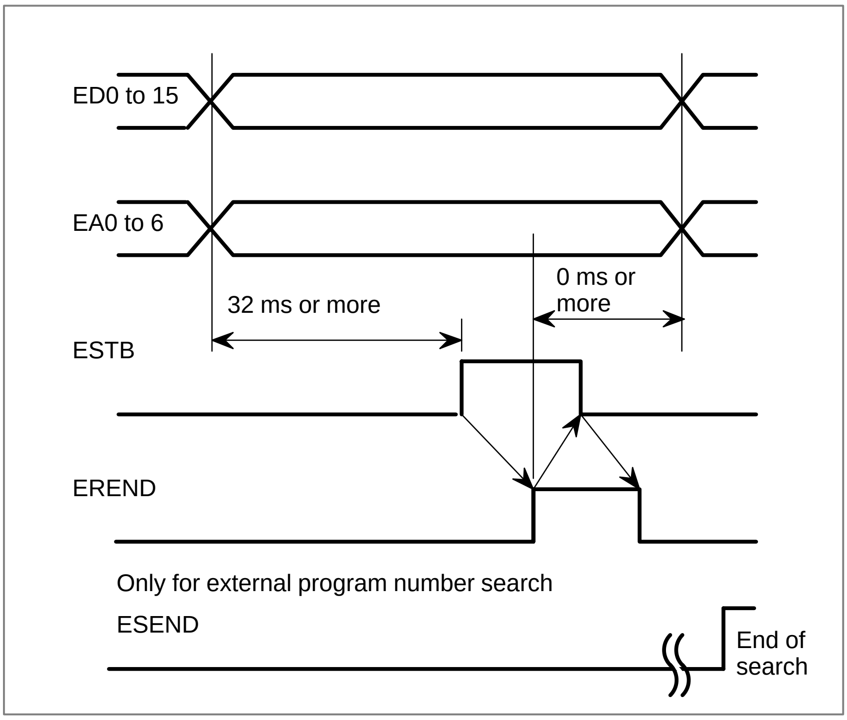
(4) After reading the address, the control unit sets the read completion signal EREND to “1”.

(5) When the EREND signal is set to “1”, the PMC sets the ESTB signal, to 0.

(6) When the ESTB signal is set to “0”, the control unit sets the EREND signal to 0.

This completes the data input procedure. New data can now be entered.

The timing diagram is shown below:



Data types accessed by external data input

No.	Item	E S E E E T A A A B 6 5 4	E E E E A A A A 3 2 1 0	ED15 to ED0			
				15141312	1110 9 8	7 6 5 4	3 2 1 0
1	External program number search	1 0 0 0	××××	Program number(BCD4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9
2	External tool compensa- tion	1 0 0 1	××××	Offset value(BCD 4 digits with sign)			
				±0 to 7	0 to 9	0 to 9	0 to 9
3	External workpiece coordi- nate system shift	1 0 1 0	axis code	Shift value(BCD 4 digits with sign)			
				±0 to 7	0 to 9	0 to 9	0 to 9
4	External machine coordi- nate system shift	1 0 1 1	axis code	Machine coordinate system shift value(binary) ±0 to 9999			
5	Alarm set	1 1 0 0	0 0 0 0	Alarm No.(binary)			
					0 to 999		
	Alarm clear	1 1 0 0	0 0 0 1	Alarm No.(binary)			
					0 to 999		
	Oper to tor message list	1 1 0 0	0 1 0 0	Message No.(binary)			
					0 to 999		
	Operator message clear	1 1 0 0	0 1 0 1	Message No.(binary)			
					0 to 999		
	Message	1 1 0 0	0×1 1	character(Character code)			
6	Substitute No. of parts required	1 1 1 0	0 0 0 0	No. of parts required(BDC4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9
	Substitute No. of parts machined	1 1 1 0	0 0 0 1	No. of parts machined(BDC4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9

WARNING

Though bits EA4 to EA6 distinguish one set of data from another, the PMC must be interlocked in order to prevent data from other functions from being transmitted when another function is in progress.

NOTE

Input an axis code according to the list below.

Axis	EA3 to EA0			
	3	2	1	0
1st axis	0	0	0	0
2nd axis	0	0	0	1
3rd axis	0	0	1	0
4th axis	0	0	1	1

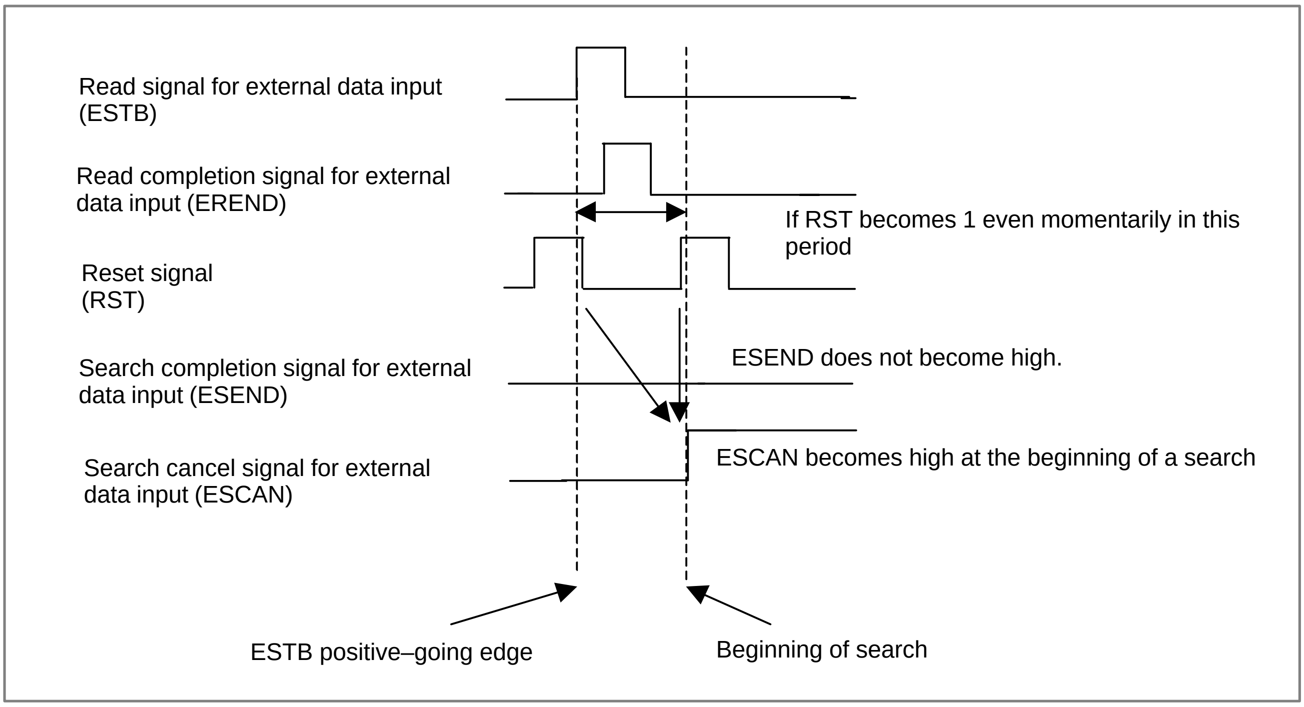
1) External Program Number Search

A program number (1 to 9999) is specified from an extended source and is selected in the CNC memory.
For machines that can load several kinds of workpieces, this function can automatically select the program to be executed corresponding to a specific workpiece.
Data for the external program number search is accepted regardless of CNC mode, but the search execution can be made only in the reset state in MEM mode.

The ESEND signal switches from “0” to “1” on completion of external program number search. This signal does not turn to “0” unless the cycle start or reset signal is input, or another search is made. Use ESEND to initiate a cycle start signal after the search.

Because a search operation is deferred until a reset occurs in the MEM mode, an external program number search attempted during a CNC operation (OP = 1) results in the program being started immediately after the end of an automatic operation in a sequence in which the automatic operation is started by checking only the ESEND signal (search completion signal for external data input). For this reason, using the ESC parameter (bit 3 of parameter No. 6300) enables an external program number search to be canceled at a CNC reset. Concretely, if a reset signal is input between the time the ESTB signal (read signal for external data input) is input and the time a search would be executed, setting the ESC parameter (bit 3 of parameter No. 6300) to 1 keeps the search from being executed. At the same time, the controller uses the ESCAN (search cancel signal for external data input) signal to inform the PMC that the search has been canceled. The ESEND signal (search completion signal for external data input) will not become 1, because the search is canceled.

The controller checks the state of the RST (NC reset) signal for a reset input between the rising edge of the ESTB signal (read signal for external data input) and the start of the search. Concretely, the controller cancels the external program number search if the RST (NC reset) signal becomes 1 even momentarily in this period. The ESCAN signal (search cancel signal for external data input) becomes high at the beginning of the search. However, it becomes low when the next search is executed (for example, when a cycle start or reset signal is input), similarly to the ESEND signal (search completion signal for external data input).



NOTE

1. The external program number search is valid when parameter ESR no. 6300#4=1.
2. In reset state the automatic operation lamp will be off. If the start button is pushed in the cycle operation stop or hold state, search execution starts from the actual position indicated by the pointer.
3. When there is no program stored in memory corresponding to the set program number, P/S alarm no. 59 will be activated.
4. Program search is not made if the program number is set to “0”. Insteadly when the start button is pushed, execution starts from the position indicated by the pointer.
5. Data for the external program number search is accepted regardless of the mode, but the search execution can be made only in the reset state. Therefore, in case that the PMC sequence, which the cycle start is executed by checking search completion signal for external data input only, is used, if the external program number search is commanded twice, the program is executed twice. (When CNC accepts a command of the program number search, the command is not cancelled even if CNC becomes reset state by external reset signal and so on.) (See Fig. 15.2)
   If the program execution after reset becomes a problem, please make the PMC sequence not to execute the cycle start after reset.

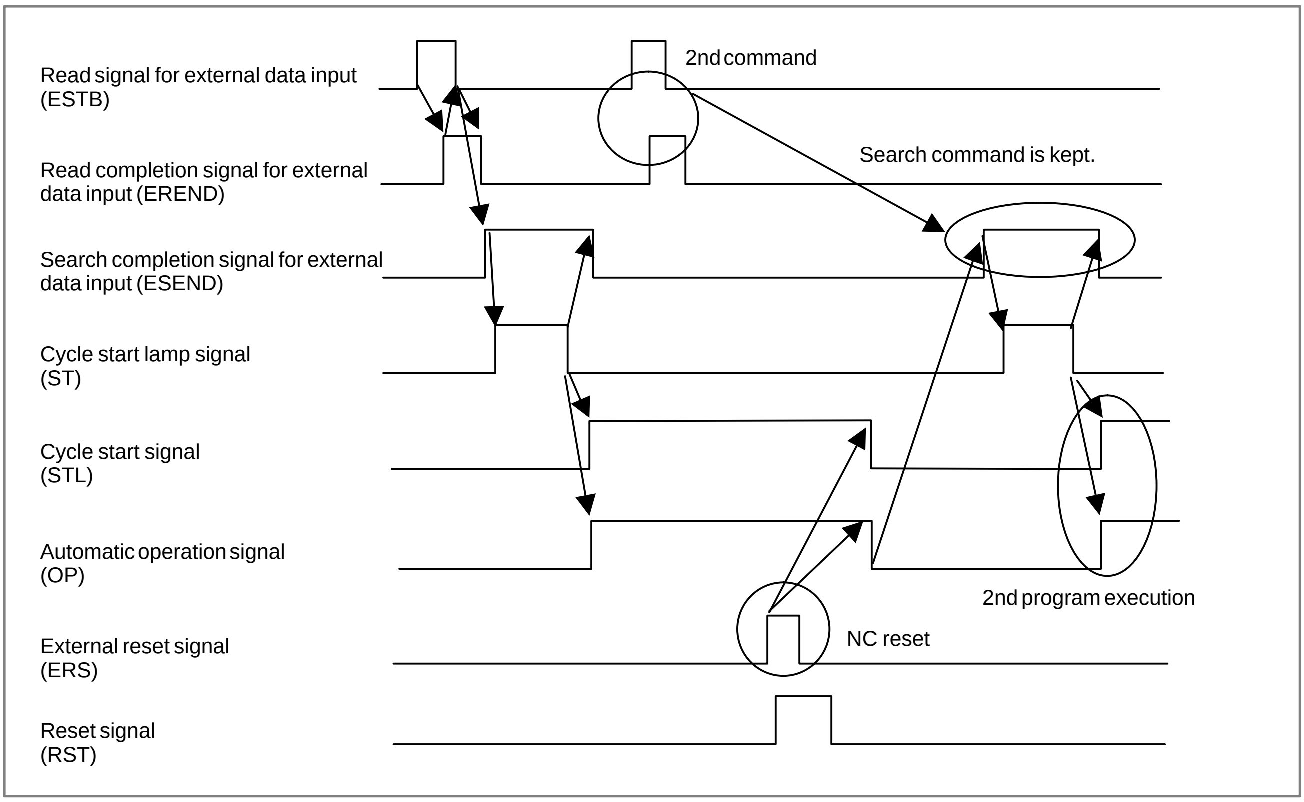


Fig. 15.2

2) External tool compensation

These signals provide for changing the tool compensation amount via the PMC. When the offset number is specified by a program, data input from the PMC is added to the offset amount. The offset amount can also be used as input data itself by specifying the input signal.

When the machine tool is equipped with automatic tools or workpiece measuring functions, the offset amount can be corrected using this function, by inputting the error from the correct value into the CNC via PMC.

If the tool compensation amount is externally input when offset number 0 is specified in a program (a offset cancel) in T series, the workpiece coordinate system shifts by the entered quantity. The external tool compensation range is 0 - $\pm$7.999mm or $\pm$0.7999 inch at a time.

Data specification for external tool compensation (For M series)

Address							Data															
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
0	0	1	ABS/INC	0	0	0	sign	MSB LSB Offset amount														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000 to 7999)

Specify “000”

0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

Specify “001” for external tool offset

Data specification for external tool compensation (For T series)

Address							Data															
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
0	0	1	ABS/INC	W/G	X/Z/R		sign	MSB LSB Offset amount														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000A7999)

EA1	EA0	Description
0	0	Specification of X axis offset
0	1	Specification of Z axis offset
1	0	Specification of tool nose radius compensation amount

0 : Specification of tool wear offset amount
1 : Specification of tool geometry offset amount

0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

Specify “001” for external tool offset

3) External workpiece coordinate system shift

The external workpiece coordinate system shift adjusts the workpiece coordinate system depending on the shift amount set via the PMC. Each axis (parameter No. 1220) has this shift amount, and it is added to all the workpiece coordinate systems for use. The shift amount is not lost by cut off of the power supply. It is not added incrementally, but each input shift amount generates a new shift amount. The range that can be input is 0-$\pm$7.999 mm or $\pm$0.7999 inch.

4) External machine coordinate system shift

The machine coordinate system can be shifted by inputting a shift value. When the shift amount is input, compensation is immediately applied to the corresponding axis and the machine starts operation. The position accuracy can be improved by combining this function with sensors.
The specification to shift the axis is the same as that for the external workpiece coordinate system shift.
The compensation value is specified in signals ED0 to ED15 using a binary code ranging from 0 to $\pm$9.999. This compensation value should be absolute and the amount the machine actually moves on input is the difference from the previously stored value. When a large amount of compensation is applied at one time, an alarm such as “excessive error on stop” may occur. In this case, input the compensation in several smaller increments.

5) External alarm message

(a) External alarm message
The external alarm message holds the CNC in an alarm condition by sending an alarm number from the external device, as well as a message that is displayed on the CRT screen of the CNC. Up to four alarm numbers and messages can be sent at a time; the alarm number ranges from 0 to 999. The CNC displays adds 1000 to the alarm number. The message for one alarm number can be up to 32 characters long. The alarm condition is reset by external data.

(b) External operator message
The external operator message sends the operator message and number from the external unit to the CNC, with a display on the CRT screen of the CNC.
Only one message can be transmitted, with a potential message length of 255 characters. The alarm number ranges from 0 to 999; from 0 to 99, the CNC adds 2000 to the number, while from 100 to 999 the number is not displayed, only the message is displayed.
However, the number of messages to be displayed with a message number can be changed by setting parameter No. 6310.

Data specification method in external message

Item	EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15 to ED0 (binary)
Alarm set	1	0	0	0	0	0	0	Alarm No.
Alarm clear	1	0	0	0	0	0	1	Alarm No.
Operator message list	1	0	0	0	1	0	0	Message No.
Operator message clear	1	0	0	0	1	0	1	Message No.
Message	1	0	0	0	×	1	1	Character (Note)

NOTE

Two characters are sent at a time (see ISO code given in the table below).
ED15 to ED8 ...... Character code in 1st character.
ED7 to ED0 .... Character code in 2nd character.
When sending only one character, fill the second slot with a code smaller than 20 and it will be ignored.

Character code table

b_8				0	0	0	0	1	1	1	1
b_7				0	0	1	1	0	0	1	1
b_6				1	1	0	0	1	1	0	0
b_5				0	1	0	1	0	1	0	1
b_4	b_3	b_2	b_1								
0	0	0	0	SP	0	@	P		–	タ	ミ
0	0	0	1	!	1	A	Q	#	ア	チ	ム
0	0	1	0	I	2	B	R	V	イ	ツ	メ
0	0	1	1	#	3	C	S	W	ウ	テ	モ
0	1	0	0	$	4	D	T	”	エ	ト	ヤ
0	1	0	1	%	5	E	U	·	オ	ナ	ユ
0	1	1	0	&	6	F	V	ヲ	カ	ニ	ヨ
0	1	1	1	’	7	G	W	ァ	キ	ヌ	ラ
1	0	0	0	(	8	H	X	ィ	ク	ネ	リ
1	0	0	1	)	9	I	Y	ゥ	ケ	ノ	ル
1	0	1	0	*	:	J	Z	ェ	コ	ハ	レ
1	0	1	1	+	;	K	[	ォ	サ	ヒ	ロ
1	1	0	0	,	<	L	o	ャ	シ	フ	ワ
1	1	0	1	=	=	M	]	ュ	ス	ヘ	ン
1	1	1	0	.	>	N	^	ョ	セ	ホ	m
1	1	1	1	/	?	O	_	ッ	ソ	マ	#

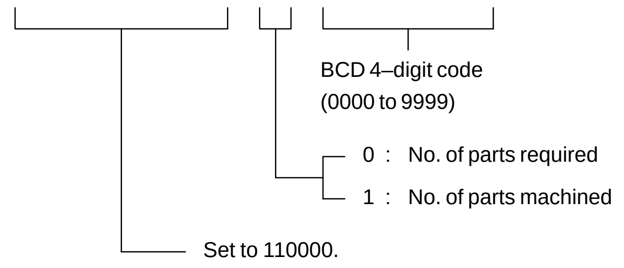
SP : Space code

6) Substituting No. of parts required and No. of parts, machined

Substitution is possible for the No. of parts required and the No. of parts machined.

Data specification for No. of parts required and No. of parts machined.

Address							Data		
EA6	EA5	EA4	EA3	EA2	EA1	EA0	ED15	• • •	ED0
1	1	0	0	0	0	0/1	MSB LSB Value of aprt nos.		



Signals

Data signals for external data input ED0 to ED15 <G000, G001>

[Classification] Input signal

[Function] These signals indicate the entered data.
The use of the 16 code signals varies with the data type.

Address signals for external data input EA0 to EA6 <G002>

[Classification] Input signal

[Function] These signals indicate the type of the entered data.

Read signal for external data input ESTB <G002#7>

[Classification] Input signal

[Function] The signal reports that the address and data are set in external data input.

When the signal is set to “1”, the control unit reads the address and data for external data input.

[Operation] The “basic procedure” describes the procedure for, and operation of, the control unit when the signal turns to “1”.

Read completion signal for external data input EREND <F060#0>

[Classification] Output signal

[Function] This signal reports that the control unit has finished reading the entered data.

[Operation] The output condition and procedure are described in the “basic procedure.”

Search completion signal for external data input ESEND <F060#1>

[Classification] Output signal

[Function] This signal reports that program number search, specified by external data input, has been completed.

[Output condition] This signal is set to “1” when:

The program number search specified by external data input is completed.

The signal is set to “0” when:

- An automatic operation is started.
- A reset occurs.

Search cancel signal for external data input ESCAN <F060#2>

[Classification] Output signal

[Function] Informs the PMC that a program number search has been canceled.

[Output condition] If a reset signal is input between the time the ESTB signal (read signal for external data input) has been input and the time a search would be executed, the external program number search function keeps the search from being executed if the ESC (bit 3 of parameter No. 6300) is 1. The controller sets, to 1, the ESCAN signal (search cancel signal for external data input) rather than the ESEND signal (search completion signal for external data input).

> **NOTE**
> This signal is valid only if the ESC parameter (bit 3 of parameter No. 6300) is 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0

	#7	#6	#5	#4	#3	#2	#1	#0
F060						ESCAN	ESEND	EREND

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3202		PSR						

[Data type] Bit

PSR Search for the program number of a protected program

0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6300				ESR	ESC			

[Data type] Bit

ESC Specifies whether the external program number search function is to execute a search if a reset signal is input between the time the ESTB signal (read signal for external data input) is input and the time the search would be executed, as follows:

0 : The search is executed after the CNC is reset.
1 : No search is executed.

ESR External program number search

0 : Disabled
1 : Enabled

6310	Number of added message numbers of external operator messages

[Data type] Word

[Valid data range] 0, 1 to 1000

For external operator messages within the range set in this parameter, message numbers each obtained by adding 2000 to the relevant message number are indicated.

NOTE

1 When this parameter is set, the power must be turned off before operation is continued.
2 If 0 or a value beyond the valid data range is set in this parameter, this parameter becomes invalid.

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	In an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have been generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.

15.3 EXTERNAL WORKPIECE NUMBER SEARCH

15.3.1 External Workpiece Number Search

General

When several part programs are stored in program storage memory, a program can be searched with the workpiece number search signals PN1 to PN16 from the machine side.

When the cycle operation is actuated in the memory operation mode under reset status, the workpiece number (program number) specified by PN1 to PN16 is searched and executed from the beginning.

Signal

Workpiece Number Search Signal PN1, PN2, PN4, PN8, PN16 <G009#0 to #4>

[Classification] Input signal

[Function] Select the number of a workpiece to be machined in the memory mode. Five code signals are provided. These signals are set in binary code to designate a workpiece number as follows:

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09
0	1	0	1	0	10
0	1	0	1	1	11

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

Workpiece number 00 is a special designation indicating “no search”. Thus, workpiece numbers ranges from 01 to 31.

NOTE
These signals are also used to specify a file number for file search during external program input. See Section 13.4, “External Program Input.”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G009				PN16	PN8	PN4	PN2	PN1

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	During an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue background editing.

Note

NOTE

1 This function can be used only in memory operation. It cannot be used during DNC operation and MDI operation.
2 Select the program number from O001 to O031.
3 Program numbers from O001 to O031 can be used. However, programs corresponding to all the program numbers do not have to be stored in memory.
4 When a program corresponding to the specified program number is not stored in memory, an alarm (No. 059) is activated when the start button is pressed.
5 Program search is performed only when the start button is pressed in the reset state. When the CNC is in the automatic operation stop state (single block stop, etc.) or pause state (feedhold stop, etc.), program search is not performed even if the start button is pressed and execution is started from the point specified by the present execution pointer.
6 To restart a program halfway through, press the start button after sequence number search in MEM mode. The workpiece number search is not performed; program execution starts from the block searched using sequence number search, because the OP signal is set by sequence number search in MEM mode and the CNC reset state is released.
7 When the start button is pressed with all PN1 to PN16 all set to “0”, program search is not performed but execution is started from the point specified by the present execution pointer. To restart operation from the start of a program which cannot be searched using this function, perform the standard program number search operation (MDI panel operation), set PN1 to PN16 to “0” and press the start button

15.3.2 Expanded External Workpiece Number Search

General

Using the EPN0 to EPN13 (expanded external workpiece number search) signals enables a search for program numbers O0001 to O9999.

Unlike the workpiece number search signal, which triggers an automatic operation after a program search, these signals can make a program search without triggering an automatic operation, because they can be used with the EPNS (expanded workpiece number search start) signal.

Signal

Expanded workpiece number search signals EPN0 to EPN13 <G024, G025#0 to 5>

[Classification] Input signal

[Function] These signals specify the number of the program to be executed in the memory mode, using a binary code. The program numbers that can be specified by these signals range from 1 to 9999.
(Program number 0 is used to make the special specification that ”no search is to be made.”

These signals are valid if the EPN parameter No. (bit 1 of parameter No. 3006) is 1.

If the EPN parameter (bit 1 of parameter No. 3006) is 0, the conventional workpiece number search signals PN1, PN2, PN4, PN8, and PN16 <G0009#0 – #4) are valid.

Expanded workpiece number search start signal EPNS <G025#7>

[Classification] Input signal

[Function] This signal causes the workpiece number search function to be executed without performing an automatic operation. When the signal changes from 1 to 0, a search is executed.

Setting the EPS parameter (bit 2 of parameter No. 3006) enables the EPNS signal and disables the conventional search function that is based on the ST (cycle start) signal.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G024	EPN7	EPN6	EPN5	EPN4	EPN3	EPN2	EPN1	EPN0

	#7	#6	#5	#4	#3	#2	#1	#0
G025	EPNS		EPN13	EPN12	EPN11	EPN10	EPN9	EPN8

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3006						EPS	EPN	

EPN Workpiece number search signals are assigned to:
0 : PN1, PN2, PN4, PN8, and PN16.
1 : EPN0 to EPN13.

EPS When a program is searched using the workpiece number search function, it is started by:
0 : Automatic operation start signal ST (when automatic operation (memory operation) is started).
1 : Workpiece number search start signal EPNS. (Search is not started by ST.)

NOTE
If this parameter is 1, the ST (cycle start) signal cannot be used to start a search.

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	In an external program number search a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.

Note

NOTE
1 This function is usable only in the memory mode; it cannot be used in any other mode.
2 If a program that corresponds to a specified program number has not been stored in memory, an alarm (No. 59) is issued when a start signal (ST or EPNS) is input.

15.4 SPINDLE OUTPUT CONTROL BY THE PMC

General

The PMC can control the speed and polarity of each spindle motor, connected by the spindle serial output/spindle analog output function.

The first to third spindles each have their own individual interfaces. By using a PMC ladder program, the user can control the spindles as desired.

This section describes how to use the PMC to control spindle rotation and provides example applications.

Switching control

This function can be used to specify the following:

· Spindle motor speed (number of rotations)

· Output polarity for each spindle motor (direction of rotation)

Usually, the CNC is used to control the speed and polarity of the first spindle motor. If a multispindle control function is added, the CNC can also control the second to third spindle motors.

This function allows the user to select whether the CNC or PMC is used to control the speed and output polarity of the spindle motors.

Specifying the spindle motor speed

The PMC can be used to specify the spindle motor speed upon executing the following:

· Switching the controller from the CNC to the PMC, by issuing SINDx signal

· Setting the spindle motor speed data, calculated by the PMC, in spindle control signals R01Ix to R12Ix

When controlled by the PMC, the spindle motor speed is not affected by any signal (for example, the spindle speed override signal) or parameter settings (for example, the maximum speed clamp parameter) related to the spindle speed command of the CNC spindle control function.

→ If the multispindle control function is added, however, the spindle stop signal *SSTPx <G0027#3, #4, #5> can be used to stop a PMC–controlled spindle.

The spindle motor speed data is obtained from the following expression. Its value can range from 0 to 4095:

$$\text{Spindle motor speed data} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

Remark) Usually, the spindle speed must be controlled. If a gear train is used to connect the spindle to the spindle motor, first obtain the maximum spindle speed at the maximum spindle motor speed.

$$\text{Spindle motor speed data} = \frac{\text{Spindle speed}}{\text{Maximum spindle speed}} \times 4095$$

By using this expression, the spindle motor speed data can easily be obtained.

Specifying the output polarity for the spindle motor

The PMC can specify the spindle motor output polarity when the following are executed:

· Switching the controller from the CNC to the PMC, by issuing an SSINx signal

· Specifying the output polarity to the SGNx signal

S–code and SF signals

To control the spindle, the PMC may be required to read the S value specified by the CNC.

If the spindle serial output/spindle analog control function is added (if the PMC can control the spindle), the S–code signals <F022 to F025> and SF signal <F007#2> can be output only when several conditions, determined by the CNC spindle control, are satisfied. In some cases, the signals cannot be used under standard conditions.

Specify the related bits of parameter No. 3705 according to the desired application, then use the S–code and SF signals.

Twelve code signals corresponding to the S value (output)

Twelve code signals corresponding to S value R01O to R12O <F036#0 to F037#3> are output to the first spindle motor. The output data is calculated from the results of the CNC spindle control. (See Section 9.3.)

Even while a spindle is subject to PMC control, an S command that is issued to the CNC is converted to spindle output data and output.

The SIND signal determines whether the speed output command, issued to the spindle motor, is obtained from the twelve code signals corresponding to the S value, or from the R01I to R12I signals calculated and specified by the PMC.

The use of this signal may simplify PMC ladder processing used to enable PMC spindle control.

Sample application 1)

Controlling the first and second spindles of a lathe system

→ Share the gear stages between the first and second spindles.

(If the first spindle uses two gears, for example, specify parameters Nos. 3743 and 3744, thus enabling the use of gears 3 and 4 for the second spindle.)

Perform the necessary setting to enable control of the first and second spindles by the PMC.

To specify a rotation command for the first spindle, enter the gears for the first spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the first spindle in the PMC control interface for the first spindle.

To specify a rotation command for the second spindle, enter the gears to be used for the second spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the second spindle in the PMC control interface for the second spindle.

Sample application 2)

Using a lathe's orientation function with the stop position of the serial spindle specified externally, specifying the S value as the angle of the stop position for spindle orientation after the spindle positioning mode has been selected

→ Use the gears that are not being used for the first spindle.

(In this application, gear 4 is used to calculate the spindle position. Set parameter No. 3744 to 360.)

Specify the M code used to set the spindle to positioning mode and stop the spindle. Enter gear 4 in GR1 and GR2.

Then, specify a spindle positioning angle with the S command. (To specify the position of 145 degrees, for example, specify S145;.)

Expression $145/360 \times 4095$ is calculated and the result is output to the twelve code signals corresponding to the S value (output signal). Enter the data in external stop position commands SHA00 to SHA11 <G078#0 to G079#3> and perform the orientation.

Signal

PMC spindle control signals

For the first spindle: **SIND, SSIN, SGN <G033#7, #6, #5>**
R01I to R12I <G032#0 to G033#3>

For the second spindle: **SIND2, SSIN2, SGN2 <G035#7, #6, #5>**
R01I2 to R12I2 <G034#0 to G035#3>

For the third spindle: **SIND3, SSIN3, SGN3 <G037#7, #6, #5>**
R01I3 to R12I3 <G036#0 to G037#3>

[Classification] Input signal

[Function] The above signals enable the control of a spindle motor by issuing commands from the PMC. Both the speed and polarity of the spindle motor (direction of rotation) can be controlled.

The speed command and polarity are usually specified by the CNC. The use of these signals allows the user to select whether the speed and polarity are controlled by the CNC or PMC.

Even if the multispindle control function is not provided, these signals allow the second and third spindles to be controlled.

When the multispindle control function and type A are being used (if the MSI bit, bit 2 of parameter No. 3709, is set to 0), the signals for the second and third spindles cannot be used.

- **Details of the signals**

- Signal used to select the spindle motor speed command SINDx

→ The above signal is used to select whether the spindle motor speed is controlled by the CNC or PMC.

1: The spindle motor is controlled according to speed commands (R01Ix to R12Ix) issued by the PMC.

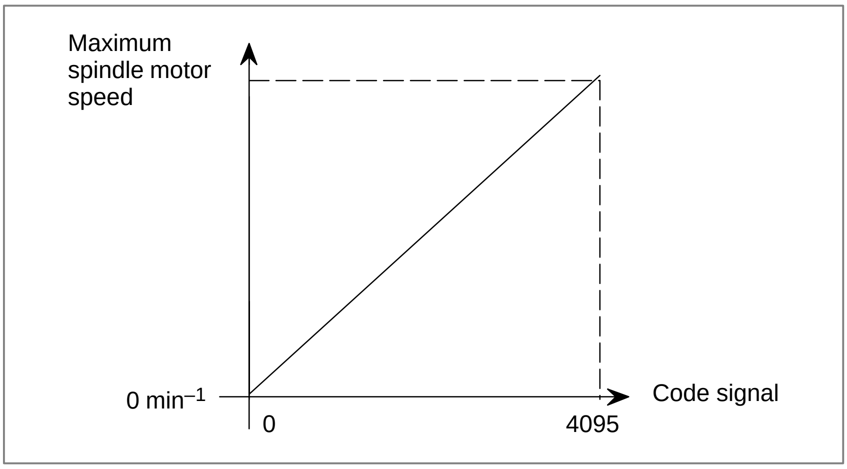
0: The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- Signals used to input the spindle motor speed command issued by the PMC R01Ix to R12Ix

→ If the PMC is being used to control the spindle motor speed command, specify, in binary format, the value obtained using the following expression.

$$\text{Value to be specified} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

(Spindle motor speed)



- Signal used to select the polarity of the spindle motor speed command, SSINx

→ The above signal selects whether the output polarity of the spindle motor speed command is controlled by the CNC or PMC.

1 : The spindle motor is controlled according to the polarity command (SGNx) issued by the PMC.

0 : The CNC controls the polarity. The polarity is determined by the TCW and CWM bits (bits 7 and 6 of parameter No. 3706) and the M03 or M04 command.

- Signal used to specify the polarity of the spindle motor selected by the PMC, SGNx

→ If the PMC is used to control the output polarity of the spindle motor speed command, specify the polarity with this signal.

1 : The output polarity of the spindle is negative.

0 : The output polarity of the spindle is positive.

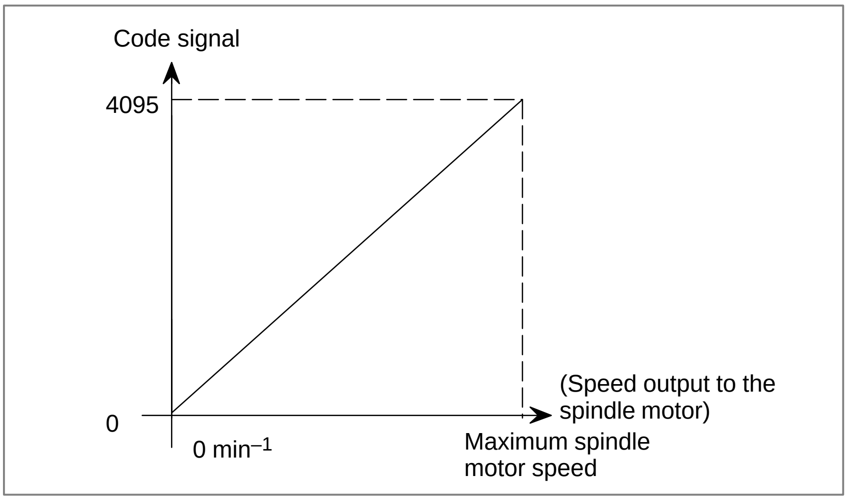
Twelve code signals corresponding to the S value R01O to R12O <G036#0 to G037#3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output of the spindle motor that is required to control the connected spindle. The converted value is sent to the PMC with twelve code signals, in propotional to the spindle motor speed output.

The speed data, the final result of the CNC spindle control, is output to the spindle motor after the spindle gear ratio, spindle speed override, speed clamp, conversion of the surface speed into the spindle speed by the constant surface speed control command, and other data have been considered.

(See Section 9.3 for an explanation of the relationship between the CNC spindle control and the speed output to the spindle motor.)



	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF					ESF

[Data type] Bit

ESF When the spindle control function (S analog output or S serial output) is used, and the constant surface speed control function is used, or bit 7 (GTT) of parameter No. 3705 is set to 1:

0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

NOTE

For the T series, this parameter is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1. For the M series, SF is not output:

(1) For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode

(2) When bit 5 (NSF) of parameter No. 3705 is set to 1

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.
1 : Output for an S command.

NOTE

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S–––;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface–speed control,

0 : SF is output.
1 : SF is not output.

SFA: The SF signal is output:

0 : When gears are switched
1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		

[Data type] Bit

MSI In multi–spindle control, the SIND signal is valid:

0 : Only when the first spindle is selected. (SIND signal for 2nd and 3rd spindle become invalid)
1 : For each spindle irrespective of whether the spindle is selected. (Each spindle has its own SIND signal.)

Note

NOTE

1 If the spindle fails to move after the PMC issues the spindle motor speed command, check the following:

Type A is selected (the MSI bit, bit 2 of parameter No. 3709, is set to 0) when the multispindle control function is used.
→ The second and third spindles cannot be controlled. The first spindle can be controlled only when the spindle selection signal SWS1 is set to “1”.

The spindle stop signal for each axis is set to “0” when the multispindle control function is being used.
→ Spindle stop signal for each axis *SSTPx <G027#3, #4, #5> stops the spindle.

M03/M04 is not specified when the CNC is being used to control the output polarity.
→ If the TCW bit, bit 7 of parameter No. 3706, is set to 1, the M03/M04 command issued to the CNC changes the output polarity for the spindle motor. If no M03/M04 command is specified after the CNC is turned on, the specified speed output is not sent to the spindle motor because the output polarity has not been determined.

2 The SF signal indicates that output of the S code to the PMC has been completed. The signal does not indicate the end of the command for specifying the spindle speed.

3 For an explanation of connecting the second spindle, see Sections 9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT and 9.9 MULTI–SPINDLE CONTROL.

4 The CNC does not issue any commands to the second and third spindles. The output polarity is controlled by the SGNx signal. It is not affected by the SSINx signal.
The speed output to the spindle motor can be controlled only when the SINDx signal is set to “1”.

Reference Item

CONNECTION MANUAL (This manual)	9.2	SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT
	9.3	SPINDLE SPEED CONTROL
	9.9	MULTI–SPINDLE CONTROL

15.5 EXTERNAL KEY INPUT

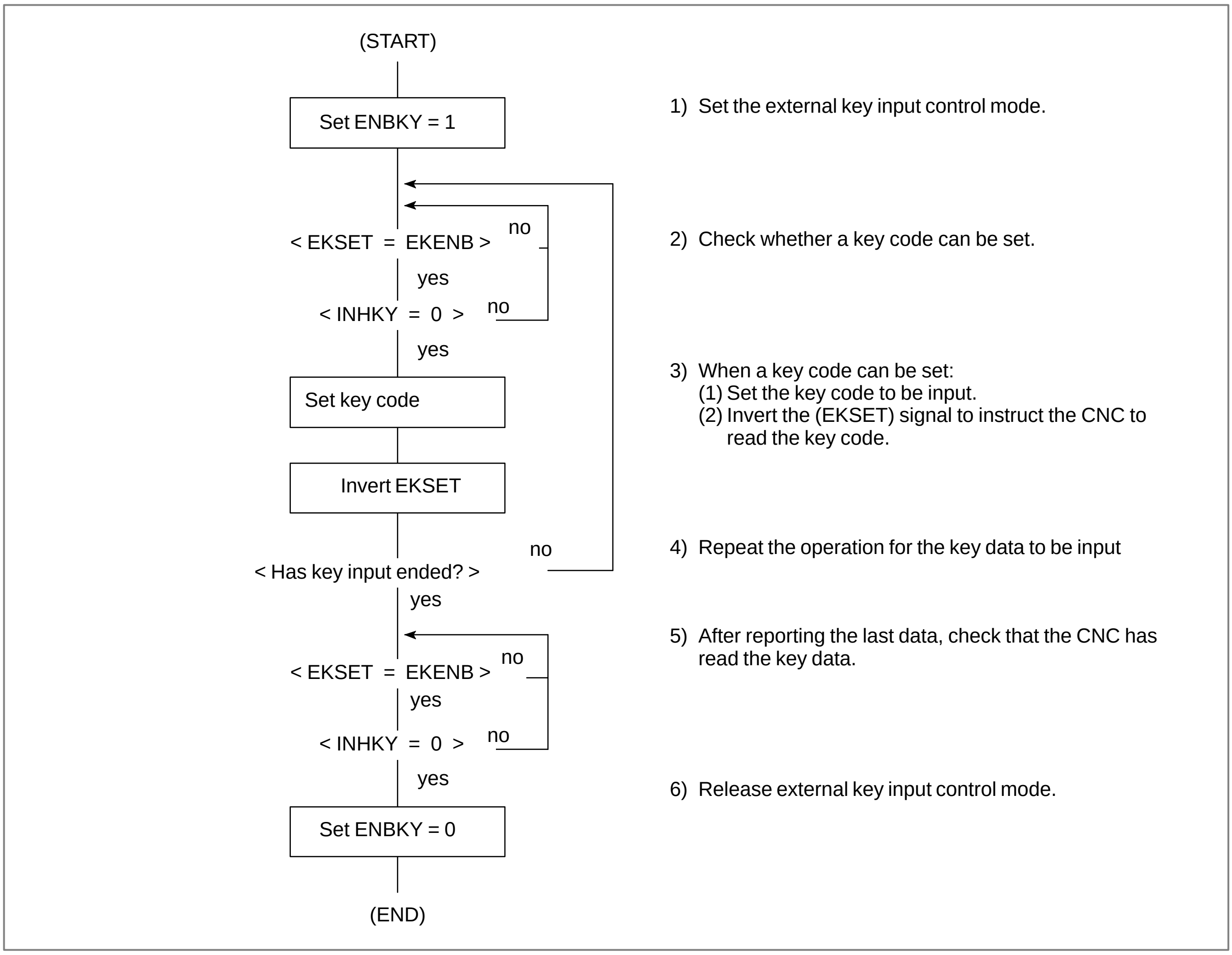
General

MDI key codes can be sent from the PMC to CNC by means of interface signals. This allows the CNC to be controlled in the same way as when the operator performs MDI key operation.

Control is realized by exchanging the following interface signals between the PMC and CNC:

Signal name	Abbreviation
External key input mode selection signal (input)	ENBKY
Key code signals (input)	EKC0 to EKC7
Key code read signal (input)	EKSET
Key code read completion signal (output)	EKENB
Key input disable signal (output)	INHKY
Program screen display mode signal (output)	PRGDPL

The processing flow in the PMC is shown below.



NOTE
Read processing is controlled by exclusive–ORing (XOR) the key code read signal (EKSET) with the read completion signal (EKENB). When the EKSET and EKENB signals differ in their logic, the CNC reads the input key code. Once reading has been completed, the CNC inverts the EKENB signal to match its logic with that of the EKSET signal.
In the PMC, on the other hand, a new key code cannot be set while the EKSET and EKENB signals differ in their logic.

Signals

External key input mode selection signal ENBKY <G066#1>

[Classification] Input signal

[Function] While this signal is turned on "1", external key input control is enabled. In external key input control mode, any MDI key operations are ignored.

Key code read signal EKSET <G066#7>

[Classification] Input signal

[Function] This signal instructs the CNC to read the input key code.

Key code signals EKC0 to EKC7 <G098>

[Classification] Input signal

[Function] These signals set an input key code. (See the MDI key code table.)

Key input disable signal INHKY<F053#0>

[Classification] Output signal

[Function] While this signal is "1", no key code is accepted in external key input control mode.

Program screen display mode signal PRGDPL <F053#1>

[Classification] Output signal

[Function] This signal is on “1” while the CNC is displaying a program screen.

Key code read completion signal EKENB <F053#7>

[Classification] Output signal

[Function] This signal reports that the CNC has read a key code.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066	EKSET						ENBKY	
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0

	#7	#6	#5	#4	#3	#2	#1	#0
F053	EKENB						PRGDPL	INHKY

MDI key code table

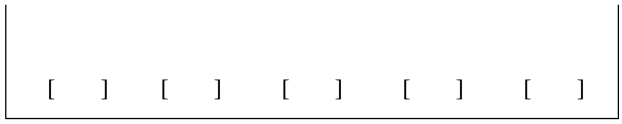
Codes in the table are given in hexadecimal. For example, A corresponds to 41H in hexadecimal. RESET corresponds to 90H in hexadecimal.

(Note 1) and (Note 2) are explained below:

NOTE

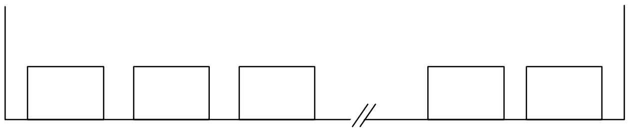
1 For the small keyboard, 0EDH is assigned to GRAPH CUSTOM .
For the standard keyboard, 0EDH is assigned to GRAPH .
0EEH is assigned to CUSTOM .

2 Handling of the soft keys
[F0] to [F9], [FR], and [FL] in the key code table are the key codes for the soft keys. They are associated with the MDI keys as shown below.
Key configuration for 7–soft key type LCD, or etc. : 5 keys + 2 keys ([F0] to [F4] and [FR], [FL])



[FL] [F4] [F3] [F2] [F1] [F0] [FR]

Key configuration for 12–soft key type LCD, or etc.: 10 keys + 2 keys ([F0] to [F9] and [FR], [FL])



[FL] [F9] [F8] [F7] ··· [F1] [F0] [FR]

MDI Key Code Table(00H–7FH)

	0	1	2	3	4	5	6	7
0			Space	0	@	P		
1				1	A	Q		
2				2	B	R		
3			#	3	C	S		
4				4	D	T		
5				5	E	U		
6			&	6	F	V		
7				7	G	W		
8			(	8	H	X		
9			)	9	I	Y		
A	; (EOB)		*		J	Z		
B			+		K	[		
C			,		L			
D			–	=	M	]		
E			.		N			
F			/	?	O			

MDI Key Code Table(80H–0FFH)

	8	9	A	B	C	D	E	F
0		RESET						[F0] (Note2)
1								[F1] (Note2)
2								[F2] (Note2)
3								[F3] (Note2)
4		INSERT						[F4] (Note2)
5		DELETE						[F5] (Note2)
6	CAN	ALTER						[F6] (Note2)
7								[F7] (Note2)
8	Cursor →	INPUT					POS	[F8] (Note2)
9	Cursor ←						PROG	[F9] (Note2)
A	Cursor ↓	HELP					OFFSET SET-TING	
B	Cursor ↑						SYSTEM	
C							MES-SAEG	
D							GRAPH (CUSTOM) (Note1)	
E	PAGE ↓						CUSTOM (Note1)	[FR] (Note2)
F	PAGE ↑						FAPT	[FL] (Note2)

15.6 DIRECT OPERATION BY PMC OR OPEN CNC

15.6.1 DNC Operation by the PMC or OPEN CNC (PC with HSSB Connection)

General

Activating memory operation in memory operation mode (MEM) with the direct operation select signal set to 1 enables machining (direct operation=DNC operation) while reading a program stored in the PMC or OPEN CNC.

Signal

Direct operation select signal DMMC <G042#7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program stored in the PMC and OPEN CNC.

[Operation] When this signal is set to “1”, the control unit operates as follows:

· When memory operation mode (MEM) is not selected, the control unit ignores this signal.

· When memory operation mode (MEM) is selected, the control unit selects direct operation mode and enables direct operation.

16 INTERFACE WITH THE POWER MATE CNC

16.1 FANUC SERVO MOTOR β SERIES I/O LINK OPTION MANUAL HANDLE INTERFACE (PERIPHERAL DEVICE CONTROL)

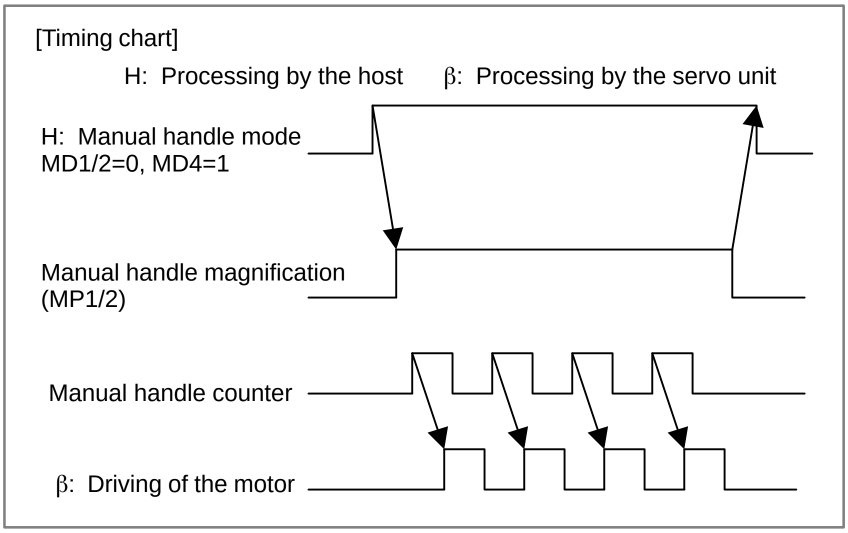
General

This function enables manual handle feed of the β servo unit with the manual pulse generator on the host side. A pulse signal generated by the manual pulse generator is sent to the β servo unit via the I/O Link. By parameter switching, a magnification can be applied to the pulse signal output from the manual pulse generator. This function can be used only with the peripheral device control interface.

Function details

After selecting the manual handle mode (MD1 (Y0#0)=0, MD2(Y0#1)=0, MD4(Y0#2)=1), the host sends a magnification (MP1(Y7#4), MP2(Y7#5)) for the manual pulse generator to the β servo unit to change the manual handle counter.

The β servo unit drives the motor by reading only a differential of the manual handle counter.



Signal (on the FS0*i*/0*i* Mate side)

Manual handle feed generator selection signals IOLBH2, IOLBH3 <G199#0, #1>

[Classification] Input signal

[Function] The signals select a manual pulse generator for feeding the β servo unit.

[Operation] A manual pulse generator for feeding the β servo unit is selected with the signals.

Input signal		Manual pulse generator for feeding the β servo unit
IOLBH3	**IOLBH2**	
0	0	First manual pulse generator
0	1	Second manual pulse generator
1	0	Third manual pulse generator
1	1	Use prohibited

NOTE
Do not switch between the manual pulse generators in the manual handle mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G199							IOLBH3	IOLBH2

Parameter (on the FS0*i*/0*i* Mate side)

	#7	#6	#5	#4	#3	#2	#1	#0
7103				IOB				

[Data type] Bit

IOB Manual handle feed of the (servo unit with the I/O Link manual pulse generator is:

0 : Disabled.
1 : Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
12330	GR7	GR6	GR5	GR4	GR3	GR2	GR1	GR0
12331	GRF	GRE	GRD	GRC	GRB	GRA	GR9	GR8

[Data type] Bit

GR0 When group 0 (channel 1) of the PMC is a Power Mate or I/O Link β, pulses of a manual pulse generator connected via the I/O Link are:

0 : Transferred to the target group.
1 : Not transferred to the target group.

GR1 When group 1 (channel 1) of the PMC is a Power Mate or I/O Link β, pulses of a manual pulse generator connected via the I/O Link are:

0 : Transferred to the target group.
1 : Not transferred to the target group.

: :
: :

GRE When group 14 (channel 1) of the PMC is a Power Mate or I/O Link β, pulses of a manual pulse generator connected via the I/O Link are:

0 : Transferred to the target group.
1 : Not transferred to the target group.

GRF When group 15 (channel 1) of the PMC is a Power Mate or I/O Link β, pulses of a manual pulse generator connected via the I/O Link are:

0 : Transferred to the target group.
1 : Not transferred to the target group.

NOTE
When a Power Mate is connected to the I/O Link, set this parameter to 1.

Signal (on the β servo unit side)

Mode selection signals MD1, MD2, MD4 <Yy+0#0, #1, #2>

[Classification] CNC (host) → Servo amplifier unit

[Function] The signals select an operation mode of the β servo unit.

[Operation] The manual handle feed mode of the β servo unit is selected with the signals.

Input signal			
MD4	MD2	MD1	
1	0	0	Handle feed

NOTE
1 Turn the manual pulse generator after placing it in the manual handle mode.
2 Do not switch between the modes during manual handle operation of the β servo unit.

Incremental feed signals MP1, MP2 <Yy+7#4, #5>

[Classification] CNC (host) → Servo amplifier unit

[Function] The signals select a magnification for the manual handle.

[Operation] The β servo unit drives the motor in the manual handle mode by the pulses obtained by multiplying the number of input pulses of the manual pulse generator by a magnification set by the signals.

Input signal		Amount of travel per division of the manual pulse generator
MP2	MP1	
0	0	1 user unit
0	1	10 user units
1	0	100 user units
1	1	(M/N) user units (M = Parameter No. 62, N = Parameter No. 63)

NOTE
1 The signals are valid when bit 5 (MP) of parameter No. 5 for the β servo unit is set to 1.
2 The signals are valid only in the manual handle mode.
3 The signals are used also as rapid traverse override signals. In the manual handle mode, the signals function as the incremental feed signals. In modes other than the manual handle mode, the signals function as the rapid traverse override signals. When the mode is switched from the manual handle mode to another mode, the setting of the signals need to be returned to the setting of the rapid traverse override signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
Yy+0						MD4	MD2	MD1

	#7	#6	#5	#4	#3	#2	#1	#0
Yy+7			MP2	MP1				

Parameter (on the β servo unit side)

	#7	#6	#5	#4	#3	#2	#1	#0
005			MP	IOH				

[Data type] Bit

IOH Manual handle feed via the I/O Link is:

0 : Disabled.
1 : Enabled.

> **CAUTION**
> When IOH is set to 1, be sure to set bit 6 (EXPLS) of parameter No. 3 on the β servo unit side to 0.

MP In manual handle feed, four–stage magnification setting using the MP1/MP2 signals for input manual handle pulses is:

0 : Disabled.
1 : Enabled.

17 PCMCIA ETHERNET FUNCTION

This chapter describes the specifications of the PCMCIA Ethernet function for Series 0*i*–C.

17.1 PCMCIA ETHERNET

The PCMCIA Ethernet function for CNC can be used the PCMCIA Ethernet card.

The PCMCIA Ethernet card, which is inserted into the memory card slot on the left side of the front liquid crystal display, is used for temporary communication.

NOTE

1 Use the PCMCIA Ethernet card for temporary communication only. Do not use the PCMCIA Ethernet card for routine communication.

2 The body of the PCMCIA Ethernet card is projected during operation. When using the PCMCIA Ethernet card, be careful not to damage the card by hitting the card with an object.
After using the PCMCIA Ethernet card, remove the card immediately to prevent the card from being damaged.

List of functions

With the PCMCIA Ethernet function, the following functions can be operated servo guide communication function and FANUC LADDER–III communication function.

17.2 SETTING THE PCMCIA ETHERNET FUNCTION

This section describes the setting of the parameters for the PCMCIA Ethernet function for the Series 0*i*–C.

17.2.1 Ethernet Parameter Setting Screen

On the Ethernet parameter setting screen, set the parameters.

Display

Procedure

1 Place the CNC in the MDI mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [ETHPRM] soft key. The Ethernet parameter setting screen appears. The Ethernet functions currently available are displayed

```
ETHERNET PARAMETER                O0000 N00000
  AVAILABLE ETHERNET

      PCMCIA CARD
      FAST ETHERNET BOARD









 MDI **** *** ***       10:00:00
(BOARD )(       )(       )(PCMCIA)(       )
```

The upper row displays ”PCMCIA CARD.”
The lower row displays the usable Ethernet option boards. When no option board is installed, no information is displayed.

5 By pressing the [PCMCIA] soft key, the parameters for the PCMCIA Ethernet card can be set.

6 By using the MDI keys and soft keys, enter and update data.

7 Switch the screen display with the page keys PAGE PAGE .

If data is already registered, the data is displayed.

```
ETHERNET PARAMETER(PCMCIA)   O0000 N00000
                             PAGE: 1/ 2

 MAC ADDRESS

(COMMON PARAMETER)
 IP ADDRESS                192.168.1.2
 SUBNET MASK               255.255.255.0
 ROUTER IP ADDRESS         192.168.1.254




>_
 MDI **** *** ***     10:00:00
(      )(      )(INPUT )(RETURN)(      )
```

```
ETHERNET PARAMETER(PCMCIA)   O0000 N00000
                             PAGE: 2/ 2
(FOCAS1/ETHERNET)
 PORT NUMBER(TCP)                    8193

 PORT NUMBER(UDP)                    8192

 TIME INTERVAL                        100




>_
 MDI **** *** ***     10:00:00
(      )(      )(INPUT )(RETURN)(      )
```

Display item and setting items

Display item

The item related to the PCMCIA Ethernet function is displayed.

Item	Description
MAC ADDRESS	PCMCIA Ethernet card MAC address

Setting items

Set the TCP/IP–related items of the PCMCIA Ethernet.

Item	Description
IP ADDRESS	Specify the IP address of the PCMCIA Ethernet. (Example of specification format: ”192.168.1.1”)
SUBNET MASK	Specify a mask address for the IP addresses of the net-work. (Example of specification format: ”255.255.255.0”)
ROUTER IP ADDRESS	Specify the IP address of the router. Specify this item when the network contains a router. (Example of specification format: ”192.168.1.254”)
PORT NUMBER (TCP)	Specify a port number. The valid input range is 5001 to 65535.
PORT NUMBER (UDP)	Set to 0.
TIME INTERVAL	Set to 0.

NOTE

The parameters for the PCMCIA Ethernet card are set to the following default values before shipment:

IP address:	192.168.1.1
Subnet mask:	255.255.255.0
Router IP address:	None
TCP port number:	8193
UDP port number:	0
Time interval:	0

17.2.2 Communication Parameter Input Method

This subsection describes the method of parameter input when the PCMCIA Ethernet function for the Series 0*i*–C is used.

Basic method of data input

The basic method of data input is described below, using an example of IP address input.

Procedure

1 Place the CNC in the MDI mode.

2 Display the Ethernet parameter screen.

3 Move the cursor to a desired input item with cursor keys.

4 Type data with MDI keys.

5 Press the [INPUT] soft key or the function key INPUT to enter the data.

> **NOTE**
> When deleting numeric data already set, enter 0. When deleting character data already set, enter SP (space).

Example) Setting 192.168.1.2 as IP address data

(a) Move the cursor to the item of IP address.

```
ETHERNET PARAMETER(PCMCIA)   O0000 N00000
                             PAGE: 1/ 2

 MAC ADDRESS                 00000E628E87

(COMMON PARAMETER)
 IP ADDRESS            192. 168. 1. 1
 SUBNET MASK           255. 255. 255. 0
 ROUTER IP ADDRESS




>_
 MDI **** *** ***      10:00:00
(      )(      )(INPUT )(RETURN)(      )
```

(b) Type 192.168.1.2 with the MDI keys.

```
ETHERNET PARAMETER(PCMCIA)   O0000 N00000
                             PAGE: 1/ 2

 MAC ADDRESS                  00000E628E87

(COMMON PARAMETER)
 IP ADDRESS                192. 168. 1. 1
 SUBNET MASK               255. 255. 255. 0
 ROUTER IP ADDRESS




>192. 168. 1. 2_
 MDI **** *** ***       10:00:00
(        )(        )(INPUT )(RETURN)(        )
```

(c) Press the [INPUT] soft key or the function key INPUT to enter the data.

This stores the parameter in the nonvolatile memory of the CNC.

```
ETHERNET PARAMETER(PCMCIA)   O0000 N00000
                             PAGE: 1/ 2

 MAC ADDRESS                  00000E628E87

(COMMON PARAMETER)
 IP ADDRESS                192. 168. 1. 1
 SUBNET MASK               255. 255. 255. 0
 ROUTER IP ADDRESS




>192. 168. 1. 2_
 MDI **** *** ***       10:00:00
(        )(        )(INPUT )(RETURN)(        )
```

NOTE

Turn on the power again so that you should make a changed parameter effective.

Or, push soft key [RESET] on the maintenance screen of PCMCIA Ethernet.

17.3 PCMCIA ETHERNET ERROR MESSAGE SCREEN

If an error occurs with the PCMCIA Ethernet function, the error message screen for the PCMCIA Ethernet function displays an error message.

Display

Procedure

1 Press the function key MESSAGE .

2 Press the continuous menu key at the right end of the soft key display.

3 Press the [ETHLOG] soft key. The Ethernet log screen appears. The Ethernet functions currently available are displayed.

```
ETHERNET LOG
  AVAILABLE ETHERNET

      PCMCIA CARD
      FAST ETHERNET BOARD










 MDI **** *** ***      10:00:00
(BOARD )(       )(       )(PCMCIA)(       )
```

The upper row displays ”PCMCIA CARD.”
The lower row displays the usable Ethernet option boards. When no option board is installed, no information is displayed.

4 By pressing the [PCMCIA] soft key, the error message screen for the PCMCIA Ethernet function can be displayed.

5 Switch the screen display with the page keys PAGE↑ PAGE↓ .

```
EMB_ETH MASTER CTRL LOG (1/2)
 *KIND OF LOG :ERROR          PAGE: 1/10
 Err Reset Driver [1]           11100400


 MDI **** *** ***      10:00:00
(RETURN)(      )(      )(      )(      )
```

[Tip]
The latest error message is displayed at the top of the screen. To the right of an error message, the date and time data of the occurrence of the error is displayed. The format of date and time data is ddhhmmss where dd represents a day, hh represents hours, mm represents minutes, and ss represents seconds.

Configuration

The PCMCIA Ethernet log screen consists of the screens below.

(1) EMB_ETH MASTER CTRL LOG screen (2 screens)
Log screen used to set the parameters of the PCMCIA Ethernet function and display error messages at the time of PCMCIA Ethernet initialization

(2) EMB_ETH FOCAS1/ETHER LOG screen (2 screens)
Log screen that displays error messages related to communication with the servo guide.

(3) EMB_ETH PMC LOG screen (2 screens)
Log screen that displays error messages related to communication with the FANUC LADDER–III.

17.4 PCMCIA ETHERNET MAINTENANCE SCREEN

With the PCMCIA Ethernet function, a dedicated maintenance screen is available.
The maintenance screen enables operations to be checked when the PCMCIA Ethernet function operates abnormally.

Display

Procedure

1 Press the function key SYSTEM .

2 Press the continuous menu key at the right end of the soft key display.

3 Press the [ETHMNT] soft key. The Ethernet maintenance screen appears. The Ethernet functions currently available are displayed.

```
ETHERNET MAINTENANCE                O0000 N00000
  AVAILABLE ETHERNET

      PCMCIA CARD
      FAST ETHERNET BOARD









 MDI **** *** ***        10:00:00
(BOARD )(       )(       )(       )(PCMCIA)
```

The upper row displays ”PCMCIA CARD.”
The lower row displays the usable Ethernet option boards. When no option board is installed, no information is displayed.

4 By pressing the [PCMCIA] soft key, the maintenance screen for the PCMCIA Ethernet function can be displayed.

5 Switch the screen display with the page keys PAGE↑ PAGE↓ .

6 The screen below is used to check the state of the communication cable and whether a communication destination exists.
Enter the IP address of a communication destination through MDI keys, then press the [PING] soft key. Communication is performed three times with the specified communication destination, and the results are displayed.

```
ETHERNET MAINTENANCE               O0000 N00000
                                   PAGE: 1/ 5
(CONNECT STATUS)
  Reply from 192.168.1.123.
  Reply from 192.168.1.123.
  Reply from 192.168.1.123.
  IP address Error(192.168..100)
  Request Timed out 192.168.1.100.
  Request Timed out 192.168.1.100.
  Request Timed out 192.168.1.100.



>_
 MDI **** *** ***      10:00:00
(RETURN)(      )(      )(      )( PING )
```

Messages displayed:

Reply from IP–address

This message indicates that a response was received from the specified communication destination and that the specified communication destination exists on the network.

Request Timed out IP–address

This message indicates that no response was received from the specified communication destination and that the specified communication destination does not exist on the network.
Check if the power to the communication destination equipment is turned on. Check also the parameter settings and network installation for errors.

IP address Error (IP–address)

The specified IP address is incorrect. Check the entered IP address.

7 The screen below is used to check the communication state of the PCMCIA Ethernet function and the error detection count of the Ethernet controller.

```
ETHERNET MAINTENANCE          O0000 N00000
                              PAGE: 2/ 5
(NETWORK STATUS)
 BAUDRATE          10MBPS /  HALF DUPLEX
 NETWORK DEVICE            PCMCIA CARD

(NETWORK STATUS : SEND)
 COLLISION                               0
 CARRIER SENSE LOST                      0
 DELAY OVER                              0
 UNDERRUN ERROR                          0
 SEND PARITY ERROR                       0


 MDI **** *** ***     10:00:00
(RETURN)(      )(RESET )(CLEAR )(       )
```

```
ETHERNET MAINTENANCE          O0000 N00000
                              PAGE: 3/ 5
(NETWORK STATUS)
 BAUDRATE          10MBPS /  HALF DUPLEX
 NETWORK DEVICE            PCMCIA CARD

(NETWORK STATUS : RECEIVE)
 ALIGNMENT ERROR                         0
 CRC ERROR                               0
 OVERFLOW                                0
 FRAME LENGTH ERROR                      0
 RECEIVE PARITY ERROR                    0


 MDI **** *** ***     10:00:00
(RETURN)(      )(RESET )(CLEAR )(       )
```

The screen consists of two pages: one page for an error detection count for transmission, and the other for an error detection count for reception.

By pressing the [CLEAR] soft key, the error detection counters for transmission and reception can be cleared to 0.

By pressing the [RESET] soft key, the current communication device can be initialized and communication can be performed from the initial state. Use this key to reset communication based on the PCMCIA Ethernet function.

8 The screen below is used to check the state of each task of the PCMCIA Ethernet function.

```
ETHERNET MAINTENANCE              O0000 N00000
                                  PAGE: 4/ 5
(TASK STATUS)
  MASTER CTRL  : WP
  FOCAS1 #0    : C
  FOCAS1 #1    : CCCCC
  FOCAS1 #2    : CCCCC
  UDP          : X
  PMC          : C



 MDI **** *** ***      10:00:00
(RETURN)(      )(      )(      )(      )
```

	Symbol	Meaning
MASTER CTRL	E	Ethernet controller being initialized
	D	Data being processed(NOTE)
	W	Waiting for data processing(NOTE)
	P	Waiting for parameter setting
	S	Parameters being set
FOCAS1 #0	X	Waiting for completion of Ethernet controller initialization
	E	Being activated
	C	Waiting for connection from the personal computer
	O	Connection being processed
	N	FOCAS1/Ethernet execution disabled
FOCAS1 #1,#2	X	Waiting for completion of Ethernet controller initialization
	C	Waiting for connection from the personal computer
	D	Data being processed(NOTE)
	W	Waiting for data processing(NOTE)
UDP	X	Not executed yet. Waiting for completion of Ethernet controller initialization.
	E	Being activated
	D	Data being processed(NOTE)
	W	Waiting for data processing(NOTE)

	Symbol	Meaning
PMC	X	Waiting for completion of Ethernet controller initialization
	D	Data being processed(NOTE)
	W	Waiting for data processing(NOTE)

NOTE

A state change occurs between the states ”Data being processed” and ”Waiting for data processing” even when communication is not performed actually.

17.5 TROUBLESHOOTING

This section describes troubleshooting and check items associated with the PCMCIA Ethernet function.

17.5.1 Check Items Related to Connection

1) Is an STP cable used for connection between the hub and PCMCIA Ethernet?
2) Is the STP cable connected correctly?
   In general, a straight cable is used for connection between the hub and communication device.
   For direct connection to the personal computer without using the hub, a cross cable is used.
3) Is the power to the hub turned on?
4) The PCMCIA Ethernet card is used only with 10BASE–T. Is a hub for 10BASE–T used when the PCMCIA Ethernet card is used?
5) Is the LED (for link display) of the connected hub turned on? (Some hubs are not provided with a link LED.)
   The LED is not turned on when the hub is not connected with the PCMCIA Ethernet or the power to the CNC is not turned on.

17.5.2 Checking the Setting of Each Parameter

This subsection describes how to check the minimum settings required for communication.

Checking the settings on the PCMCIA Ethernet side

1) Is the MAC address of the PCMCIA Ethernet displayed?
   - A unique MAC address is assigned to each PCMCIA Ethernet card. When a PCMCIA Ethernet card is selected and inserted, MAC address display is provided.
2) Is a correct IP address set?
   - Check if an IP address already specified for another device is set.
3) Is a correct subnet mask set?
   - The subnet mask setting must match the subnet mask setting on the communication destination device.
4) Is a correct router IP address set when communication via a router is performed?

Checking the settings on the personal computer side

1) Is a correct IP address set?
   - Check if an IP address already specified for another device is set.
2) Is a correct subnet mask set?
   - The subnet mask setting must match the subnet mask setting on the communication destination device.
3) Is a correct router IP address set when communication via a router is performed?

17.5.3 Checking Communication

This subsection describes how to check the state of communication between the CNC and personal computer.

Checking the connection status and settings

If communication with the CNC is not satisfactory or fails from time to time, check the communication link by using the method described below. The ping command is used to check communication.

Checking from the PCMCIA Ethernet side

See Item 6 of Section 17.4, ”PCMCIA ETHERNET MAINTENANCE SCREEN”.

If no response is received from the remote device, the cause is considered to be a hardware connection error and/or software setting error. Check the hardware connection and software settings.

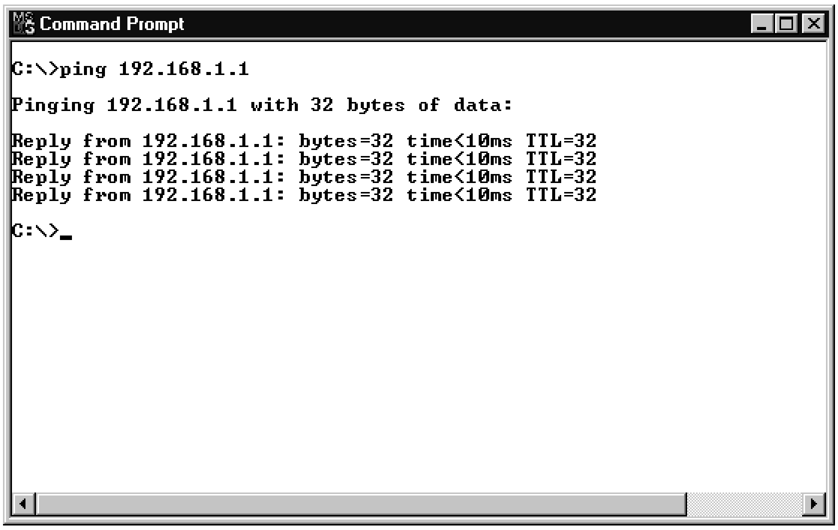
Checking from the personal computer side

An example where a personal computer (OS: Windows 2000) is used is described below.

Method of checking:

Open the command prompt, then enter ”ping NC–IP–address”. A normal connection has been established if a response is received.
The example below supposes that the IP address of the CNC is 192.168.1.1.

1) When a response is received (normal)



2) When no response is received (error)

```
Command Prompt

C:\>ping 192.168.1.1

Pinging 192.168.1.1 with 32 bytes of data:

Request timed out.
Request timed out.
Request timed out.
Request timed out.

C:\>_
```

If no response is received from the CNC, the cause is considered to be a hardware connection error and/or software setting error. Check the hardware connection and software settings.

Checking the influence of noise

The method of checking communication errors caused by noise is described below.

The ping command is used for this checking as well.

The –t option of the ping command is used. Until the ”Ctrl + C” keys are pressed simultaneously, ping packets are transmitted.

```
Command Prompt

C:\>ping -t 192.168.1.1

Pinging 192.168.1.1 with 32 bytes of data:

Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
Reply from 192.168.1.1: bytes=32 time<10ms TTL=32
^C
C:\>^C_
```

1. Influence of noise from peripheral equipment (device)

   1) Turn on the power to the machine with the PCMCIA Ethernet function for which a noise influence check is to be made, and ensure that communication is enabled.

   2) Press the emergency stop button of the machine to turn off servo/spindle amplifier activation, then issue a ping command from the personal computer.

3) Count the number of lost packets (to which no response is returned).

   If lost packets occur in this state, there is probably an influence of noise from peripheral equipment.

   Action: Locate the noise source and recheck the cabling to eliminate the influence of noise.

2. Influence of noise from the installed machine

   1) Next, release the emergency stop state of the machine to turn on servo/spindle amplifier activation, then issue another ping command from the personal computer.

   2) Count the number of lost packets.

      If this number is greater than the number of Item 1 above, the cause is considered to be noise generated by the machine itself. In general, the grounding of the machine or the grounding of the communication destination is defective.

      Action: Check the grounding of the machine and the communication destination, and insulate the machine from the communication backbone.

17.6 ERROR MESSAGES

If an error occurs with the embedded Ethernet function, the log screen of the PCMCIA Ethernet function displays an error message.
This section describes error messages displayed on the log screen.
The major error messages are described below.

If an error occurs, display the log screen and check the error message to identify the cause of the error.
Multiple error messages may be displayed for an error. So, check the display times of error messages.

17.6.1 EMB_ETH MASTER CTRL LOG Screen

`OWN IP ADDRESS IS NOTHING`
The IP address of the local node is not set. Set an IP address correctly.

`OWN IP ADDRESS(???) IS INVALID`
The setting (???) of the IP address of the local node is incorrect. Correct the IP address.

`SUBNET MASK IS NOTHING`
The subnet mask of the local node is not set. Set a correct subnet mask.

`SUBNET MASK(???) IS INVALID`
The setting (???) of the subnet mask of the local node is incorrect. Correct the subnet mask.

`ROUTER IP ADDRESS(???) IS INVALID`
The setting (???) of the IP address of the router is incorrect. Correct the IP address of the router.

`TCP PORT NUMBER(???) IS INVALID`
The setting (???) of the TCP port number is incorrect. Correct the TCP port number.

`UDP PORT NUMBER(???) IS INVALID`
The setting (???) of the UDP port number is incorrect. Correct the UDP port number.

`UDP INTERVAL TIME(???) IS INVALID`
The setting (???) of the time interval for UDP transmission is incorrect. Correct the time interval.

17.6.2 EMB_ETH FOCAS1/ETHER LOG Screen

`TCP PORT NUMBER(???) IS INVALID`

The setting (???) of the TCP port number is incorrect. Correct the TCP port number.

`Illegal Broadcast IP ADDRESS`

The broadcast address for UDP transmission is incorrect. Correct the subnet mask and IP address of the local node.

`Illegal Power-on Date or Time`

The current time setting of the CNC is incorrect. Correct the clock of the CNC.

`ALL TASKS(C1) ARE BUSY`

The FOCAS1/Ethernet function, on which the servo guide communication function is based, is already communicating with five applications. Terminate unnecessary communication applications on the personal computer.

If the cable is disconnected before communication is completed, the PCMCIA Ethernet may need to be reset and initialized.

`Err accept() [???]`

An error occurred when a connection request from the personal computer is being awaited. The error code is [???]. This error message is output, for example, when the PCMCIA Ethernet is reset.

`Err recv() [???]`

An error occurred during data reception. The error code is [???].

This error message is output, for example, when the PCMCIA Ethernet is reset before communication is closed.

17.7 GLOSSARY FOR ETHERNET

This section briefly describes Ethernet–related terms.
The descriptions below provide minimum information only. For further information, refer to relevant publications available on the market.

TCP/IP

For Ethernet–based communication, the TCP/IP (Transmission Control Protocol/Internet Protocol) protocol is generally used. A protocol is a set of rules used to ensure smooth communication between communication devices connected via a transmission line. The TCP/IP protocol is a part of the hierarchical structure consisting of the protocols and services indicated below.

Hierarchy	Protocol/network service
Application layer	User services such as FTP
Transport layer	Protocols such as TCP and UDP
Network layer	Protocols such as IP and ICMP
Data link layer	Protocols such as ARP and RARP
MAC layer	
Physical layer	Hardware such as cables and devices

In general, the TCP/IP protocol is a generic term that represents the protocols installed in the transport layer and network layer.

IP address (INET address)

With TCP/IP, an address referred to as an IP address (INET address) is used to identify a specified communication device among the communication devices connected via Ethernet. So, for communication using TCP/IP, each communication device connected to Ethernet must have a unique IP address assigned.

An IP address is four octets (bytes) long. Usually, an IP address is represented by four 8–bit (octet or byte) fields separated by a period from each other. Each octet can have a value from 0 to 255.

An IP address consists of the address of the network to which the communication device is connected, and the host address of the communication device. Networks are classified into three classes by group size: class A, class B, and class C.

	First octet value	Network address section	Host address section
Class A	0 to 127	xxx.xxx.xxx.xxx	xxx.xxx.xxx.xxx
Class B	128 to 191	xxx.xxx.xxx.xxx	xxx.xxx.xxx.xxx
Class C	192 to 223	xxx.xxx.xxx.xxx	xxx.xxx.xxx.xxx

(A hatched portion indicates the section of each address.)

If a network supports no more than 255 communication devices, class C is generally used.

The IP addresses of all communication devices on one network have the same network address, and only the host address of each communication device is unique on the network.

An IP address with its network address and host address all set to 0 or 255 is unusable.

IP addresses are internationally managed systematically. This means that before an IP address can be used, the IP address must be obtained formally from the international organization.

If the network used by a user is a local network closed within the user's environment (not connected to an outside network), unique IP addresses may be set freely under the control and responsibility of the user. For a local network, the following network addresses can be used without formal registration: 1 address (10) for class A, 16 addresses (172.16 to 172.31) for class B, and 256 addresses (192.168.0 to 192.168.255) for class C. So, it is recommended that IP addresses with these network addresses be used for a local network.

Subnet mask (mask address)

Mask address for indicating the network address section of an IP address.
For a network of class A, specify 255.0.0.0.
For a network of class B, specify 255.255.0.0.
For a network of class C, specify 255.255.255.0.

MAC address (Ethernet address)

A MAC address is assigned to the Ethernet control board of each communication device, and is used to identify each communication device on the MAC layer (lower part of the data link layer). A unique address obtained from an international organization is used so that no address duplication occurs among Ethernet control board suppliers.

Port number

The port number is a 16–bit integer used to associate the transport layer (TCP or UDP) of TCP/IP and a process of the application layer. Port numbers from 0 to about 8000 are called well–known port numbers and assigned to standard applications (such as Telnet and FTP). The assignment of port numbers is described in Assigned Numbers [RFC1340].
When using the FOCAS1/Ethernet function and DNC1/Ethernet function, assign port numbers other than the well–known port numbers.

Broadcast

Transmitting a message to all nodes in the same segment

Client

Device or application that requests a service

Server

Device or application that provides a service

18 TROUBLE DIAGNOSIS

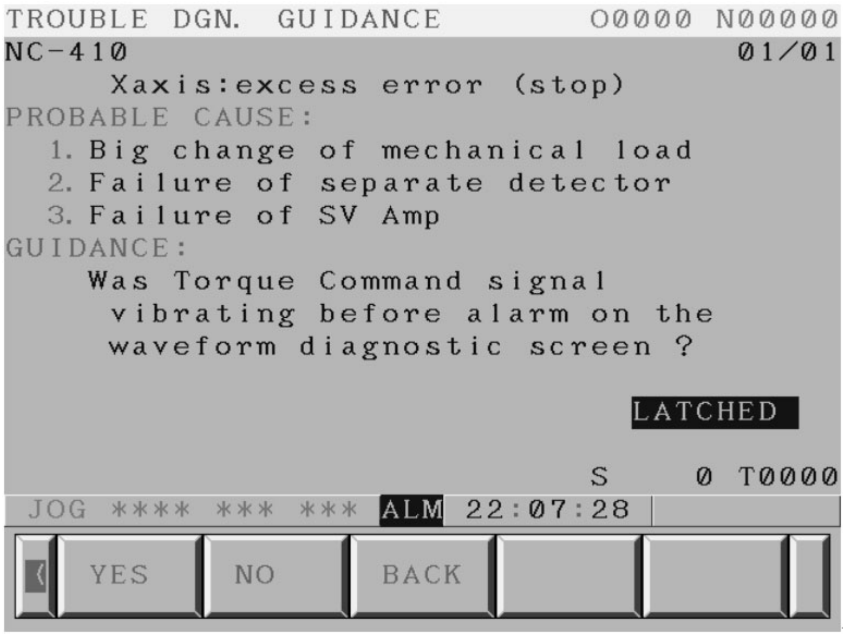
18.1 TROBLE DIAGNOSIS

18.1.1 Outline

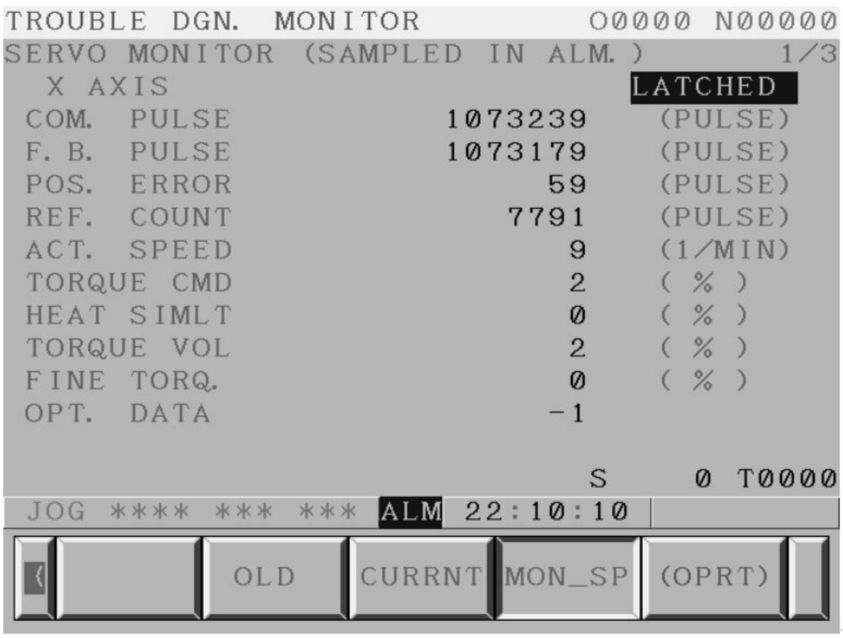
Investigating the cause of Servo/Spindle/CNC alarms becomes easier by diagnosis according to the guidance message.
And when the thermal simulation or disturbance torque of servo axis exceeds the trouble forecast level, a trouble forecast signal can be output.

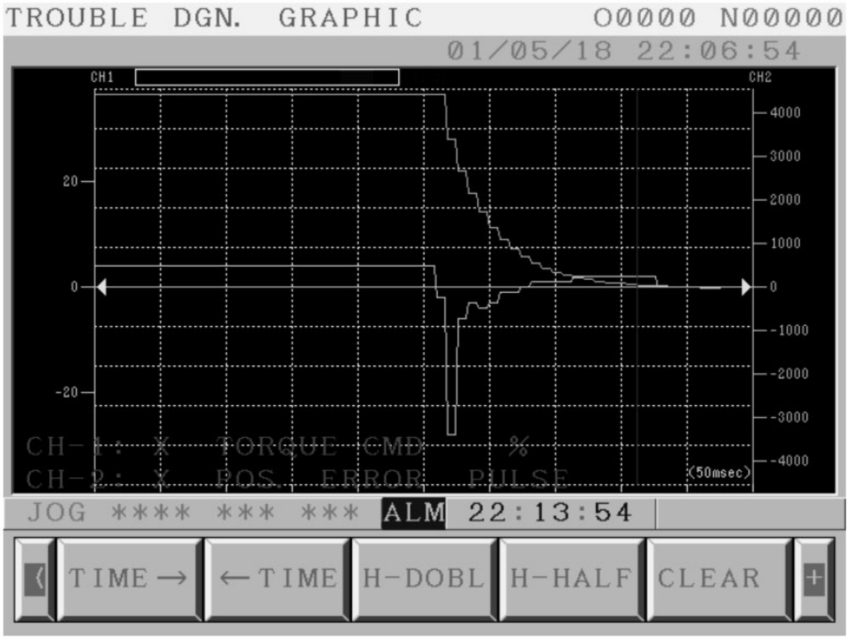
Step of diagnosis

1 Answer the question of the guidance message in the trouble diagnosis guidance screen by pushing soft keys [YES]/[NO].

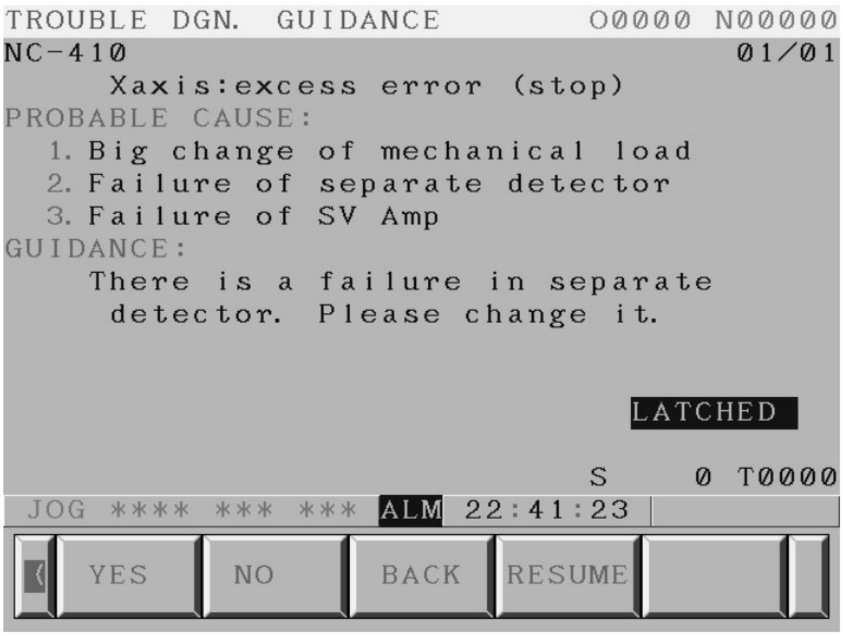


2 See the servo/spindle monitor information in the trouble diagnosis monitor screen and waveform of servo/spindle in the trouble diagnosis graphic screen in case of need according to the guidance message.





3 Finaly, the guidance message in the trouble diagnosis guidance screen shows the probable cause of alarm and the method to remove the problem.



18.1.2 Trouble Diagnosis Guidance Screen

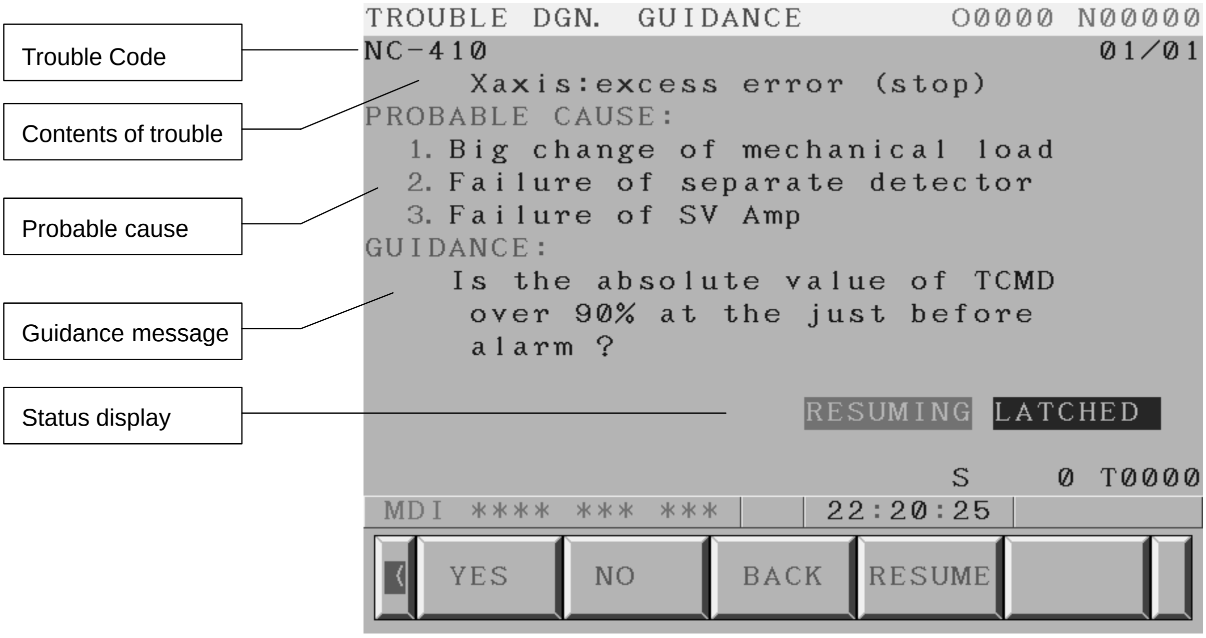
The trouble diagnosis guidance screen displays the guidance message to investigate the cause of an alarm.

Display

Display procedure

1 Press function key [Message].

2 Press the continuous menu key [>] and press soft key [GUIDE].



Contents of Display

Trouble Code: Code for identifing alarm.

Contents of Trouble: Alarm Meassage.

Probable Cause: Probable Cause of alarm.

Guidance message: Question to find the cause or answer to remove trouble is displayed.

Status display

[RESUMING]: When press soft key [RESUME] , the guidance message which was displayed before CNC power turned off is displayed again. Then [RESUMING] appears.

[LATCHED]/[SAMPLING]: [LATCHED] shows that servo/sprindle monitor information is memorized. Alarm No. in the guidance screen and alarm no. which occurred when servo/sprindle monitor data was memorized may be different. In this case the diagnosis may not done correctly. And then [LATCHED] blinks.
[SAMPLING] shows that servo/sprindle monitor information is not memorized yet. Servo/spindle monitor information can be referred in Trouble Diagnosis Monitor Screen.

Operation

Change of Guidance

[YES]/[NO]: Check contents of guidance message, and, answer by pressing soft key [YES] or [NO]. Then the next guidance message is displayed.
In some cases CNC automatically checks and judges contents of guidance. In this case the next guidance message is automatically displayed.
Automatic diagnosis is not done in case that CNC power turns off once after servo /spindle monitor information is memorized.

[BACK]: Guidance message returns back 1 step.
It is possible to trace back the guidance message when [YES]/[NO] is pressed by mistake.

[RESUME]: Guidance message which was displayed before CNC power tured off is displayed agein.If guidance is not done once, soft key [RESUME] does not appear. It is not possible to go back to the step before the point where soft key [BACK] is pressed.
Status display [RESUMING] is displayed during the guidance that starts by pressing [RESUME].

Change of alarm

When several alarms occur in same time, pressing page key [Page↓]/[Page↑] can select the guidance message.

18.1.3 Trouble Diagnosis Monitor Screen

Trouble Diagnosis Monitor Screen memorizes and displays servo/spindle monitor information for investigating servo/spindle alarm.

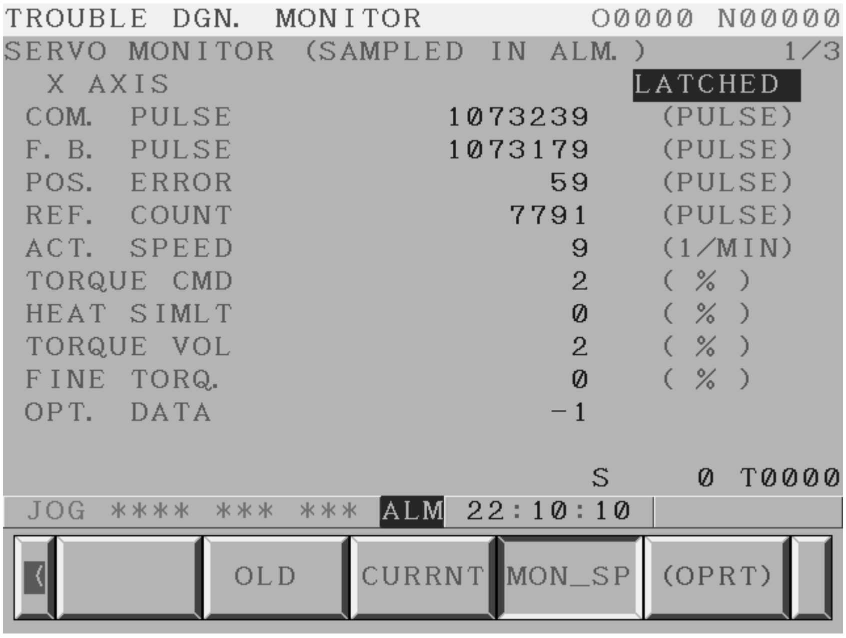
Three kinds of data, ”Data when the alarm occurs”, ”Data just before the alarm occurs”, ”Current data” can be selected and displayed.

Display

Display procedure

1 Press function key [Message].

2 Press the continuous menu key [>] and press soft key [MONIT].



Sample of displaying data of X axis (servo) when the alarm occurred.

Data displayed in Monitor Screen

Displayed data in Monitor screen is showed below.
Display range is the range which can be displayed on screen and not capacity of system.

1) Data of servo motor

Data (Unit)	Data type	Display range	Required parameter
Accumulated command pulse (pulse)	2 Word	±99999999	
Accumulated feedback pulse (pulse)	2 Word	±99999999	
Position error (pulse)	2 Word	±99999999	
Reference counter (pulse)	2 Word	±99999999	
Actual speed (min^{-1})	1 Word	–32768 to +32768	
Command current (%)	1 Word	±300	
Thermal simulation data (%)	1 Word	0 to 300	
Torque (%)	1 Word	±300	
Disturbance torque (%) Note 1)	1 Word	±14564	No.2104 (Cut), No2142 (Rapid)
Optional data Note 2)	1 Word	–32768 to +32767	
Optional data 2 Note 2)	1 Word	–32768 to +32768	
R–phase current (%)	1 Word	±300	
Effective current (%)	1 Word	±300	
Pulse coder AMR data	1 Word	0 to 255	
Internal neglect counter	1 Word	0 to +32767	
External neglect counter	1 Word	0 to +32767	
Internal correction counter	1 Word	0 to +32767	
External correction counter	1 Word	0 to +32767	
V–ready off information	1 Word	–32768 to +32768	

NOTE

1 Disturbance torque (%)
Set the following parameters to display the ratio of load torque to unexpected disturbance detection threshold.
– No.2104 (unexpected disturbance detection threshold for cutting)
– No.2142 (unexpected disturbance detection threshold for rapid traverse)
If these parameters are set to 0, load torque is displayed as it is. In this case, ±7282 means the max torque of servo moter.

2 Optional data 1, Optional data 2
Optional data 1 and Optional data 2 are used for maintenance by FANUC serviceman.

2) Data of spindle motor

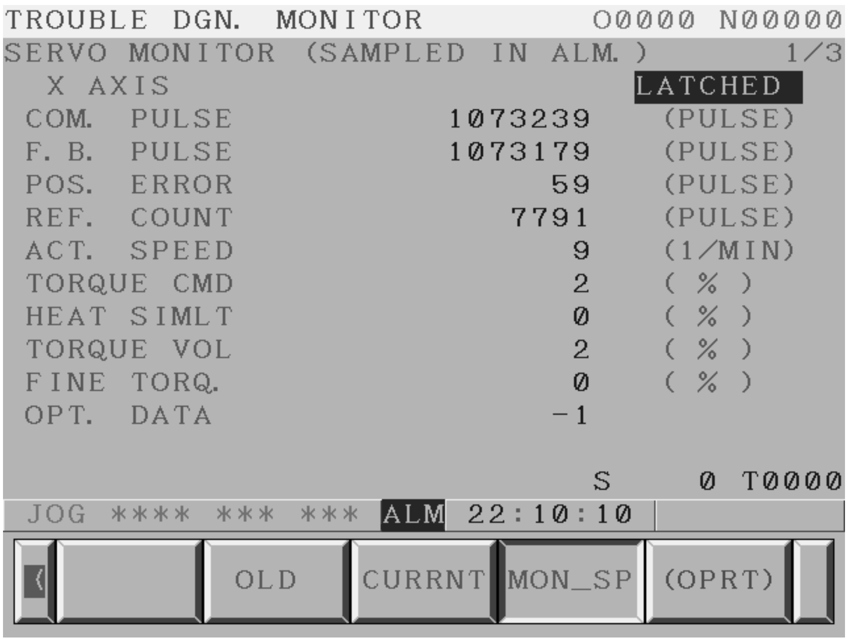
Data (Unit)	Data type	Display range	Required parameter
Operation mode	Character	*****	
Gear select command	Character	*****	
Command pulse (pulse)	2 Word	±99999999	
Command speed (min^{-1}) Note 1)	1 Word	–32768 to +32767	No.4020 (Main)/No.4196 (Sub)
Spindle speed (min^{-1})	2 Word	±99999999	
Motor speed (min^{-1}) Note 1)	1 Word	–32768 to +32767	No.4020 (Main)/No.4196 (Sub)
Load meter (%)	1 Word	0 to 300	
Position error (pulse)	2 Word	±99999999	
Synchronization error (pulse)	2 Word	±99999999	
Input signals	Character	*****	
Output signals	Character	*****	

NOTE

1 Command speed (min^{-1}), Motor speed (min^{-1})
Set the following parameters to display Command speed (min^{-1}) and Motor speed.
– No.4020 (Maximum motor speed for Main spindle)
– No.4196 (Maximum motor speed for Sub spindle) (When Spindle switch function is used.)

Operation

Display of Servo monitor data



Servo monitor information is switched by pressing soft key [NEW]/[OLD]/[CURRNT].
Soft key [NEW] and [OLD] is displayed alternately.

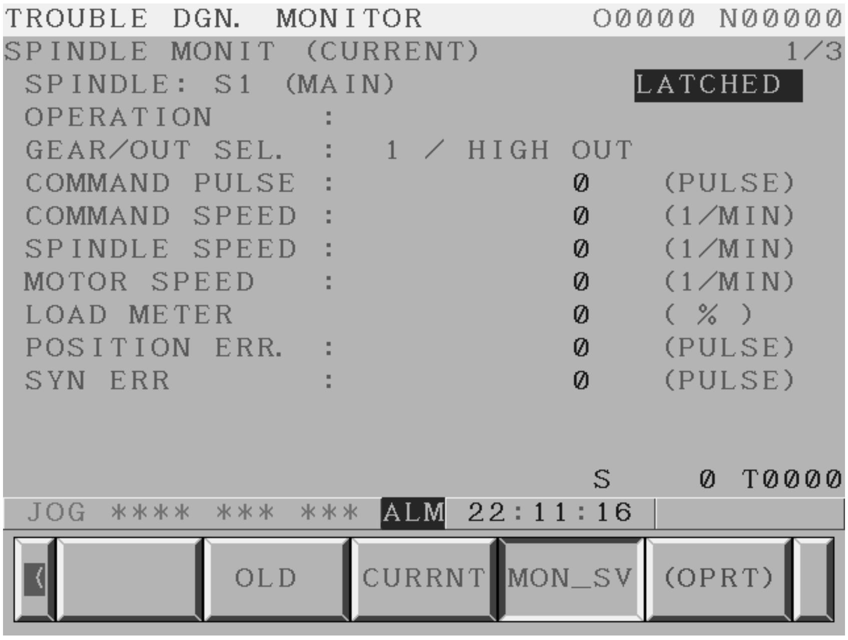
[NEW]: Data when the alarm occurs

[OLD]: Data just before the alarm occurs

[CURRNT]: Current data

[MON_SP]: Spindle monitor information is displayed.

Display of Spindle monitor information



Spindle monitor information is switched by pressing soft key [NEW]/[OLD]/[CURRNT].
Soft key [NEW] and [OLD] is displayed alternately.

[NEW]: Data when the alarm occurs

[OLD]: Data just before the alarm occurs

[CURRNT]: Current data

[MON_SV]: Servo monitor information is displayed.

Change of displayed axis

Displayed axis is switched by pressing cursor key [←]/[→].

Clear of memorized data

Soft key [CLEAR] appears by pressing soft key [(OPER)]. By pressing soft keys [CLEAR] and [EXEC], ”Data when the alarm occurs” and ”Data just before the alarm occurs” are cleared. And status display ”LATCHED” is altered to ”SAMPLING”.

When servo/spindle alarm occurs in ”SAMPLING” status, ”Data when the alarm occurs” and ”Data just before the alarm occurs” are memorized, and status display ”SAMPLING” is altered to ”LATCHED”.

When clear operation is done in either servo monitor screen or spindle monitor screen, both of servo and spindle information are cleared. And Display data of Trouble Diagnosis Graphic screen is also cleared.

18.1.4 Trouble Diagnosis Parameter Screen

Data type, data unit and trouble forecast level in the trouble diagnosis graphic screen are set in Trouble Diagnosis Parameter Screen.

Display

Display procedure

1 Press function key [Message].

2 Press the continuous menu key [>] and press soft key [W.GRPH].

3 Press soft key [W.PRM].

Contents of Display

AXIS: AXIS for Waveform display. Manual setting is available. In some cases AXIS data is set automatically for diagnosis in Trouble Diagnosis Guidance Screen.

DATA KIND: Data kind for Wave form display. Manual setting is available. In some cases data kind is set automatically for diagnosis in Trouble Diagnosis Guidance Screen.

DATA UNIT: Unit of display data. This data is set automatically according to data kind. Manual setting is invalid.

FORECAST LV: This data decides boundary value to output Trouble forecast signal. Manual setting is available.

Operation

Data Setting

Move cursor by Cursor key and input number by MDI key and press input key [INPUT] to set data.

AXIS: In case of servo axis, input control axis number.
(Example) Set ”1” for first servo axis.
In case of spindle axis, input ”10 + spindle number”.
(Example) Set ”11” for first spindle axis.

DATA KIND: Input data number value according to the following table.

- Data of Servo motor

Data number	Data
1	Accumulated command pulse (pulse)
2	Accumulated feedback pulse (pulse)
3	Position error (pulse)
4	Reference counter (pulse)
5	Actual speed (min^{-1})
6	Command current (%)
7	Thermal simulation data (%)
8	Torque (%)
9	Disturbance torque (%)
10	Optional data
11	R–phase current (%)
12	Effective current (%)
13	Pulse coder AMR data
14	Optional data 2

- Data of Spindle motor

Data number	Data
15	Actual speed (min^{-1})
16	Load meter (%)
17	Position error (pulse)
18	Actual speed (min^{-1}) (80msec)
19	Load meter (%) (80msec)

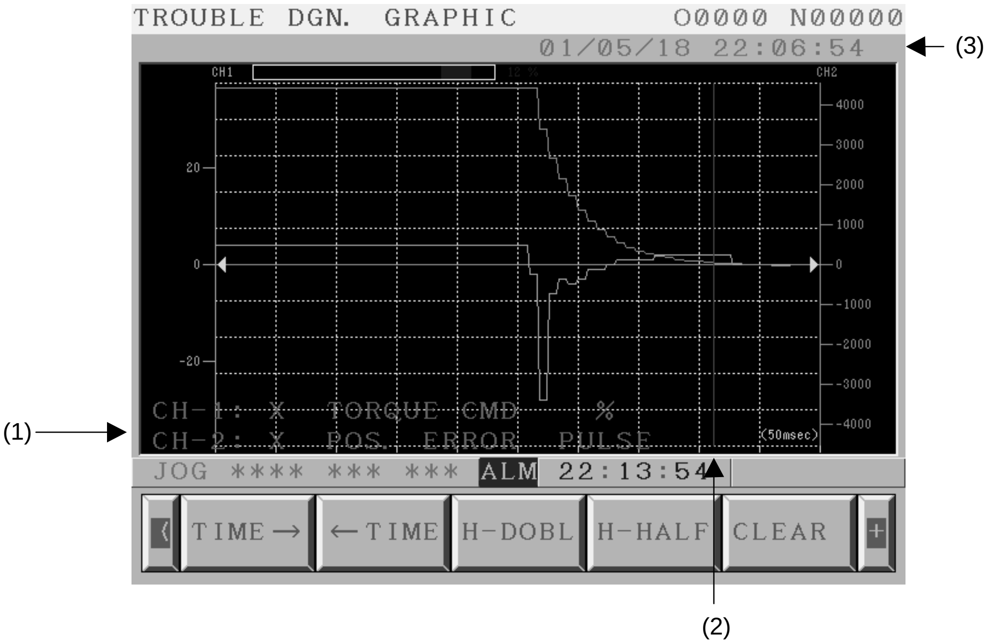
18.1.5 Trouble Diagnosis Graphic Screen

Servo/spindle data is automatically memorized for several seconds before alarm occurs and display and waveform of data can be displayed in Trouble Diagnosis Graphic Screen.
Maximum 2 kinds of data are displayed in the same time.

Display

Display procedure

1 Press function key [Message].
2 Press the continuous menu key [>] and press soft key [W.GRPH].
3 Press soft key [G–ADJ.].



Contents of display

(1) Contents of displayed waveform:
Channel No. : Axis name Data Kind Data unit
(2) Red vertical line shows the position (time) of alarm.
(3) Data and time of alarm

Operation

Change of position and magnification

When Soft key [G–ADJ.] is pressed, the following soft keys appear.

```
[<] [ W.PRM ] [ G–ADJ. ] [ TRB_LV ] [        ] [ (OPRT) ] [ ]
[<] [ TIME→ ] [ TIME← ] [ H–DOBL ] [ H–HALF ] [ (OPRT) ] [>]
[<] [ CH–1↑ ] [ CH–1↓ ] [ V–DOBL ] [ V–HALF ] [ (OPRT) ] [>]
[<] [ CH–2↑ ] [ CH–2↓ ] [ V–DOBL ] [ V–HALF ] [ (OPRT) ] [>]
```

[TIME→]: Shift the waveform rightward.
[TIME←]: Shift the waveform leftward.
[H–DOBL]: Double the time scale of the waveform
[H–HALF]: Half the time scale of the waveform
[V–DOBL]: Double the height scale of the waveform
[V–HALF]: Half the height scale of the waveform
[CH–1↑]: Shift the zero point upward
[CH–1↓]: Shift the zero point downward
[CH–2↑]: Shift the zero point upward
[CH–2↓]: Shift the zero point downward

The time scale is a common scale for the channel 1 and 2.
The position and magnification of the height scale can be set for each channel.

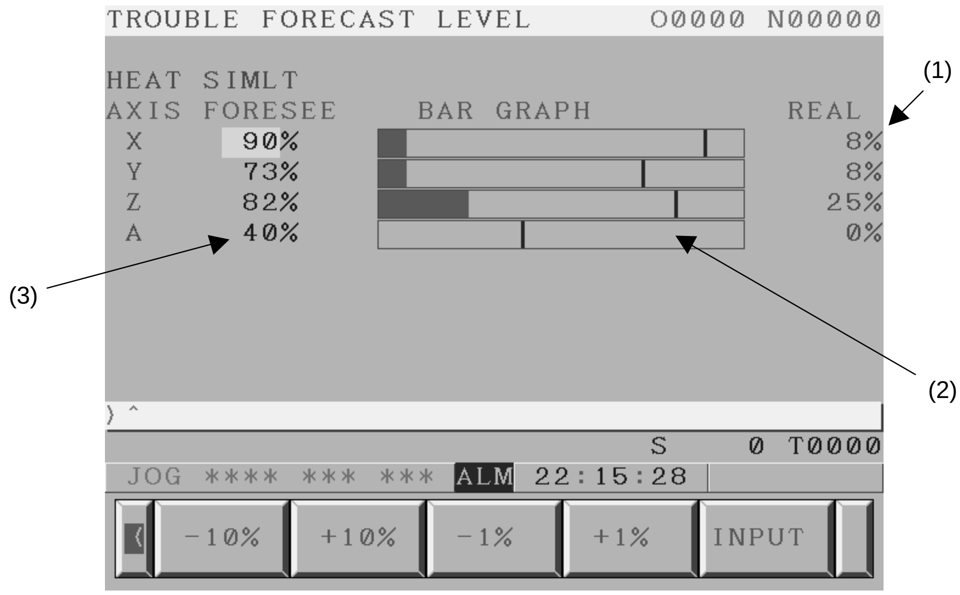
18.1.6 Trouble Forecast Level Setting Screen (Only for Servo Axis)

Trouble forecast level is set in this screen.
Two trouble forecast levels, thermal simulation and disturbance torque, can be set.

Display

Display procedure

1 Press function key [Message].
2 Press the continuous menu key [>] and press soft key [W.GRPH].
3 Press soft key [TRB LV].



Contents of display

(1) Current value of thermal simulation/disturbance torque is displayed with sign.
(2) Current value of thermal simulation or disturbance torque is displayed by bar graphic.
Light Blue part: Current value of thermal simulation/disturbance torque.
Red part: Trouble forecast level.
(3) Trouble forecast level.

Operation

Setting Trouble forecast level

1 Select thermal simulation or disturbance torque by page keys [Page↑]/[Page↓].

2 Select axis by cursor keys [↑]/[↓].

3 Input numerical value by MDI key and press [INPUT] key.

Trouble forecast level is input into parameter No.8860 and 8861.

Change of Trouble forecast level

When Soft key [TRB LV] is pressed, the following soft keys appear.

[<] [GUIDE] [W.PRM] [G–ADJ.] [TRB_LV] [(OPRT)] [ ]

[<] [–10%] [+10%] [–1%] [+1%] [INPUT] [ ]

[–10%]: Subtract 10% from Trouble forecast level.
[+10%]: Add 10% to Trouble forecast level.
[–1%]: Subtract 1% from Trouble forecast level.
[+1%]: Add 1% to Trouble forecast level.
[INPUT]: Input Trouble forecast level by MDI key.

Trouble forecast signal

When thermal simulation or disturbance torque of servo axis exceeds trouble forecast level, Trouble forecast signal TDFSVx (F298.0 – F298.3) is output.

Parameters TRSx (No.8853/8854) need to be set to 1 to perform the trouble forecast.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8850								MDG

[Data type] Bit

MDG Trouble diagnosis function is:

0 : Available.
1 : Not available.

	#7	#6	#5	#4	#3	#2	#1	#0
8853	TRS8	TRS7	TRS6	TRS5	TRS4	TRS3	TRS2	TRS1

[Data type] Bit

TRS1 to TRS8 Trouble forecast is:

0 : Not available.
1 : Available.

8860	Trouble forecast level for thermal simulation
8861	Trouble forecast level for disturbance torque

[Data type] WORD AXIS

[Unit of data] %

[Valid data range] 0 to 100%

	#7	#6	#5	#4	#3	#2	#1	#0
13110								JPN

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

[Data type] Bit

JPN As the display language used for trouble diagnosis and machine alarm diagnosis:

0 : English has precedence.
For machine alarm diagnosis, the GUIE_USR.MEM file has precedence.

1 : Japanese has precedence.
For machine alarm diagnosis, the GUIJ_USR.MEM file has precedence.

Signal

Trouble forecast signal TDF1 to TDF7 <F0298>

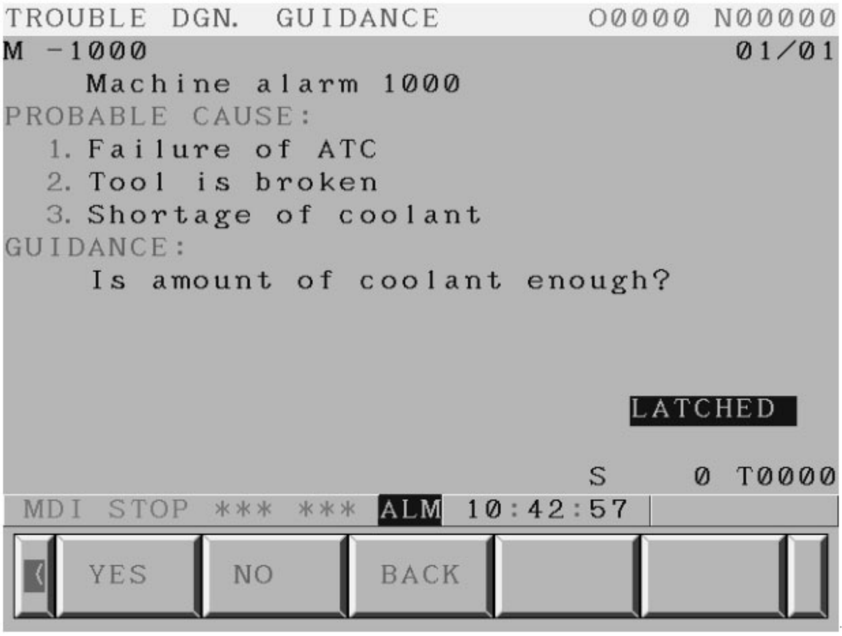
[Classification] Output signal

[Output condition] When thermal simulation or disturbance torque of servo axis exceeds the trouble forecast level, TDFSVx turn to 1.

18.2 MACHINE ALARM DIAGNOSIS

18.2.1 Outline

Machine alarms (External alarm message and Macro alarm) can be diagnosed on the trouble diagnosis guidance screen in addition to CNC alarms.



Example of a trouble diagnosis guidance screen

Kind of diagnosed alarms

The following alarms can be diagnosed.

1 External alarm message (Alarm No.1000–1999)

2 Macro alarm (#3000) (Alarm No.3000–3200)

> **NOTE**
> Option functions are separately needed to use the above alarms.

Required environment

The following environment is needed to make the data which is displayed on the trouble diagnosis guidance screen.

1 Microsoft® Excel 97 or later version

2 MS–DOS® (Command prompt of Microsoft® Windows® is available.)

3 A personal computer included an operating system (OS) on which the above application can be executed.

Microsoft, Windows, MS–DOS are registered trademarks of Microsoft Corporation of the USA. Microsoft Excel 97 is a product whose copyright is owned by Microsoft Corporation of the USA.

18.2.2 Making Guidance Tables

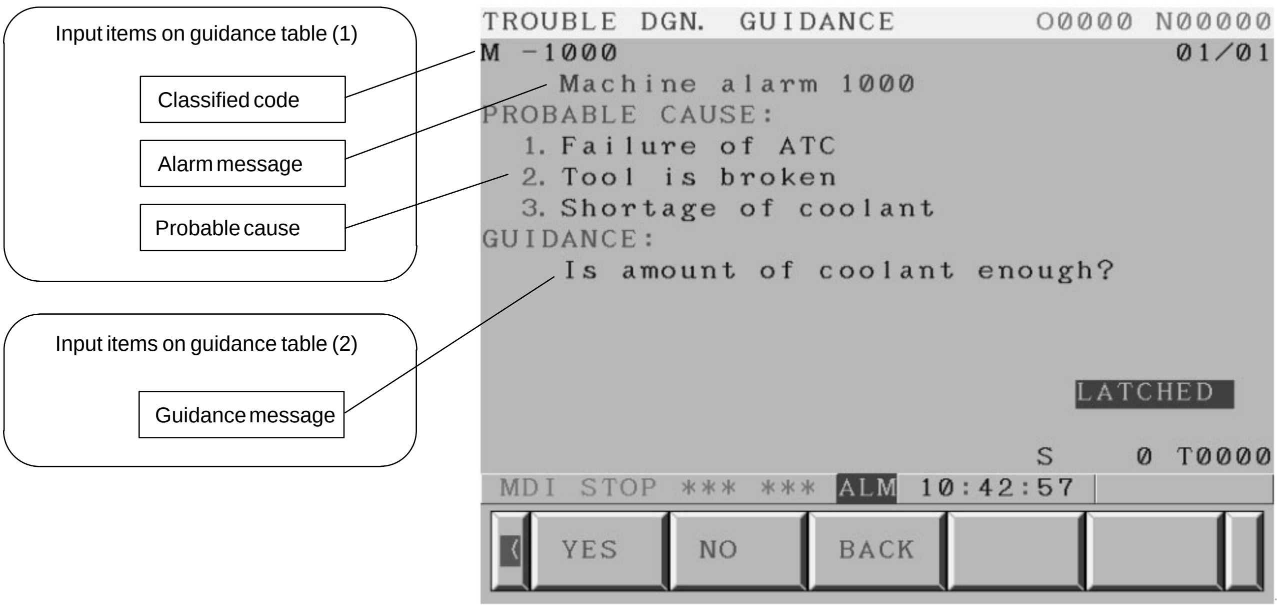
Guidance tables for diagnosis of machine alarms are made by using an Excel sheet which is provided by FANUC.
Making procedure of guidance tables is as follows.

1 Making a guidance table (1) (Using Excel)

2 Making a guidance table (2) (Using Excel)

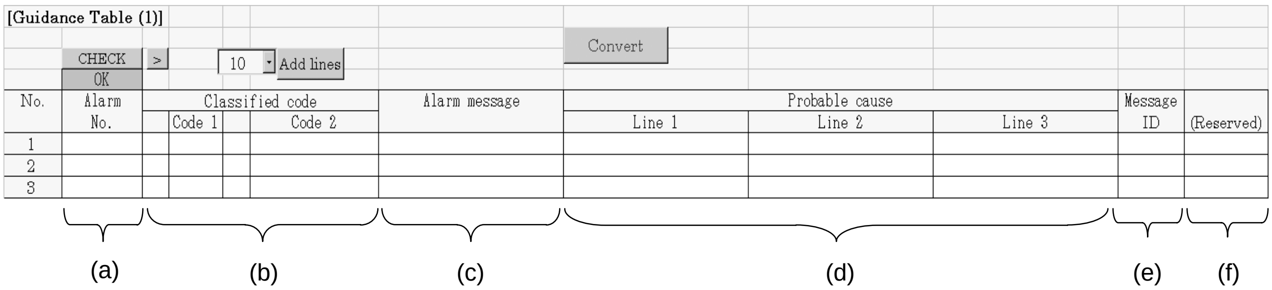
3 Converting an Excel sheet to a memory card file.
(Using Excel and MS–DOS command)

Input items and a trouble diagnosis guidance display



Making a guidance table (1)

Alarm No., Classified code, Alarm message, Probable cause and etc. are registered into a guidance table (1).



Input items

(a) Alarm No.
Input alarm numbers of the external alarm message or the macro alarm.
Max 5 figures (Range: 0–65535).
Ex.) In case of the external alarm message alarm No.1000
”1000”

(b) Classified code
Two codes (Code 1 and Code 2) are available.
These codes are displayed at 1st line on the trouble diagnosis guidance screen as ”[Code 1] – [Code 2]”.

- Code 1
  Input the type of the alarm and so on.
  Max 2 characters.

CAUTION
A character string ”NC” is reserved as the code 1 of CNC alarm. Do not use ”NC” as the code 1.

- Code 2
  Input the alarm No. and so on.
  Max 16 characters.

(c) Alarm message
Max 32 characters * 1 line.
It is displayed at 2nd line on the trouble diagnosis guidance screen.

(d) Probable cause
Max 32 characters * Max 3 lines.
It is displayed at the part of ”PROBABLE CAUSE” on the trouble diagnosis guidance screen.

(e) Message ID
A Message ID specifies a guidance message which should be displayed at first when an alarm occurs.
Max 6 characters with capital letters.
Ex.) In case of the first guidance message for the external alarm message alarm No.1000
”1000A”

Please refer to ”2.2 Making a guidance table (2)” about the message ID and the guidance message.

(f) (Reserved)
Please input nothing at this item.

Operation of buttons

- [Check] button
  When [CheckECK] button is pushed, a range of number, a number of characters and invalid character codes are checked.
  Alarm numbers are checked if they are within 0–65535. But alarm numbers in the external alarm message and the macro alarm are actually available.
  If the result is OK, a string of the cell under the button is changed from ”Unchecked” to ”OK”. If the result is NG, the string is changed to ”NG”. And the cell color becomes green. And a comment, which can display the cause of error, is added to the cell.

  Ex.)

	CHECK
	Unchecked
No.	Alarm No.
1	1000
2	1001

	CHECK
	OK
No.	Alarm No.
1	1000
2	1001

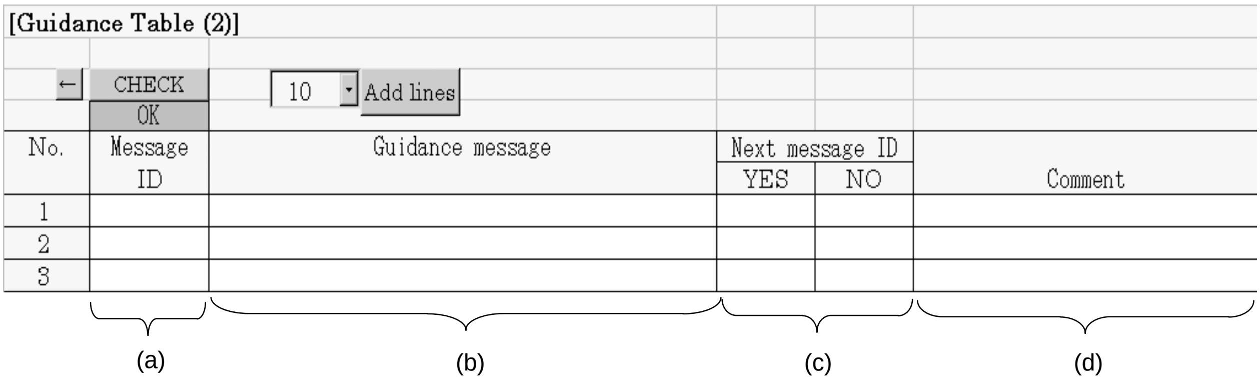
	CHECK	>		10	Add lines
	NG				
No.	Alarm No.	Classified code			
			Code 1		Code 2
1	1000				
2	123456				
3					
4					
5					
6					

Out of range.
(0 – 65535)

- [Add lines] button
  Please push [Add lines] button in order to increase the number of alarms.
  The number is specified in a cell on the left side of the button.
- [Convert] button
  When [Convert] button is pushed, both guidance tables (1) and (2) on the Excel sheet are converted to the text type data.
  Please refer to ”2.3 Converting from an Excel file to a memory card file.”
- [>] button
  WhenECK [>] ECKbutton is pushed, the guidance table (2) is displayed.

Making a guidance table (2)

”Guidance message” is registered into a guidance table (2). ”Guidance message” is a question and an instruction to an operator.



Input items

(a) Message ID

A Message ID specifies a guidance message.
Max 6 characters with capital letters.

Ex.) In case of the first message for the external alarm message alarm No.1000
”1000A”

Ex.) In case of the second message for the external alarm message alarm No.1000
”1000B”

> **CAUTION**
> Message IDs starting with ”M” are reserved for a jump to MTB’s guidance table from a CNC guidance table.
> Ex.) ”M205”, ”M407”
> Do not use a message ID starting with ”M” for the other purpose.
> Please refer to Item ”Jump from CNC guidance table to MTB’s guidance table”.

(b) Guidance message

Write a question, an instruction and so on to operators.
Max 32 characters * Max 4 lines.
It is displayed at the part of ”GUIDANCE” on the trouble diagnosis guidance screen.

- If 32 characters are displayed in one line, the next character is automatically displayed in a new line.
- If a new line is required on purpose, write ”\n”. ”\n” is not counted in the number of characters.
- The total number of lines must be within 4 lines.

(c) Next message ID

Next message ID specifies a guidance message which is displayed after an operator pushes a soft–key [YES]/[NO] to answer a question.
Max 6 characters with capital letters.

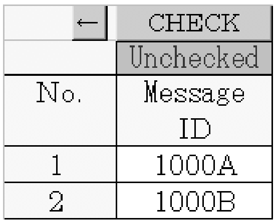
- Input ”–1” when there is no next message ID to display after a soft–key [YES]/[NO] is pushed.

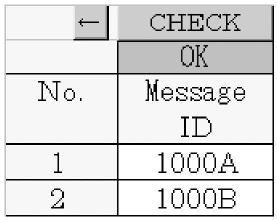
- Input ”–1” for both YES and NO in case of the end of a diagnosis.

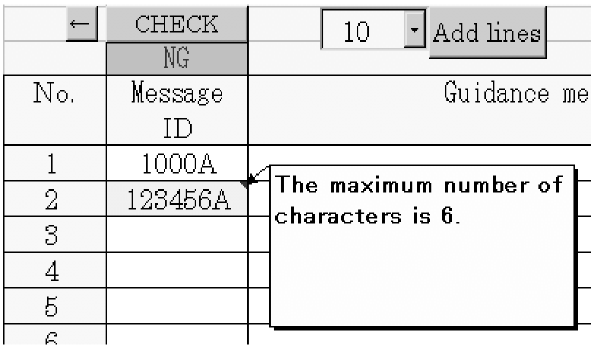
(d) Notes
It is possible to write some notes here.

Operation of buttons

- [Check] button
When [Check] button is pushed, a range of number, a number of characters and invalid character codes are checked.
If the result is OK, a string of the cell under the button is changed from ”Unchecked” to ”OK”. If the result is NG, the string is changed to ”NG”. And the cell color becomes green. And a comment, which can display the cause of error, is added to the cell.
Ex.)







And when a message ID starting with ”M” is found the cell color becomes sky blue.
Ex.)

No.	Message ID
1	1000A
2	1000B
3	M205

- [Add lines] button
Please push [Add lines] button in order to increase the number of the guidance messages.
The number is specified in a cell on the left side of the button.
- [<] button
When [<] button is pushed, the guidance table (1) is displayed.

Converting an Excel sheet to a memory card file

Guidance tables on an Excel sheet is need to be converted to a memory card file which CNC can reads. A procedure of conversion is as follows.

A procedure of conversion

(1) Finish to making both guidance tables (1) and (2), and then push [Convert] button on a guidance table (1).

- In PC, a sub folder (directory) named ”GUIDEDAT” is made in the folder where an Excel sheet is installed. And text type files are made in this sub folder.

(2) Execute ”TXT2MEM.BAT” at MS–DOS prompt. ”TXT2MEM.BAT” is also installed in the same folder as an Excel sheet.

Ex.) In case that an Excel sheet is installed in a folder ”D:\FLOW\”
Type ”TXT2MEM” and push an enter key.

```
D:\FLOW>TXT2MEM[Enter]
```

(3) A file named ”GUIE_USR.MEM” (for English) or ”GUIJ_USR.MEM” (for Japanese) is made in the same folder as an Excel sheet.

(4) Load ”GUIE_USR.MEM” or ”GUIJ_USR.MEM” into a CNC via a memory card by the boot function. Then it is possible to diagnose the machine alarms.
It is possible to load both of ”GUIE_USR.MEM” and ”GUIJ_USR.MEM”. In this case one of them can be selected by a parameter JPN (No.13110#0). When this parameter is set, power must be powered off.
When only one of them is loaded, loaded file is available irrespective of a parameter JPN (No.13110#0).

Jump from CNC guidance table to MTB’s guidance table

As a result of CNC alarm diagnosis, the alarm may be caused by machine trouble.
Considering such a case, it is possible to jump to MTB’s guidance table from CNC guidance table by the special message IDs starting with ”M”.
If the flowing special message IDs stating with ”M” are created in MTB’s guidance table, a jump to MTB’s guidance table from CNC guidance table becomes available.

Reserved message IDs

No.	Message ID	Alarm name	Presumed cause
1	M205	Rigid mode DI OFF	The rigid mode DI signal (G061.0) is not set to 1 when the rigid tapping is executed.
2	M409	Abnormal load detected	A mechanical collision or twist occurred, resulting in a load torque higher than a normal operation value.
3	M410	Excessive stop error	A mechanical collision or twist occurred, disabling an axis from reaching a target position.
4	M411	Excessive move error	A mechanical collision or twist occurred, disabling an axis from moving.
5	M420	Excessive torque difference	Two axes to be moved synchronously lost mechanical synchronism with each other, resulting in a large torque difference.
6	M421	Excessive semi–closed loop error	With a closed–loop machine, a shift occurred between the motion of the motor and the motion of the separate detector for a cause such as a mechanical twist.
7	M436	OVC alarm	A mechanical collision or twist occurred, resulting in a large load and the flow of an excessive current.

CAUTION

All message IDs starting with ”M” are reserved for a jump to MTB’s guidance table from a CNC guidance table.
In the future, a message ID to which a jump is made from a CNC guidance table may be added. So, do no use a message ID starting with ”M” which may or may not be included in the table above for the other purpose.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
13110								JPN

[Data type] Bit

JPN Language used in the trouble diagnosis and the machine alarm diagnosis

0 : English is prior.
In case of the machine alarm diagnosis, a file ”GUIE_USR.MEM” is prior.

1 : Japanese is prior.
In case of the machine alarm diagnosis, a file ”GUIJ_USR.MEM” is prior.

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

Caution

1 This function is an option function.

2 Equipment for making the guidance tables (Microsoft Excel and a personal computer with an operating system (OS) and so on) is separately needed.

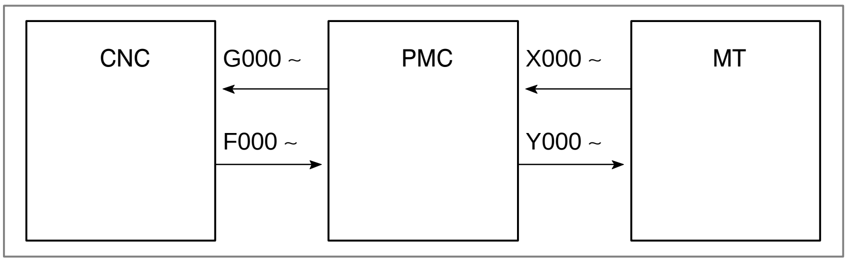
APPENDIX

A INTERFACE BETWEEN CNC AND PMC

A.1 LIST OF ADDRESSES

A.1.1 Series 0*i*/0*i* Mate Address List

The figure below illustrates the addresses of interface signals between the CNC and PMC.

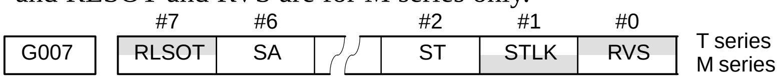


Following shows table of addresses:

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

[Example 1]

Signals EXLM and ST are common signals, STLK is for T series only and RLSOT and RVS are for M series only.



Notes when using the PMC–SA1

When the PMC–SA1 is used, transmittable signals are restricted as shown below.

Signal address	Description
G000 to G255	Signals transmitted from the PMC to the CNC when the PMC–SA1 is used
F000 to F255	Signals transmitted from the CNC to the PMC when the PMC–SA1 is used

As seen from the above table, signals at G256 and higher and those at F256 cannot be used in a system having the PMC–SA1. Therefore, features requiring these signals cannot be used.

Notes when using the PMC–SB7

When the PMC–SB7 is used, transmittable signals are restricted as shown below.

Signal address	Description
G000 to G512	Signals transmitted from the PMC to the CNC when the PMC–SB7 is used
F000 to F512	Signals transmitted from the CNC to the PMC when the PMC–SB7 is used

MT → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0	
X000									
X001									
X002									
X003									
X004	SKIP	ESKIP SKIP6	–MIT2 SKIP5	+MIT2 SKIP4	–MIT1 SKIP3	+MIT1 SKIP2	ZAE SKIP8	XAE SKIP7	(T series)
	SKIP	ESKIP				ZAE	YAE	XAE	(M series)
X005									
X006									
X007									
X008				*ESP					
X009					*DEC4	*DEC3	*DEC2	*DEC1	
X010									
X011									
X012									

- **T series/M series**

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G003								
G004			MFIN3	MFIN2	FIN			
G005	BFIN	AFL		BFIN	TFIN	SFIN	EFIN	MFIN
G006		SKIPP		OVC		*ABSM		SRN
G007	RLSOT	EXLM	*FLWU	RLSOT3		ST	STLK	
G008	ERS	RRW	*SP	*ESP	*BSL		*CSL	*IT
G009				PN16	PN8	PN4	PN2	PN1
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0
G013								
G014							ROV2	ROV1
G015								
G016	F1D							
G017								
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019	RT		MP2	MP1	HS3D	HS3C	HS3B	HS3A
G020								
G021								
G022								
G023			NOINPS					
G024	EPN7	EPN6	EPN5	EPN4	EPN3	EPN2	EPN1	EPN0

	#7	#6	#5	#4	#3	#2	#1	#0
G025	EPNS		EPN13	EPN12	EPN11	EPN10	EPN9	EPN8
G026								
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC	SPSTP	*SCPF	*SUCPF		GR2	GR1	
G029		*SSTP	SOR	SAR				GR21
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G031								
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
G038	*BECLP	*BEUCP			SPPHS	SPSYC		
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040	WOSET	PRC	S2TLS					
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042	DMMC				HS3ID	HS3IC	HS3IB	HS3IA
G043	ZRN		DNCI			MD4	MD2	MD1
G044							MLK	BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G046	DRN	KEY4	KEY3	KEY2	KEY1		SBK	
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0

	#7	#6	#5	#4	#3	#2	#1	#0
G050							*TLV9	*TLV8
G051								
G052								
G053	CDZ	SMZ			UINT			TMRON
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
G056								
G057								
G058					EXWT	EXSTP	EXRD	MINP
G059								
G060	*TSB							
G061			RGTSP2	RGTSP1				RGTAP
G062		RTNT					*CRTOF	
G063			NOZAGC					
G064								
G065								
G066	EKSET			RTRCT			ENBKY	IGNVRY
G067	HCREQ	HCABT						
G068								
G069								
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072	RCHHGA	MFNHGA	INCMDA	OVRA	DEFMDA	NRROA	ROTAA	INDXA
G073				DSCNA		MPOFA	SLVA	MORCMA
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB

	#7	#6	#5	#4	#3	#2	#1	#0
G075	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076	RCHHGB	MFNHGB	INCMDB	OVRB	DEFMDB	NRROB	ROTAB	INDXB
G077				DSCNB		MPOFB	SLVB	MORCMB
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08
G082	Reserved for order–made macro							
G083	Reserved for order–made macro							
G084								
G085								
G086								
G087								
G088								
G089								
G090								
G091					SRLNI3	SRLNI2	SRLNI1	SRLNI0
G092				BGEN	BGIALM	BGION	IOLS	IOLACK
G093								
G094								
G095								
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	#7	#6	#5	#4	#3	#2	#1	#0
G100					+J4	+J3	+J2	+J1
G101					*+ED24	*+ED23	*+ED22	*+ED21
G102					–J4	–J3	–J2	–J1
G103					*–ED24	*–ED23	*–ED22	*–ED21
G104					+EXL4	+EXL3	+EXL2	+EXL1
G105					–EXL4	–EXL3	–EXL2	–EXL1
G106					MI4	MI3	MI2	MI1
G107					*+ED34	*+ED33	*+ED32	*+ED31
G108					MLK4	MLK3	MLK2	MLK1
G109					*–ED34	*–ED33	*–ED32	*–ED31
G110					+LM4	+LM3	+LM2	+LM1
G111								
G112					–LM4	–LM3	–LM2	–LM1
G113								
G114					*+L4	*+L3	*+L2	*+L1
G115								
G116					*–L4	*–L3	*–L2	*–L1
G117								
G118					*+ED4	*+ED3	*+ED2	*+ED1
G119								
G120					*–ED4	*–ED3	*–ED2	*–ED1
G121								
G122								
G123								

	#7	#6	#5	#4	#3	#2	#1	#0
G124					DTCH4	DTCH3	DTCH2	DTCH1
G125					IUDD4	IUDD3	IUDD2	IUDD1
G126					SVF4	SVF3	SVF2	SVF1
G127					CDF4	CDF3	CDF2	CDF1
G128								
G129								
G130					*IT4	*IT3	*IT2	*IT1
G131								
G132					+MIT4	+MIT3	+MIT2	+MIT1
G133								
G134					–MIT4	–MIT3	–MIT2	–MIT1
G135								
G136					EAX4	EAX3	EAX2	EAX1
G137								
G138					SYNC4	SYNC3	SYNC2	SYNC1
G139								
G140					SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1
G141								
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA	ELCKZA	EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A

	#7	#6	#5	#4	#3	#2	#1	#0
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E
G152								
G153								
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB	ELCKZB	EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G162								
G163								
G164								
G165								
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC	ELCKZC	EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

	#7	#6	#5	#4	#3	#2	#1	#0
G174								
G175								
G176								
G177								
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD	ELCKZD	EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D
G186								
G187								
G188								
G189								
G190								
G191								
G192					IGVRY4	IGVRY3	IGVRY2	IGVRY1
G193								
G194								
G195								
G196								
G197								
G198					NPOS4	NPOS3	NPOS2	NPOS1

	#7	#6	#5	#4	#3	#2	#1	#0
G199								
G200					EASIP4	EASIP3	EASIP2	EASIP1
G201								
G202								
G203								
G204								
G205								
G206								
G207								
G208								
G209								
G210								
G211								
G212								
G213								
G214								
G215								
G216								
G217								
G218								
G219								
G220								
G221								
G222								
G223								

	#7	#6	#5	#4	#3	#2	#1	#0
G224								
G225								
G226								
G227								
G228								
G229								
G230								
G231								
G232								
G233								
G234								
G235								
G236								
G237								
G238								
G239								
G240								
G241								
G242								
G243								
G244								
G245								
G246								
G247								
G248								

	#7	#6	#5	#4	#3	#2	#1	#0
G249								
G250								
G251								
G252								
G253								
G254								
G255								
G256								
G257								
G258								
G259								
G260								
G261								
G262								
G263								
G264								
G265								
G266								
G267								
G268								
G269								
G270								
G271								
G272								
G273								

	#7	#6	#5	#4	#3	#2	#1	#0
G274				CSFI1				
G275								
G276								
G277								
G278								
G279								
G280								
G281								
G282								
G283								
G284								
G285								
G286								
G287								
G288								
G289								
G290								
G291								
G292								
G293								
G294								
G295								
G296								
G297								
G298								

	#7	#6	#5	#4	#3	#2	#1	#0
G299								
G300								
G301								
G302								
G303								
G304								
G305								
G306								
G307								
G308								
G309								
G310								
G311								
G349					SVSCK4	SVSCK3	SVSCK2	SVSCK1
G359					NOINP4	NOINP3	NOINP2	NOINP1

CNC → PMC

Address / Bit number

Address	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP	SA	STL	SPL				RWD
F001	MA		TAP	ENB	DEN	BAL	RST	AL
F002	MDRN	CUT		SRNMV	THRD	CSS	RPDO	INCH
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF	MAFL	MSBK	MABSM	MMLK	MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2
F006								
F007	BF			BF	TF	SF	EFD	MF
F008			MF3	MF2				EF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308
F018								
F019								
F020								
F021								
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16

	#7	#6	#5	#4	#3	#2	#1	#0
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24
F034						GR3O	GR2O	GR1O
F035								SPAL
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O
F038					ENB3	ENB2	SUCLP	SCLP
F039								
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
F042								
F043								
F044				SYCAL	FSPPH	FSPSY	FSCSL	
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
F047				EXOFA			INCSTA	PC1DTA
F048				CSPENA				
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB

	#7	#6	#5	#4	#3	#2	#1	#0
F050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
F051				EXOFB			INCSTB	PC1DTB
F052								
F053	EKENB			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
F060						ESCAN	ESEND	EREND
F061					HCEXE	HCAB2	BCLP	BUCLP
F062	PRTSF			S2MES	S1MES			AICC
F063	PSYN							
F064						TLCHI	TLNW	TLCH
F065				RTRCTF			RGSPM	RGSPP
F066			PECK2				RTPT	G08MD
F067								
F068								
F069								
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071	PSW16	PSW15	PSW14	PSW13	PSW12	PSW11	PSW10	PSW09
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								

	#7	#6	#5	#4	#3	#2	#1	#0
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O	RTAP		MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	– J4O	+ J4O	– J3O	+ J3O	– J2O	+ J2O	– J1O	+ J1O
F082								
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090						ABTSP2	ABTSP1	ABTQSV
F091								
F092								
F093								
F094					ZP4	ZP3	ZP2	ZP1
F095								
F096					ZP24	ZP23	ZP22	ZP21
F097								
F098					ZP34	ZP33	ZP32	ZP31
F099								

	#7	#6	#5	#4	#3	#2	#1	#0
F100					ZP44	ZP43	ZP42	ZP41
F101								
F102					MV4	MV3	MV2	MV1
F103								
F104					INP4	INP3	INP2	INP1
F105								
F106					MVD4	MVD3	MVD2	MVD1
F107								
F108					MMI4	MMI3	MMI2	MMI1
F109								
F110					MDTCH4	MDTCH3	MDTCH2	MDTCH1
F111								
F112					EADEN4	EADEN3	EADEN2	EADEN1
F113								
F114					TRQL4	TRQL3	TRQL2	TRQL1
F115								
F116								
F117								
F118								
F119								
F120					ZRF4	ZRF3	ZRF2	ZRF1
F121								
F122								HDO0
F123								
F124					+OT4	+OT3	+OT2	+OT1

	#7	#6	#5	#4	#3	#2	#1	#0
F125								
F126					–OT4	–OT3	–OT2	–OT1
F127								
F128								
F129	*EAXSL		EOV0					
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131							EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134							EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137							EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140							EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
F143								
F144								
F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
F146								
F147								
F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
F149								

	#7	#6	#5	#4	#3	#2	#1	#0
F150								
F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168								
F169								
F170								
F171								
F172	PBATL	PBATZ						
F173								
F174								

	#7	#6	#5	#4	#3	#2	#1	#0
F175								
F176								
F177	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F178					SRLNO3	SRLNO2	SRLNO1	SRLNO0
F179								
F180					CLRCH4	CLRCH3	CLRCH2	CLRCH1
F181								
F182					EACNT4	EACNT3	EACNT2	EACNT1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								
F196								
F197								
F198								
F199								

	#7	#6	#5	#4	#3	#2	#1	#0
F200								
F201								
F202								
F203								
F204								
F205								
F206								
F207								
F208								
F209								
F210								
F211								
F212								
F213								
F214								
F215								
F216								
F217								
F218								
F219								
F220								
F221								
F222								
F223								
F224								

	#7	#6	#5	#4	#3	#2	#1	#0
F225								
F226								
F227								
F228								
F229								
F230								
F231								
F232								
F233								
F234								
F235								
F236								
F237								
F238								
F239								
F240								
F241								
F242								
F243								
F244								
F245								
F246								
F247								
F248								
F249								

	#7	#6	#5	#4	#3	#2	#1	#0
F250								
F251								
F252								
F253								
F254								
F255								
F256								
F257								
F258								
F259								
F260								
F261								
F262								
F263								
F264								
F265								
F266								
F267								
F268								
F269								
F270								
F271								
F272								
F273								
F274				CSFO1				

	#7	#6	#5	#4	#3	#2	#1	#0
F275								
F276								
F277								
F278								
F279								
F280								
F281								
F282								
F283								
F284								
F285								
F286								
F287								
F288								
F289								
F290								
F291								
F292								
F293								
F294								
F295								
F296								
F297								
F298					TDFSV4	TDFSV3	TDFSV2	TDFSV1
F299								

	#7	#6	#5	#4	#3	#2	#1	#0
F300								
F301								
F302								
F303								
F304								
F305								
F306								
F307								
F308								
F309								
F310								
F311								
F312								
F313								
F314								
F315								
F349					TSA4	TSA3	TSA2	TSA1

A.2 SIGNAL SUMMARY

A.2.1 Signal Summary (In Order of Functions)

○ : Available
– : Unavailable

Function	Signal name	Symbol	Address	T series	M series	Section
Data input/output with I/O Link	Power Mate background operation signal	BGEN	G092#4	○	○	13.5
	Power Mate read/write alarm signal	BGIALM	G092#3	○	○	13.5
	Power Mate read/write busy signal	BGION	G092#2	○	○	13.5
	Slave diagnosis selection signal	EDGN	F177#7	○	○	13.5
	Slave parameter selection signal	EPARM	F177#6	○	○	13.5
	Slave program selection signal	EPRG	F177#4	○	○	13.5
	Slave external read start signal	ERDIO	F177#1	○	○	13.5
	Slave read/write stop signal	ESTPIO	F177#2	○	○	13.5
	Slave macro variable selection signal	EVAR	F177#5	○	○	13.5
	Slave external write start signal	EWTIO	F177#3	○	○	13.5
	External read start signal	EXRD	G058#1	○	○	13.5, 13.3
	External read/punch stop signal	EXSTP	G058#2	○	○	13.5, 13.3
	External punch start signal	EXWT	G058#3	○	○	13.5, 13.3
	I/O Link check signal	IOLACK	G092#0	○	○	13.5
	Slave I/O Link selection signal	IOLINK	F177#0	○	○	13.5
	I/O Link specification signal	IOLS	G092#1	○	○	13.5
	Read/punch alarm signal	RPALM	F053#3	○	○	13.5, 13.3
	Read/punch busy signal	PRBSY	F053#2	○	○	13.5, 13.3
	Group number specification signals	SRLNI0 to SRLNI3	G091#0 to #3	○	○	13.5
	Group number output signals	SRLNO0 to SRLNO3	F178#0 to #3	○	○	13.5
External I/O device control	External read start signal	EXRD	G058#1	○	○	13.3
	External punch start signal	EXWT	G058#3	○	○	
	External read/punch stop signal	EXSTP	G058#2	○	○	
	Background editing signal	BGEACT	F053#4	○	○	
	Read/punch busy signal	RPBSY	F053#2	○	○	
	Read/punch alarm signal	RPALM	F053#3	○	○	
Alarm signal	Alarm signal	AL	F001#0	○	○	2.4
	Battery alarm signal	BAL	F001#2	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Abnormal load detection	Abnormal load detection ignore signal	IUDD1 to IUDD4	G125	○	○	2.9
	Servo axis abnormal load detected signal	ABTQSV	F090#0	○	○	
	First–spindle abnormal load detected signal	ABTSP1	F090#1	○	○	
	Second–spindle abnormal load detected signal	ABTSP2	F090#2	○	○	
Position display neglect	Position display neglect signals	NPOS1 to NPOS4	G198	○	○	12.1.10
Multiple M commands in a single block	2nd M function code signals	M200 to M215	F014 to F015	○	○	8.3
	3rd M function code signals	M300 to M315	F016 to F017	○	○	
	2nd M function strobe signal	MF2	F008#4	○	○	
	3rd M function strobe signal	MF3	F008#5	○	○	
Inch/metric conversion	Inch input signal	INCH	F002#0	○	○	11.4
Index table indexing (M series)	B–axis clamp signal	BCLP	F061#1	—	○	11.10
	B–axis clamp completion signal	*BECLP	G038#7	—	○	
	B–axis unclamp signal	BUCLP	F061#0	—	○	
	B–axis unclamp completion signal	*BEUCP	G038#6	—	○	
In–position check	In–position signals	INP1 to INP4	F104	○	○	7.2.5.1
In–position check disable signal	In–position check disable signal	NOINPS	G023#5	○	○	7.2.5.3
	Each axis in–position check disable signal	NOINP1 to NOINP4	G359#0 to #3	○	○	7.2.5.3
AI advanced control	AI advanced control mode signal	AICC	F062#0	—	○	7.1.13
F 1–digit feed (M series)	F 1–digit feed selection signal	F1D	G016#7	—	○	7.1.5
Error detection (T series)	Error detection signal	SMZ	G053#6	○	—	7.2.5.4
Overtravel signal	Overtravel signals	*+L1 to *+L4	G114	○	○	2.3.1
		*–L1 to *–L4	G116	○	○	
Override cancel	Override cancel signal	OVC	G006#4	○	○	7.1.7.3
Feedrate override	Feedrate override signals	*FV0 to *FV7	G012	○	○	7.1.7.2
Optional block skip/addition of optional block skip	Optional block skip signals	BDT1, BDT2 to BDT9	G044#0, G045	○	○	5.5
	Optional block skip check signals	MBDT1, MBDT2 to MBDT9	G004#0, F005	○	○	
External key input	External key input mode selection signal	ENBKY	G066#1	○	○	15.5
	Key code signals	EKC0 to EKC7	G098	○	○	
	Key code read signal	EKSET	G066#7	○	○	
	Key code read completion signal	EKENB	F053#7	○	○	
	Key input disable signal	INHKY	F053#0	○	○	
	Program screen display mode signal	PRGDPL	F053#1	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
External deceleration	External deceleration signals	*+ED1 to *+ED4	G118	○	○	7.1.9
		*–ED1 to *–ED4	G120	○	○	
	External deceleration signals 2	*+ED21 to *+ED24	G101#0 to #3	○	○	
		*–ED21 to *–ED24	G103#0 to #3	○	○	
	External deceleration signals 3	*+ED31 to *+ED34	G107#0 to #3	○	○	
		*–ED31 to *–ED34	G109#0 to #3	○	○	
External data input	Data signals for external data input	ED0 to ED15	G000, G001	○	○	15.2
	Address signals for external data input	EA0 to EA6	G002#0 to #6	○	○	
	Read signal for external data input	ESTB	G002#7	○	○	
	Read completion signal for external data input	EREND	F060#0	○	○	
	Search completion signal for external data input	ESEND	F060#1	○	○	
	Search cancel signal for external data input	ESCAN	F060#2	○	○	
External motion function (M series)	External operation signal	EF	F008#0	—	○	11.7
External program input	External program input start signal	MINP	G058#0	○	○	13.4
External workpiece number search	Workpiece number search signals	PN1, PN2, PN4, PN8, PN16	G009#0 to 4	○	○	15.3
Expanded External Workpiece Number Search	Expanded workpiece number search signals	EPN0 to EPN13	G024#0 to G025#5	○	○	15.3.2
	Expanded workpiece number search start signal	EPNS	G025#7	○	○	
Custom macro signal	Input signals for custom macro	UI000 to UI015	G054, G055	○	○	11.5.1
	Output signals for custom macro	UO000 to UO015	F054, F055	○	○	
		UO100 to UO131	F056 to F059	○	○	
Run hour and part count display	Target part count reached signal	PRTSF	F062#7	○	○	12.1.11
	General–purpose integrating meter start signal	TMRON	G053#0	○	○	
Screen erase/automatic screen erase	Automatic screen erase disable signal	*CRTOF	G062#1	○	○	12.1.17
Screen hard copy function	Hard copy request signal	HCREQ	G067#7	○	○	13.6
	Hard copy stop request signal	HCABT	G067#6	○	○	
	Hard copy in–progress signal	HCEXE	F061#3	○	○	
	Hard copy stop request acceptance flag	HCAB2	F061#2	○	○	
Simple synchronous control	Simple synchronous axis selection signals	SYNC1 to SYNC4	G138	○	○	1.7
	Simple synchronous manual feed axis selection signals	SYNCJ1 to SYNCJ4	G140	—	○	
Angular axis control	Angular axis control disable signal for perpendicular axis	NOZAGC	G063#5	○	○	1.8

Function	Signal name	Symbol	Address	T series	M series	Section
Tool life management	Tool change signal	TLCH	F064#0	○	○	10.3
	Tool change reset signal	TLRST	G048#7	○	○	
	Individual tool change signal	TLCHI	F064#2	—	○	
	Tool life arrival notice signal	TLCHB	F064#3	—	○	10.3.2
	Individual tool change reset signal	TLRSTI	G048#6	—	○	10.3
	Tool skip signal	TLSKP	G048#5	○	○	
	New tool selection signal	TLNW	F064#1	○	○	
	Tool group number selection signals	TL01 to TL256	G047#0 to G048#0	—	○	
		TL01 to TL64	G047#0 to #6	○	—	
	Tool life count override signals	*TLV0 to *TLV9	G049#0 to G050#1	—	○	
Automatic tool length measurement (M series)/automatic tool offset (T series)	Measuring position reached signals	XAE	X004#0	○	○	14.2
		YAE	X004#1	—	○	
		ZAE	X004#2	—	○	
		ZAE	X004#1	○	—	
Direct input of tool offset value measured B (T series)	Tool offset value write mode selection signal	GOQSM	G039#7	○	—	14.4.2
	Tool offset value write signals	+MIT1, +MIT2	X004#2, #4	○	—	
		–MIT1, –MIT2	X004#3, #5	○	—	
	Tool offset number selection signals	OFN0 to OFN5	G039#0 to #5	○	—	
	Workpiece coordinate system shift value write mode selection signal	WOQSM	G039#6	○	—	
	Workpiece coordinate system shift value write signal	WOSET	G040#7	○	—	
Input of tool offset value measured A (T series)	Position record signal	PRC	G040#6	○	—	14.4.1
High–speed M/S/T/B interface	Miscellaneous function completion signal	MFIN	G005#0	○	○	8.4
	Spindle function completion signal	SFIN	G005#2	○	○	
	Tool function completion signal	TFIN	G005#3	○	○	
	2nd auxiliary function completion signal	BFIN	G005#4	○	—	
		BFIN	G005#7	—	○	
	2nd M function completion signal	MFIN2	G004#4	○	○	
	3rd M function completion signal	MFIN3	G004#5	○	○	
	External operation signal for high–speed interface	EFD	F007#1	—	○	
	External operation function completion signal	EFIN	G005#1	—	○	
High–speed skip signal	High–speed skip status signals	HDO0	F122#0	○	○	14.3.2
Trouble diagnosis	Trouble forecast signals	TDFSV1 to TDFSV4	F298#0 to #3	○	○	18.1

Function	Signal name	Symbol	Address	T series	M series	Section
Canned cycle (M series)/canned cycle for hole machining (T series)	Tapping signal	TAP	F001#5	○	○	11.6
Servo off (mechanical handle)	Servo off signals	SVF1 to SVF4	G126	○	○	1.2.8
Servo rotational speed check	Servo rotational speed check enable signal	SVSCK1 to SVSCK4	G349#0 to #3	○	○	2.10
	Servo rotational speed too–low alarm signal	TSA1 to TSA4	F349#0 to #3	○	○	
Cycle start/feed hold	Cycle start signal	ST	G007#2	○	○	5.1
	Feed hold signal	*SP	G008#5	○	○	
	Automatic operation signal	OP	F000#7	○	○	
	Cycle start lamp signal	STL	F000#5	○	○	
	Feed hold lamp signal	SPL	F000#4	○	○	
Cs–axis coordinate establishment function	Cs–axis coordinate establishment request signal	CSFI1	G274#4	○	○	9.8.2
	Cs–axis coordinate establishment alarm signal	CSFO1	F274#4	○	○	
	Cs–axis coordinate establishment status signal	CSPENA	F048#4	○	○	
Cs contour control	Cs contour control change signal	CON	G027#7	○	○	9.8.1
	Cs contour control change completion signal	FSCSL	F044#1	○	○	
	Cs contour control mode fine acceleration/deceleration function disable signal	CDF1 to CDF4	G127#0 to #3	○	○	
Outputting the movement state of an axis	Axis moving signals	MV1 to MV4	F102	○	○	1.2.5
	Axis moving direction signals	MVD1 to MVD4	F106	○	○	
Actual spindle speed output	Actual spindle speed signals	AR0 to AR15	F040, F041	○	—	9.6
Constant surface speed control	Constant surface speed signal	CSS	F002#2	○	○	9.4
Spindle positioning	Spindle stop completion signal	SPSTP	G028#6	○	—	9.7
	Spindle unclamp signal	SUCLP	F038#1	○	—	
	Spindle unclamp completion signal	*SUCPF	G028#4	○	—	
	Spindle clamp signal	SCLP	F038#0	○	—	
	Spindle clamp completion signal	*SCPF	G028#5	○	—	
Spindle orientation	Spindle orientation external stop position command signals	SHA00 to SHA11	G078#0 to G079#3	○	○	9.12
		SHB00 to SBH11	G080#0 to G081#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Torque limit command LOW signals (serial spindle)	TLMLA	G070#0	○	○	9.2
		TLMLB	G074#0	○	○	
	Torque limit command HIGH signals (serial spindle)	TLMHA	G070#1	○	○	
		TLMHB	G074#1	○	○	
	Clutch/gear signals (serial spindle)	CTH1A, CTH2A	G070#3, #2	○	○	
		CTH1B, CTH2B	G074#3, #2	○	○	
	CCW command signals (serial spindle)	SRVA	G070#4	○	○	
		SRVB	G074#4	○	○	
	CW command signals (serial spindle)	SFRA	G070#5	○	○	
		SFRB	G074#5	○	○	
	Orientation command signals (serial spindle)	ORCMA	G070#6	○	○	
		ORCMB	G074#6	○	○	
	Machine ready signals (serial spindle)	MRDYA	G070#7	○	○	
		MRDYB	G074#7	○	○	
	Alarm reset signals (serial spindle)	ARSTA	G071#0	○	○	
		ARSTB	G075#0	○	○	
	Emergency stop signals (serial spindle)	*ESPA	G071#1	○	○	
		*ESPB	G075#1	○	○	
	Spindle selection signals (serial spindle)	SPSLA	G071#2	○	○	
		SPSLB	G075#2	○	○	
	Power line switch completion signals (serial spindle)	MCFNA	G071#3	○	○	
		MCFNB	G075#3	○	○	
	Soft start/stop cancel signals (serial spindle)	SOCNA	G071#4	○	○	
		SOCNB	G075#4	○	○	
	Speed integral signals (serial spindle)	INTGA	G071#5	○	○	
		INTGB	G075#5	○	○	
	Output switch request signals (serial spindle)	RSLA	G071#6	○	○	
		RSLB	G075#6	○	○	
	Power line status check signals (serial spindle)	RCHA	G071#7	○	○	
		RCHB	G075#7	○	○	
	Orientation stop position change command signals (serial spindle)	INDXA	G072#0	○	○	
		INDXB	G076#0	○	○	
	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAA	G072#1	○	○	
		ROTAB	G076#1	○	○	
	Shortcut command signals for orientation stop position change (serial spindle)	NRROA	G072#2	○	○	
		NRROB	G076#2	○	○	
	Differential speed mode command signals (serial spindle)	DEFMDA	G072#3	○	○	
		DEFMDB	G076#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Analog override signals (serial spindle)	OVRA	G072#4	○	○	9.2
		OVRB	G076#4	○	○	
	Incremental command externally set orientation signals (serial spindle)	INCMDA	G072#5	○	○	
		INCMDB	G076#5	○	○	
	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGA	G072#6	○	○	
		MFNHGB	G076#6	○	○	
	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGA	G072#7	○	○	
		RCHHGB	G076#7	○	○	
	Magnetic sensor orientation command signals (serial spindle)	MORCMA	G073#0	○	○	
		MORCMB	G077#0	○	○	
	Subordinate operation mode command signals (serial spindle)	SLVA	G073#1	○	○	
		SLVB	G077#1	○	○	
	Motor power cutoff command signals (serial spindle)	MPOFA	G073#2	○	○	
		MPOFB	G077#2	○	○	
	Disconnection detection disbale signal	DSCNA	G073#4	○	○	
		DSCNB	G077#4	○	○	
	Alarm signals (serial spindle)	ALMA	F045#0	○	○	
		ALMB	F049#0	○	○	
	Speed zero signals (serial spindle)	SSTA	F045#1	○	○	
		SSTB	F049#1	○	○	
	Speed detection signals (serial spindle)	SDTA	F045#2	○	○	
		SDTB	F049#2	○	○	
	Speed arrival signals (serial spindle)	SARA	F045#3	○	○	
		SARB	F049#3	○	○	
	Load detection signals 1 (serial spindle)	LDT1A	F045#4	○	○	
		LDT1B	F049#4	○	○	
	Load detection signals 2 (serial spindle)	LDT2A	F045#5	○	○	
		LDT2B	F049#5	○	○	
	Torque limit signals (serial spindle)	TLMA	F045#6	○	○	
		TLMB	F049#6	○	○	
	Orientation completion signals (serial spindle)	ORARA	F045#7	○	○	
		ORARB	F049#7	○	○	
	Power line switch signals (serial spindle)	CHPA	F046#0	○	○	
		CHPB	F050#0	○	○	
	Spindle switch completion signals (serial spindle)	CFINA	F046#1	○	○	
		CFINB	F050#1	○	○	
	Output switch signals (serial spindle)	RCHPA	F046#2	○	○	
		RCHPB	F050#2	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Output switch completion signals (serial spindle)	RCFNA	F046#3	○	○	9.2
		RCFNB	F050#3	○	○	
	Subordinate operation status signals (serial spindle)	SLVSA	F046#4	○	○	
		SLVSB	F050#4	○	○	
	Position coder orientation proximity signal (serial spindle)	PORA2A	F046#5	○	○	
		PORA2B	F050#5	○	○	
	Magnetic sensor orientation completion signals (serial spindle)	MORA1A	F046#6	○	○	
		MORA1B	F050#6	○	○	
	Magnetic sensor orientation proximity signals (serial spindle)	MORA2A	F046#7	○	○	
		MORA2B	F050#7	○	○	
	Position coder one–rotation signal detection status signals (serial spindle)	PC1DTA	F047#0	○	○	
		PC1DTB	F051#0	○	○	
	Incremental orientation mode signals (serial spindle)	INCSTA	F047#1	○	○	
		INCSTB	F051#1	○	○	
	Motor activation off status signal	EXOFA	F047#4	○	○	
		EXOFB	F051#4	○	○	
Spindle speed control	Spindle stop signal	*SSTP	G029#6	○	○	9.3
	Spindle orientation signal	SOR	G029#5	○	○	
	Spindle speed override signals	SOV0 to SOV7	G030	○	○	
	Spindle speed arrival signal	SAR	G029#4	○	○	
	Spindle enable signal	ENB	F001#4	○	○	
	Gear selection signals (output)	GR1O, GR2O, GR3O	F034#0 to #2	—	○	
	Gear selection signals (input)	GR1, GR2	G028#1, #2	○	○	
	S 12–bit code signals	R01O to R12O	F036#0 to F037#3	○	○	
Spindle speed fluctuation detection	Spindle fluctuation detection alarm signal	SPAL	F035#0	○	○	9.5
Spindle synchronous control	Spindle synchronous control signal	SPSYC	G038#2	○	○	9.11
	Spindle phase synchronous control signal	SPPHS	G038#3	○	○	
	Spindle synchronous speed control completion signal	FSPSY	F044#2	○	○	
	Spindle phase synchronous control completion signal	FSPPH	F044#3	○	○	
	Phase error monitor signal	SYCAL	F044#4	○	○	
Manual handle feed	Manual handle feed axis selection signals	HS1A to HS1D	G018#0 to #3	○	○	3.2
		HS2A to HS2D	G018#4 to #7	○	○	
		HS3A to HS3D	G019#0 to #3	○	○	
	Manual handle feed amount selection signals (incremental feed signals)	MP1, MP2	G019#4, #5	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Manual handle interrupt	Manual handle interrupt axis selection signals	HS1IA to HS1ID	G041#0 to #3	○	○	3.3
		HS2IA to HS2ID	G041#4 to #7	○	○	
		HS3IA to HS3ID	G042#0 to #3	○	○	
Manual reference position return	Manual reference position return selection signal	ZRN	G043#7	○	○	4.1
	Manual reference position return selection check signal	MREF	F004#5	○	○	
	Reference position return deceleration signal	*DEC1 to *DEC4	X009	○	○	
	Reference position return end signal	ZP1 to ZP4	F094	○	○	
	Reference position establishment signal	ZRF1 to ZRF4	F120	○	○	
Jog feed/incremental feed	Feed axis and direction selection signals	+J1 to +J4	G100	○	○	3.1
		–J1 to –J4	G102	○	○	
	Manual feedrate override signals	*JV0 to *JV15	G010, G011	○	○	
	Manual rapid traverse selection signals	RT	G019#7	○	○	
CNC ready signal	CNC ready signal	MA	F001#7	○	○	2.2
	Servo ready signal	SA	F000#6	○	○	
Small hole peck drilling cycle (M series)	Overload torque signal	SKIP	X004#7	—	○	11.14
	Small hole peck drilling–in–progress signal	PECK2	F066#5	—	○	
Status output signal	Rapid traversing signal	RPDO	F002#1	○	○	2.7
	Cutting feed signal	CUT	F002#6	○	○	
Single block	Single block signal	SBK	G046#1	○	○	5.3.3
	Single block check signal	MSBK	F004#3	○	○	
Skip function	Skip signals	SKIP	X004#7	○	○	14.3.1
		SKIPP	G006#6	○	—	
Start lock/interlock	Start lock signal	STLK	G007#1	○	—	2.5
	Interlock signal	*IT	G008#0	○	○	
	Interlock signal for each axis	*IT1 to *IT4	G130	○	○	
	Manual feed interlock signal for each axis and direction	+MIT1, +MIT2	X004#2, #4	○	—	
	Manual feed interlock signal for each axis and direction	–MIT1, –MIT2	X004#3, #5	○	—	
	Interlock signal for each axis and direction	+MIT1 to +MIT4	G132#0 to #3	—	○	
		–MIT1 to –MIT4	G134#0 to #3	—	○	
	Cutting block start interlock signal	*CSL	G008#1	○	○	
	Block start interlock signal	*BSL	G008#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Stored stroke check	Stored stroke limit switching signals in axis direction	+EXL1 to +EXL4	G104	○	○	2.3.2
		–EXL1 to –EXL4	G105	○	○	
	Stored stroke limit change signal	EXLM	G007#6	○	○	
	Stroke limit external setting signals	+LM1 to +LM4	G110	—	○	
		–LM1 to –LM4	G110	—	○	
	Stroke limit release signal	RLSOT	G007#7	—	○	
	Stroke limit reached signals	+OT1 to +OT4	F124	—	○	
		–OT1 to –OT4	F126	—	○	
Stored stroke limit 2, 3	Stroke limit 3 release signal	RLSOT3	G007#4	○	○	2.3.3
Stored stroke limit	Overtravel alarm in–progress signal	OTP1 to OTP4	F124#0 to #3	○	○	2.3.1
Absolute position detection	Absolute position detector battery voltage zero alarm signal	PBATZ	F172#6	○	○	1.4.2
	Absolute position detector battery voltage low alarm signal	PBATL	F172#7	○	○	
Controlled axis detach	Controlled axis detach signals	DTCH1 to DTCH4	G124#0 to #3	○	○	1.2.4
	Controlled axis detach status signals	MDTCH1 to MDTCH4	F110#0 to #3	○	○	
Advanced preview	Advanced preview control mode signal	G08MD	F066#0	○	○	7.1.12
Software operator's panel	Software operator's panel signal (MD1)	MD1O	F073#0	○	○	12.1.14
	Software operator's panel signal (MD2)	MD2O	F073#1	○	○	
	Software operator's panel signal (MD4)	MD4O	F073#2	○	○	
	Software operator's panel signal (ZRN)	ZRNO	F073#4	○	○	
	Software operator's panel signals (+J1 to +J4)	+J1O to +J4O	F081#0, #2, #4, #6	○	○	
	Software operator's panel signals (–J1 to –J4)	–J1O to –J4O	F081#1, #3, #5, #7	○	○	
	Software operator's panel signal (RT)	RTO	F077#6	○	○	
	Software operator's panel signal (HS1A)	HS1AO	F077#0	○	○	
	Software operator's panel signal (HS1B)	HS1BO	F077#1	○	○	
	Software operator's panel signal (HS1C)	HS1CO	F077#2	○	○	
	Software operator's panel signal (HS1D)	HS1DO	F077#3	○	○	
	Software operator's panel signal (MP1)	MP1O	F076#0	○	○	
	Software operator's panel signal (MP2)	MP2O	F076#1	○	○	
	Software operator's panel signals (*JV0 to *JV15)	*JVO0 to *JV15O	F079, F080	○	○	
	Software operator's panel signals (*FV0 to *FV7)	*FV0O to *FV7O	F078	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Software operator's panel	Software operator's panel signal (ROV1)	ROV1O	F076#4	○	○	12.1.14
	Software operator's panel signal (ROV2)	ROV2O	F076#5	○	○	
	Software operator's panel signal (BDT)	BDTO	F075#2	○	○	
	Software operator's panel signal (SBK)	SBKO	F075#3	○	○	
	Software operator's panel signal (MLK)	MLKO	F075#4	○	○	
	Software operator's panel signal (DRN)	DRNO	F075#5	○	○	
	Software operator's panel signals (KEY1 to KEY4)	KEYO	F075#6	○	○	
	Software operator's panel signal (*SP)	SPO	F075#7	○	○	
	Software operator's panel general–purpose switch signals	OUT0 to OUT7	F072	○	○	
Second reference position return/3rd, 4th reference position return	2nd reference position return completion signals	ZP21 to ZP24	F096	○	○	4.5
	3rd reference position return completion signals	ZP31 to ZP34	F098	○	○	
	4th reference position return completion signals	ZP41 to ZP44	F100	○	○	
Multi–stage skip	Skip signals	SKIP2 to SKIP6, SKIP7, SKIP8	X004#2 to #6, #0, #1	○	○	14.3.3
Canned cycle (M series)/multiple repetitive turning canned cycle (T series)	Chamfering signal	CDZ	G053#7	○	—	11.8
Chuck/tailstock barrier (T series)	Tailstock barrier selection signal	*TSB	G060#7	○	—	2.3.4
Butt–type reference position return setting	Torque limit reached signals for butt–type reference position setting	CLRCH1 to CLRCH8	F180	○	○	4.6
DNC operation	DNC operation selection signal	DNCI	G043#5	○	○	5.9
	DNC operation selection confirmation signal	MRMT	F003#4	○	○	
Dry run	Dry run signal	DRN	G046#7	○	○	5.3.2
	Dry run check signal	MDRN	F002#7	○	○	
Torque limit skip (T series)	Torque limit reached signals	TRQL1 to TRQL8	F114	○	—	14.3.4
Measurement direct input of tool offset value measured B for 2–spindle lathe	Spindle measurement select signal	S2TLS	G040#5	○	—	14.4
	Spindle 1 under measurement signal	S1MES	F062#3	○	—	
	Spindle 2 under measurement signal	S2MES	F062#4	○	—	
Thread cutting	Thread cutting signal	THRD	F002#3	○	○	6.4.1
Rapid traverse override	Rapid traverse override signals	ROV1, ROV2	G014#0, #1	○	○	7.1.7.1
	1% step rapid traverse override selection signals	HROV	G096#7	○	○	
	1% step rapid traverse override signals	*HROV0 to *HROV6	G096#0 to #6	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
General purpose retract	Retract signal	RTRCT	G066#4	○	○	1.9
	Retract completion signal	RTRCTF	F065#4	○	○	
Direct operation by PMC/Open CNC	Direct operation selection signal	DMMC	G042#7	○	○	15.6
PMC axis control	Control axis selection signals (PMC axis control)	EAX1 to EAX4	G136	○	○	15.1
	Axis control superimposed command signal	EASIP1 to EASIP4	G200	○	○	
	Axis control command signals (PMC axis control)	EC0A to EC6A	G143#0 to #6	○	○	
		EC0B to EC6B	G155#0 to #6	○	○	
		EC0C to EC6C	G167#0 to #6	○	○	
		EC0D to EC6D	G179#0 to #6	○	○	
	Axis control feedrate signals (PMC axis control)	EIF0A to EIF15A	G144, G145	○	○	
		EIF0B to EIF15B	G156, G157	○	○	
		EIF0C to EIF15C	G168, G169	○	○	
		EIF0D to EIF15D	G180, G181	○	○	
	Axis control command read signals (PMC axis control)	EBUFA	G142#7	○	○	
		EBUFB	G154#7	○	○	
		EBUFC	G166#7	○	○	
		EBUFD	G178#7	○	○	
	Axis control data signals (PMC axis control)	EID0A to EID31A	G146 to G149	○	○	
		EID0B to EID31B	G158 to G161	○	○	
		EID0C to EID31C	G170 to G173	○	○	
		EID0D to EID31D	G182 to G185	○	○	
	Axis control command read completion signals (PMC axis control)	EBSYA	F130#7	○	○	
		EBSYB	F133#7	○	○	
		EBSYC	F136#7	○	○	
		EBSYD	F139#7	○	○	
	Reset signals (PMC axis control)	ECLRA	G142#6	○	○	
		ECLRB	G154#6	○	○	
		ECLRC	G166#6	○	○	
		ECLRD	G178#6	○	○	
	Axis control temporary stop signals (PMC axis control)	ESTPA	G142#5	○	○	
		ESTPB	G154#5	○	○	
		ESTPC	G166#5	○	○	
		ESTPD	G178#5	○	○	
	Block stop signals (PMC axis control)	ESBKA	G142#3	○	○	
		ESBKB	G154#3	○	○	
		ESBKC	G166#3	○	○	
		ESBKD	G178#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
PMC axis control	Block stop disable signals (PMC axis control)	EMSBKA	G143#7	○	○	15.1
		EMSBKB	G155#7	○	○	
		EMSBKC	G167#7	○	○	
		EMSBKD	G179#7	○	○	
	Auxiliary function code signals (PMC axis control)	EM11A to EM48A	F132, F142	○	○	
		EM11B to EM48B	F135, F145	○	○	
		EM11C to EM48C	F138, F148	○	○	
		EM11D to EM48D	F141, F151	○	○	
	Auxiliary function strobe signals (PMC axis control)	EMFA	F131#0	○	○	
		EMFB	F134#0	○	○	
		EMFC	F137#0	○	○	
		EMFD	F140#0	○	○	
	Auxiliary function completion signals (PMC axis control)	EFINA	G142#0	○	○	
		EFINB	G154#0	○	○	
		EFINC	G166#0	○	○	
		EFIND	G178#0	○	○	
	Servo off signals (PMC axis control)	ESOFA	G142#4	○	○	
		ESOFB	G154#4	○	○	
		ESOFC	G166#4	○	○	
		ESOFD	G178#4	○	○	
	Buffering disable signals (PMC axis control)	EMBUFA	G142#2	○	○	
		EMBUFB	G154#2	○	○	
		EMBUFC	G166#2	○	○	
		EMBUFD	G178#2	○	○	
	Accumulated zero check signal	ELCKZA	G142#1	○	○	
		ELCKZB	G154#1	○	○	
		ELCKZC	G166#1	○	○	
		ELCKZD	G178#1	○	○	
	Control axis selection status signals (PMC axis control)	*EAXSL	F129#7	○	○	
	In–position signals (PMC axis control)	EINPA	F130#0	○	○	
		EINPB	F133#0	○	○	
		EINPC	F136#0	○	○	
		EINPD	F139#0	○	○	
	Following zero checking signals (PMC axis control)	ECKZA	F130#1	○	○	
		ECKZB	F133#1	○	○	
		ECKZC	F136#1	○	○	
		ECKZD	F139#1	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
PMC axis control	Alarm signals (PMC axis control)	EIALA	F130#2	○	○	15.1
		EIALB	F133#2	○	○	
		EIALC	F136#2	○	○	
		EIALD	F139#2	○	○	
	Axis moving signals (PMC axis control)	EGENA	F130#4	○	○	
		EGENB	F133#4	○	○	
		EGENC	F136#4	○	○	
		EGEND	F139#4	○	○	
	Auxiliary function executing signals (PMC axis control)	EDENA	F130#3	○	○	
		EDENB	F133#3	○	○	
		EDENC	F136#3	○	○	
		EDEND	F139#3	○	○	
	Negative–direction overtravel signals (PMC axis control)	EOTNA	F130#6	○	○	
		EOTNB	F133#6	○	○	
		EOTNC	F136#6	○	○	
		EOTND	F139#6	○	○	
	Positive–direction overtravel signals (PMC axis control)	EOTPA	F130#5	○	○	
		EOTPB	F133#5	○	○	
		EOTPC	F136#5	○	○	
		EOTPD	F139#5	○	○	
	Feedrate override signals (PMC axis control)	*FV0E to *FV7E	G151	○	○	
	Override cancellation signal (PMC axis control)	OVCE	G150#5	○	○	
	Rapid traverse override signals (PMC axis control)	ROV1E, ROV2E	G150#0, #1	○	○	
	Dry run signal (PMC axis control)	DRNE	G150#7	○	○	
	Manual rapid traverse selection signal (PMC axis control)	RTE	G150#6	○	○	
	Override 0% signal (PMC axis control)	EOV0	F129#5	○	○	
	Skip signal (PMC axis control)	ESKIP	X004#6	○	○	
	Distribution completion signals (PMC axis control)	EADEN1 to EADEN4	F112	○	○	
	Buffer full signals (PMC axis control)	EABUFA	F131#1	○	○	
		EABUFB	F134#1	○	○	
		EABUFC	F137#1	○	○	
		EABUFD	F140#1	○	○	
	Controlling signals (PMC axis control)	EACNT1 to EACNT4	F182	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle output control by the PMC	Spindle motor speed command selection signals	SIND	G033#7	○	○	15.4
		SIND2	G035#7	○	○	
		SIND3	G037#7	○	○	
	Spindle motor speed command signals	R01I to R12I	G032#0 to G033#3	○	○	
		R01I2 to R12I2	G034#0 to G035#3	○	○	
		R01I3 to R12I3	G036#0 to G037#3	○	○	
	Spindle motor command polarity selection signals	SSIN	G033#6	○	○	
		SSIN2	G035#6	○	○	
		SSIN3	G037#6	○	○	
	Spindle motor command polarity command signals	SGN	G033#5	○	○	
		SGN2	G035#5	○	○	
		SGN3	G037#5	○	○	
Emergency stop	Emergency stop signal	*ESP	G008#4	○	○	2.1
		*ESP	X008#4	○	○	
FANUC SERVO MOTOR β SERIES I/O LINK OPTION MANUAL HANDLE INTERFACE (PERIPHERAL DEVICE CONTROL)	Manual handle feed generator selection signals	IOLBH2	G199#0	○	○	16.1
	Manual handle feed generator selection signals	IOLBH3	G199#1	○	○	
VRDY off alarm ignore signal	All–axis VRDY off alarm ignore signal	IGNVRY	G066#0	○	○	2.8
	Each–axis VRDY off alarm ignore signal	IGVRY1 to IGVRY4	G192	○	○	
Follow–up	Follow–up signal	*FLWU	G007#5	○	○	1.2.7
Program restart	Program restart signal	SRN	G006#0	○	○	5.7
	Program restart under way signal	SRNMV	F002#4	○	○	
Position switch	Position switch signals	PSW01 to PSW16	F070#0 to F071#7	○	○	1.2.8
Miscellaneous function/2nd auxiliary function	Miscellaneous function code signals	M00 to M31	F010 to F013	○	○	8.1
	Miscellaneous function strobe signal	MF	F007#0	○	○	
	Decode M signals	DM00	F009#7	○	○	
		DM01	F009#6	○	○	
		DM02	F009#5	○	○	
		DM30	F009#4	○	○	
	Spindle function code signals	S00 to S31	F022 to F025	○	○	
	Spindle function strobe signal	SF	F007#2	○	○	
	Tool function code signals	T00 to T31	F026 to F029	○	○	
	Tool function strobe signal	TF	F007#3	○	○	
	2nd auxiliary function code signals	B00 to B31	F030 to F033	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Miscellaneous function/2nd auxiliary function	2nd auxiliary function strobe signal	BF	F007#4	○	—	8.1
		BF	F007#7	—	○	
	End signal	FIN	G004#3	○	○	
	Distribution end signal	DEN	F001#3	○	○	
Auxiliary function lock	Auxiliary function lock signal	AFL	G005#6	○	○	8.2
	Auxiliary function lock check signal	MAFL	F004#4	○	○	
Polygonal turning	Polygon synchronization under way signal	PSYN	F063#7	○	—	6.9.1
Machine lock	All–axis machine lock signal	MLK	G044#1	○	○	5.3.1
	Each–axis machine lock signal	MLK1 to MLK4	G108	○	○	
	All–axis machine lock check signal	MMLK	F004#1	○	○	
Manual absolute on/off	Manual absolute signal	*ABSM	G006#2	○	○	5.4
	Manual absolute check signal	MABSM	F004#2	○	○	
Multi–spindle control (T series)	Spindle selection signals	SWS1	G027#0	○	○	9.9
		SWS2	G027#1	○	○	
		SWS3	G027#2	○	○	
	Individual spindle stop signals	*SSTP1	G027#3	○	○	
		*SSTP2	G027#4	○	○	
		*SSTP3	G027#5	○	○	
	Gear selection signals (input)	GR21	G029#0	○	○	
	2nd position coder selection signal	PC2SLC	G028#7	○	○	
	Spindle enable signals	ENB2	F038#2	○	○	
		ENB3	F038#3	○	○	
Mirror image	Mirror image signals	MI1 to MI4	G106	○	○	1.2.6
	Mirror image check signals	MMI1 to MMI4	F108	○	○	
Memory protection key	Memory protection signals	KEY1 to KEY4	G046#3 to #6	○	○	12.2.3
Mode selection	Mode selection signals	MD1, MD2, MD4	G043#3 to #2	○	○	2.6
	Manual data input selection check signal	MMDI	F003#3	○	○	
Mode selection	Automatic operation selection check signal	MMEM	F003#5	○	○	
	Memory edit selection check signal	MEDT	F003#6	○	○	
	Manual handle feed selection check signal	MH	F003#1	○	○	
	Incremental feed selection check signal	MINC	F003#0	○	○	
	Jog feed selection check signal	MJ	F003#2	○	○	
	TEACH IN selection check signal	MTCHIN	F003#7	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Rigid tapping	Rigid tapping signal	RGTAP	G061#0	○	○	9.10
	Spindle rotation direction signals	RGSPP	F065#0	—	○	
		RGSPM	F065#1	—	○	
	Rigid tapping–in–progress signal	RTAP	F076#3	○	○	
	Rigid tapping spindle selection signals	RGTSP1, RGTSP2	G061#4, #5	○	—	
Retraction for rigid tapping	Rigid tapping retraction start signal	RTNT	G062#6	—	○	5.11
	Rigid tapping retraction completion signal	RTPT	F066#1	—	○	
Reset and rewind	External reset signal	ERS	G008#7	○	○	5.2
	Reset & rewind signal	RRW	G008#6	○	○	
	Resetting signal	RST	F001#1	○	○	
	Rewinding signal	RWD	F000#0	○	○	
Interrupt type custom macro	Interrupt signal for custom macro	UINT	G053#3	○	○	11.5.2

A.2.2 List of Signals (In Order of Symbols)

○	:	Available
–	:	Unavailable

Group	Symbol	Signal name	Address	T series	M series	Reference item
*	*+ED1 to *+ED4	External deceleration signal	G118	○	○	7.1.9
	*+ED21 to *+ED24	External deceleration signal 2	G101#0 to #3	○	○	7.1.9
	*+ED31 to *+ED34	External deceleration signal 3	G107#0 to #3	○	○	7.1.9
	*+L1 to *+L4	Overtravel signal	G114	○	○	2.3.1
	*-ED1 to *-ED4	External deceleration signal	G120	○	○	7.1.9
	*–ED21 to *–ED24	External deceleration signal 2	G103#0 to #3	○	○	7.1.9
	*–ED31 to *–ED34	External deceleration signal 3	G109#0 to #3	○	○	7.1.9
	*-L1 to *-L4	Overtravel signal	G116	○	○	2.3.1
	*ABSM	Manual absolute signal	G006#2	○	○	5.4
	*BECLP	B–axis clamp completion signal	G038#7	–	○	11.10
	*BEUCP	B–axis unclamp completion signal	G038#6	–	○	11.10
	*BSL	Block start interlock signal	G008#3	○	○	2.5
	*CRTOF	Automatic erase CRT screen display cancel signal	G062#1	○	○	12.1.17
	*CSL	Cutting block start interlock signal	G008#1	○	○	2.5
	*DEC1 to *DEC4	Deceleration signal for reference position return	X009	○	○	4.1
	*EAXSL	Control axis selection status signal(PMC axis control)	F129#7	○	○	15.1
	*ESP	Emergency stop signal	X008#4	○	○	2.1
	*ESP		G008#4	○	○	
	*ESPA	Emergency stop signal (serial spindle)	G071#1	○	○	9.2
	*ESPB		G075#1	○	○	
	*FLWU	Follow–up signal	G007#5	○	○	1.2.7
	*FV0 to *FV7	Feedrate override signal	G012	○	○	7.1.7.2
	*FV0E to *FV7E	Feedrate override signal (PMC axis control)	G151	○	○	15.1
	*FV0O to *FV7O	Software operator's panel signal(*FV0 to *FV7)	F078	○	○	12.1.14
	*HROV0 to *HROV6	1% step rapid traverse override signal	G096#0 to #6	○	○	7.1.7.1
	*IT	Interlock signal	G008#0	○	○	2.5
	*IT1 to *IT4	Interlock signal for each axis	G130	○	○	2.5
	*JV0 to *JV15	Manual feedrate override signal	G010,G011	○	○	3.1
	*JV0O to *JV15O	Software operator's panel signal(*JV0 to *JV15)	F079,F080	○	○	12.1.14
	*PLSST	Polygon spindle stop signal	G038#0	○	–	6.10.2
	*SCPF	Spindle clamp completion signal	G028#5	○	–	9.7
	*SP	Feed hold signal	G008#5	○	○	5.1
	*SSTP	Spindle stop signal	G029#6	○	○	9.3

Group	Symbol	Signal name	Address	T series	M series	Reference item
*	*SSTP1	Individual spindle stop signals	G027#3	○	○	9.9
	*SSTP2		G027#4	○	○	
	*SSTP3		G027#5	○	○	
	*SUCPF	Spindle unclamp completion signal	G028#4	○	–	9.7
	*TLV0 to *TLV9	Tool life count override signal	G049#0 to G050#1	–	○	10.3
	*TSB	Tailstock barrier select signal	G060#7	○	–	2.3.4
+	+EXL1 to +EXL4	Axis direction dependent stored stroke limit switch signal	G104	○	○	2.3.2
	+J1 to +J4	Feed axis and direction selection signal	G100	○	○	3.1
	+J1O to +J4O	Software operator's panel signal(+J1 to +J4)	F081#0,#2,#4, #6	○	○	12.1.14
	+LM1 to +LM4	Stroke limit external setting signal	G110	–	○	2.3.2
	+MIT1,+MIT2	Manual feed interlock signal for each axis	X004#2,#4	○	–	2.5
	+MIT1,+MIT2	Tool offset write signal	X004#2,#4	○	–	14.4.2
	+MIT1 to +MIT4	Interlock signal for each axis and direction	G132#0 to #3	–	○	2.5
	+OT1 to +OT4	Stroke limit reached signals	F124	–	○	2.3.2
–	–EXL1 to –EXL4	Axis direction dependent stored stroke limit switch signal	G105	○	○	2.3.2
	–J1 to -J4	Feed axis and direction selection signal	G102	○	○	3.1
	–J1O to –J4O	Software operator's panel signal(–J1 to –J4)	F081#1,#3,#5, #7	○	○	12.1.14
	–LM1 to –LM4	Stroke limit external setting signal	G112	–	○	2.3.2
	–MIT1,–MIT2	Manual feed interlock signal for each axis	X004#3,#5	○	–	2.5
	–MIT1,–MIT2	Tool offset write signal	X004#3,#5	○	–	14.4.2
	–MIT1 to –MIT4	Interlock signal for each axis and direction	G134#0 to #3	–	○	2.5
	–OT1 to –OT4	Stroke limit reached signals	F126	–	○	2.3.2
A	ABTQSV	Servo axis abnormal load detected signal	F090#0	○	○	2.9
	ABTSP1	First–spindle abnormal load detected signal	F090#1	○	○	2.9
	ABTSP2	Second–spindle abnormal load detected signal	F090#2	○	○	2.9
	AFL	Miscellaneous function lock signal	G005#6	○	○	8.2
	AICC	AI advanced control mode signal	F062#0	–	○	7.1.13
	AL	Alarm signal	F001#0	○	○	2.4
	ALMA	Alarm signal (serial spindle)	F045#0	○	○	9.2
	ALMB		F049#0	○	○	
	AR0 to AR15	Actual spindle speed signal	F040,F041	○	–	9.6
	ARSTA	Alarm reset signal (serial spindle)	G071#0	○	○	9.2
	ARSTB		G075#0	○	○	
B	B00 to B31	2nd auxiliary function code signal	F030 to F033	○	○	8.1
	BAL	Battery alarm signal	F001#2	○	○	2.4
	BCLP	B–axis clamp signal	F061#1	–	○	11.10

Group	Symbol	Signal name	Address	T series	M series	Reference item
B	BDT1,BDT2 to BDT9	Optional block skip signal	G044#0,G045	○	○	5.5
	BDTO	Software operator's panel signal(BDT)	F075#2	○	○	12.1.14
	BF	2nd auxiliary function strobe signal	F007#4	○	–	8.1
	BF		F007#7	–	○	
	BFIN	2nd auxiliary function completion signal	G005#4	○	–	8.4
	BFIN		G005#7	–	○	
	BGEACT	Background busy signal	F053#4	○	○	13.5, 13.3
	BGEN	Power Mate background busy signal	G092#4	○	○	13.5
	BGIALM	Power Mate read/write alarm signal	G092#3	○	○	
	BGION	Power Mate read/write inprogress signal	G092#2	○	○	
	BUCLP	B–axis unclamp signal	F061#0	–	○	11.10
C	CDF1 to CDF4	Cs contour control mode fine acceleration/deceleration function disable signal	G127#0 to #3	○	○	9.8.1
	CDZ	Chamferring signal	G053#7	○	–	11.8
	CFINA	Spindle switch completion signal (serial spindle)	F046#1	○	○	9.2
	CFINB		F050#1	○	○	
	CHPA	Power line switch signal (serial spindle)	F046#0	○	○	9.2
	CHPB		F050#0	○	○	
	CLRCH1 to CLRCH4	Torque limit reach signals for butt–type reference position setting	F180	○	○	4.6
	CON	Cs contour control change signal	G027#7	○	○	9.8.1
	CSFI1	Cs–axis coordinate establishment request signal	G274#4	○	○	9.8.2
	CSFO1	Cs–axis coordinate establishment alarm signal	F274#4	○	○	9.8.2
	CSPENA	Cs–axis coordinate establishment status signal	F048#4	○	○	9.8.2
	CSS	Constant surface speed signal	F002#2	○	○	9.4
	CTH1A,CTH2A	Clutch/gear signal (serial spindle)	G070#3,#2	○	○	9.2
	CTH1B,CTH2B		G074#3,#2	○	○	
	CUT	Cutting feed signal	F002#6	○	○	2.7
D	DEFMDA	Differential mode command signal (serial spindle)	G072#3	○	○	9.2
	DEFMDB		G076#3	○	○	
	DEN	Distribution end signal	F001#3	○	○	8.1
	DM00	Decode M signal	F009#7	○	○	8.1
	DM01		F009#6	○	○	
	DM02		F009#5	○	○	
	DM30		F009#4	○	○	
	DMMC	Direct operation select signal	G042#7	○	○	15.6
	DNCI	DNC operation select signal	G043#5	○	○	5.9
	DRN	Dry run signal	G046#7	○	○	5.3.2
	DRNE	Dry run signal (PMC axis control)	G150#7	○	○	15.1
	DRNO	Software operator's panel signal(DRN)	F075#5	○	○	12.1.14

Group	Symbol	Signal name	Address	T series	M series	Referen ce item
D	DSCNA	Disconnection detection disbale signal (serial spindle)	G073#4	○	○	9.2
	DSCNB		G077#4	○	○	
	DTCH1 to DTCH4	Controlled axis detach signals	G124#0 to #3	○	○	1.2.4
E	EA0 to EA6	Address signal for external data input	G002#0 to #6	○	○	15.2
	EABUFA	Buffer full signal (PMC axis control)	F131#1	○	○	15.1
	EABUFB		F134#1	○	○	
	EABUFC		F137#1	○	○	
	EABUFD		F140#1	○	○	
	EACNT1 to EACNT4	Controlling signal (PMC axis control)	F182	○	○	15.1
	EADEN1 to EADEN4	Distribution completion signal(PMC axis control)	F112	○	○	
	EAX1 to EAX4	Control axis select signal (PMC axis control)	G136	○	○	
	EASIP1 to EASIP4	Axis control superimposed command signal	G200	○	○	
	EBSYA	Axis control command read completion signal (PMC axis control)	F130#7	○	○	15.1
	EBSYB		F133#7	○	○	
	EBSYC		F136#7	○	○	
	EBSYD		F139#7	○	○	
	EBUFA	Axis control command read signal(PMC axis control)	G142#7	○	○	15.1
	EBUFB		G154#7	○	○	
	EBUFC		G166#7	○	○	
	EBUFD		G178#7	○	○	
	EC0A to EC6A	Axis control command signal (PMC axis control)	G143#0 to #6	○	○	
	EC0B to EC6B		G155#0 to #6	○	○	
	EC0C to EC6C		G167#0 to #6	○	○	
	EC0D to EC6D		G179#0 to #6	○	○	
	ECKZA	Following zero checking signal (PMC axis control)	F130#1	○	○	
	ECKZB		F133#1	○	○	
	ECKZC		F136#1	○	○	
	ECKZD		F139#1	○	○	
	ECLRA	Reset signal (PMC axis control)	G142#6	○	○	
	ECLRB		G154#6	○	○	
	ECLRC		G166#6	○	○	
	ECLRD		G178#6	○	○	
	ED0 to ED15	Data signal for external data input	G000,G001	○	○	15.2
	EDENA	Auxiliary function executing signal (PMC axis control)	F130#3	○	○	15.1
	EDENB		F133#3	○	○	
	EDENC		F136#3	○	○	
	EDEND		F139#3	○	○	
	EDGN	Slave diagnosis selection signal	F177#7	○	○	13.5

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EF	External operation signal	F008#0	–	○	11.7
	EFD	External operation signal for high–speed interface	F007#1	–	○	8.4
	EFIN	External operation function completion signal	G005#1	–	○	
	EFINA	Auxiliary function completion signal (PMC axis control)	G142#0	○	○	15.1
	EFINB		G154#0	○	○	
	EFINC		G166#0	○	○	
	EFIND		G178#0	○	○	
	EGENA	Axis moving signal (PMC axis control)	F130#4	○	○	15.1
	EGENB		F133#4	○	○	
	EGENC		F136#4	○	○	
	EGEND		F139#4	○	○	
	EIALA	Alarm signal (PMC axis control)	F130#2	○	○	
	EIALB		F133#2	○	○	
	EIALC		F136#2	○	○	
	EIALD		F139#2	○	○	
	EID0A to EID31A	Axis control data signal (PMC axis control)	G146 to G149	○	○	
	EID0B to EID31B		G158 to G161	○	○	
	EID0C to EID31C		G170 to G173	○	○	
	EID0D to EID31D		G182 to G185	○	○	
	EIF0A to EIF15A	Axis control feedrate signal (PMC axis control)	G144,G145	○	○	
	EIF0B to EIF15B		G156,G157	○	○	
	EIF0C to EIF15C		G168,G169	○	○	
	EIF0D to EIF15D		G180,G181	○	○	
	EINPA	In–position signal (PMC axis control)	F130#0	○	○	
	EINPB		F133#0	○	○	
	EINPC		F136#0	○	○	
	EINPD		F139#0	○	○	
	EKC0 to EKC7	Key code signal	G098	○	○	15.5
	EKENB	Key code read completion signal	F053#7	○	○	
	EKSET	key code read signal	G066#7	○	○	
	ELCKZA	Accumulated zero check signal	G142#1	○	○	15.1
	ELCKZB		G154#1	○	○	
	ELCKZC		G166#1	○	○	
	ELCKZD		G178#1	○	○	
	EM11A to EM48A	Auxiliary function code signal (PMC axis control)	F132,F142	○	○	
	EM11B to EM48B		F135,F145	○	○	
	EM11C to EM48C		F138,F148	○	○	
	EM11D to EM48D		F141,F151	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EMBUFA	Buffering disable signal (PMC axis control)	G142#2	○	○	15.1
	EMBUFB		G154#2	○	○	
	EMBUFC		G166#2	○	○	
	EMBUFD		G178#2	○	○	
	EMFA	Auxiliary function strobe signal (PMC axis control)	F131#0	○	○	
	EMFB		F134#0	○	○	
	EMFC		F137#0	○	○	
	EMFD		F140#0	○	○	
	EMSBKA	Block stop disable signal (PMC axis control)	G143#7	○	○	15.1
	EMSBKB		G155#7	○	○	
	EMSBKC		G167#7	○	○	
	EMSBKD		G179#7	○	○	
	ENB	Spindle enable signal	F001#4	○	○	9.3
	ENB2		F038#2	○	–	9.9
	ENB3		F038#3	○	–	
	ENBKY	External key input mode selection signal	G066#1	○	○	15.5
	EOTNA	Negative–direction overtravel signal (PMC axis control)	F130#6	○	○	15.1
	EOTNB		F133#6	○	○	
	EOTNC		F136#6	○	○	
	EOTND		F139#6	○	○	
	EOTPA	Positive–direction overtravel signal (PMC axis control)	F130#5	○	○	
	EOTPB		F133#5	○	○	
	EOTPC		F136#5	○	○	
	EOTPD		F139#5	○	○	
	EOV0	Override 0% signal (PMC axis control)	F129#5	○	○	
	EPARM	Slave parameter selection signal	F177#6	○	○	13.5
	EPN0 to EPN13	Expanded workpiece number search signals	G024#0 to G025#5	○	○	15.3.2
	EPNS	Expanded workpiece number search start signal	G025#7	○	○	
	EPRG	Slave program selection signal	F177#4	○	○	13.5
	ERDIO	Slave external read start signal	F177#1	○	○	
	EREND	Read completion signal for external data input	F060#0	○	○	15.2
	ERS	External reset signal	G008#7	○	○	5.2
	ESBKA	Block stop signal (PMC axis control)	G142#3	○	○	15.1
	ESBKB		G154#3	○	○	
	ESBKC		G166#3	○	○	
	ESBKD		G178#3	○	○	
	ESEND	Search completion signal for external data input	F060#1	○	○	15.2

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	ESKIP	Skip signal (PMC axis control)	X004#6	○	○	15.1
	ESOFA	Servo off signal (PMC axis control)	G142#4	○	○	
	ESOFB		G154#4	○	○	
	ESOFC		G166#4	○	○	
	ESOFD		G178#4	○	○	
	ESTB	Read signal for external data input	G002#7	○	○	15.2
	ESCAN	Search cancel signal for external data input	F060#2	○	○	
	ESTPA	Axis control temporary stop signal (PMC axis control)	G142#5	○	○	15.1
	ESTPB		G154#5	○	○	
	ESTPC		G166#5	○	○	
	ESTPD		G178#5	○	○	
	ESTPIO	Slave read/write stop signal	F177#2	○	○	13.5
	EVAR	Slave macro variable selection signal	F177#5	○	○	
	EWTIO	Slave external write start signal	F177#3	○	○	
	EXLM	Stored stroke limit select signal	G007#6	○	○	2.3.2
	EXOFA	Motor activation off status signal (serial spindle)	F047#4	○	○	9.2
	EXOFB		F051#4	○	○	
	EXRD	External read start signal	G058#1	○	○	13.5, 13.3
	EXSTP	External read/punch stop signal	G058#2	○	○	
	EXWT	External punch start signal	G058#3	○	○	
F	F1D	F1–digit feed select signal	G016#7	–	○	7.1.5
	FIN	Completion signal	G004#3	○	○	8.1
	FSCSL	Cs contour control change completion signal	F044#1	○	○	9.8.1
	FSPPH	Spindle phase synchronous control completion signal	F044#3	○	○	9.11
	FSPSY	Spindle synchronous speed control completion signal	F044#2	○	○	
G	G08MD	Advanced preview control mode signal	F066#0	–	○	7.1.12
	GOQSM	Tool offset value write mode select signal	G039#7	○	–	14.4.2
	GR1,GR2	Gear selection signal (input)	G028#1,#2	○	○	9.3
	GR1O,GR2O,GR3O	Gear selection signal (output)	F034#0 to #2	–	○	
	GR21	Gear selection signal (input)	G029#0	○	○	9.9
H	HCAB2	Hard copy stop request acceptance flag	F061#2	○	○	13.6
	HCABT	Hard copy stop request signal	G067#6	○	○	13.6
	HCEXE	Hard copy in–progress signal	F061#3	○	○	13.6
	HCREQ	Hard copy request signal	G067#7	○	○	13.6
	HDO0	High–speed skip status signal	F122#0	○	○	14.3.2
	HROV	1% step rapid traverse override select signal	G096#7	○	○	7.1.7.1
	HS1A to HS1D	Manual handle feed axis selection signal	G018#0 to #3	○	○	3.2

Group	Symbol	Signal name	Address	T series	M series	Reference item
H	HS1AO	Software operator's panel signal(HS1A)	F077#0	○	○	12.1.14
	HS1BO	Software operator's panel signal(HS1B)	F077#1	○	○	
	HS1CO	Software operator's panel signal(HS1C)	F077#2	○	○	
	HS1DO	Software operator's panel signal(HS1D)	F077#3	○	○	
	HS1IA to HS1ID	Manual handle interruption axis select signal	G041#0 to #3	○	○	3.3
	HS2A to HS2D	Manual handle feed axis selection signal	G018#4 to #7	○	○	3.2
	HS2IA to HS2ID	Manual handle interruption axis select signal	G041#4 to #7	○	○	3.3
	HS3A to HS3D	Manual handle feed axis selection signal	G019#0 to #3	○	○	3.2
	HS3IA to HS3ID	Manual handle interruption axis select signal	G042#0 to #3	○	○	3.3
I	IGNVRY	All–axis VRDY OFF alarm ignore signal	G066#0	○	○	2.8
	IGVRY1 to IGVRY4	Each–axis VRDY OFF alarm ignore signal	G192	○	○	
	INCH	Inch input signal	F002#0	○	○	11.4
	INCMDA	Incremental command external setting type orientation signal (serial spindle)	G072#5	○	○	9.2
	INCMDB		G076#5	○	○	
	INCSTA	Incremental method orientation signal (serial spindle)	F047#1	○	○	
	INCSTB		F051#1	○	○	
	INDXA	Orientation stop position change signal (serial spindle)	G072#0	○	○	
	INDXB		G076#0	○	○	
	INHKY	Key input disable signal	F053#0	○	○	15.5
	INP1 to INP4	In–position signal	F104	○	○	7.2.5.1
	INTGA	Signal for controlling velocity integration (serial spindle)	G071#5	○	○	9.2
	INTGB		G075#5	○	○	
	IOLACK	I/O Link confirmation signal	G092#0	○	○	13.5
	IOLBH2	Manual handle feed generator selection signals	G199#0	○	○	16.1
	IOLBH3		G199#1	○	○	
	IOLNK	Slave I/O Link selection signal	F177#0	○	○	13.5
	IOLS	I/O Link specification signal	G092#1	○	○	13.5
	IUDD1 to IUDD4	Abnormal load detection ignore signal	G125	○	○	2.9
K	KEY1 to KEY4	Memory protect signal	G046#3 to #6	○	○	12.2.3
	KEYO	Software operator's panel signal(KEY1 to KEY4)	F075#6	○	○	12.1.14
L	LDT1A	Load detection signal 1 (serial spindle)	F045#4	○	○	9.2
	LDT1B		F049#4	○	○	
	LDT2A	Load detection signal 2 (serial spindle)	F045#5	○	○	
	LDT2B		F049#5	○	○	
M	M00 to M31	Miscellaneous function code signal	F010 to F013	○	○	8.1
	M200 to M215	2nd M function code signal	F014 to F015	○	○	8.3
	M300 to M315	3rd M function code signal	F016 to F017	○	○	8.3
	MA	CNC ready signal	F001#7	○	○	2.2
	MABSM	Manual absolute check signal	F004#2	○	○	5.4

Group	Symbol	Signal name	Address	T series	M series	Reference item
M	MAFL	Miscellaneous function lock check signal	F004#4	○	○	8.2
	MBDT1,MBDT2 to MBDT9	Optional block skip check signal	F004#0,F005	○	○	5.5
	MCFNA	Power line switch completion signal (serial spindle)	G071#3	○	○	9.2
	MCFNB		G075#3	○	○	
	MCHK	Check mode handle valid signal	G067#3	○	–	5.3.4
	MD1,MD2,MD4	Mode selection signal	G043#0 to #2	○	○	2.6
	MD1O	Software operator's panel signal(MD1)	F073#0	○	○	12.1.14
	MD2O	Software operator's panel signal(MD2)	F073#1	○	○	
	MD4O	Software operator's panel signal(MD4)	F073#2	○	○	
	MDRN	Dry run check signal	F002#7	○	○	5.3.2
	MDTCH1 to MDTCH4	Controlled axis detach status signals	F110#0 to #3	○	○	1.2.4
	MEDT	Memory edit select check signal	F003#6	○	○	2.6
	MF	Auxiliary function strobe signal	F007#0	○	○	8.1
	MF2	2nd M function strobe signal	F008#4	○	○	8.3
	MF3	3rd M function strobe signal	F008#5	○	○	
	MFIN	Auxiliary function completion signal	G005#0	○	○	8.4
	MFIN2	2nd M function completion signal	G004#4	○	○	
	MFIN3	3rd M function completion signal	G004#5	○	○	
	MFNHGA	Main spindle MCC status signal while changing spindles signal (serial spindle)	G072#6	○	○	9.2
	MFNHGB		G076#6	○	○	
	MH	Manual handle feed select check signal	F003#1	○	○	2.6
	MI1 to MI4	Mirror image signal	G106	○	○	1.2.6
	MINC	Incremental feed select check signal	F003#0	○	○	2.6
	MINP	External program input start signal	G058#0	○	○	13.4
	MJ	JOG feed select check signal	F003#2	○	○	2.6
	MLK	All–axis machine lock signal	G044#1	○	○	5.3.1
	MLK1 to MLK4	Each–axis machine lock signal	G108	○	○	
	MLKO	Software operator's panel signal(MLK)	F075#4	○	○	12.1.14
	MMDI	Manual data input select check signal	F003#3	○	○	2.6
	MMEM	Automatic operation select check signal	F003#5	○	○	
	MMI1 to MMI4	Mirror image check signal	F108	○	○	1.2.6
	MMLK	All–axis machine lock check signal	F004#1	○	○	5.3.1
	MMOD	Check mode signal	G067#2	○	–	5.3.4
	MNCHG	Inversion inhibition signal	F091#1	○	–	
	MORA1A	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	F046#6	○	○	9.2
	MORA1B		F050#6	○	○	
	MORA2A	Signal for approximate spindle orientation with a magnetic sensor (serial spindle)	F046#7	○	○	
	MORA2B		F050#7	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
M	MORCMA	Command for spindle orientaion with a magnetic sensor (serial spindle)	G073#0	○	○	9.2
	MORCMB		G077#0	○	○	
	MP1,MP2	Manual handle feed amount selection signal (incremental feed signal)	G019#4,#5	○	○	3.2
	MP1O	Software operator's panel signal(MP1)	F076#0	○	○	12.1.14
	MP2O	Software operator's panel signal(MP2)	F076#1	○	○	
	MPOFA	Motor power stop signal (serial spindle)	G073#2	○	○	9.2
	MPOFB		G077#2	○	○	
	MRDYA	Machine ready signal (serial spindle)	G070#7	○	○	
	MRDYB		G074#7	○	○	
	MREF	Manual reference position return selection check signal	F004#5	○	○	4.1
	MRMT	DNC operation select check signal	F003#4	○	○	5.9
	MRVM	Check mode backward movement inhibition signal	G067#1	○	–	5.3.4
	MRVMD	Check mode backward movement signal	F091#0	○	–	
	MRVSP	Backward movement inhibition signal	F091#2	○	–	
	MSBK	Single block check signal	F004#3	○	○	5.3.3
	MTCHIN	TEACH IN select check signal	F003#7	○	○	2.6
	MV1 to MV4	Axis moving signal	F102	○	○	1.2.5
	MVD1 to MVD4	Axis moving direction signal	F106	○	○	
N	NOINP1 to NOINP4	Each axis in–position check disable signal	G359#0 to #3	○	○	7.2.5.3
	NOINPS	In–position check disable signal	G023#5	○	○	7.2.5.3
	NOZAGC	Perpendicular/angular axis control disable signal	G063#5	○	○	1.8
	NPOS1 to NPOS4	Position display neglect signal	G198	○	○	12.1.10
	NRROA	Short–distant movement command while changing the orientation stop position signal (serial spindle)	G072#2	○	○	9.2
	NRROB		G076#2	○	○	
O	OFN0 to OFN5,OFN6	Tool offset number select signal	G039#0 to #5,G040#0	○	–	14.4.2
	OP	Automatic operation signal	F000#7	○	○	5.1
	ORARA	Orientation completion signal (serial spindle)	F045#7	○	○	9.2
	ORARB		F049#7	○	○	
	ORCMA	Orientation command signal (serial spindle)	G070#6	○	○	
	ORCMB		G074#6	○	○	
	OTP1 to OTP4	Overtravel alarm in–progress signal	F124#0 to #3	○	○	2.3.1
	OUT0 to OUT7	Software operator's panel general–purpose switch signal	F072	○	○	12.1.14
	OVC	Override cancel signal	G006#4	○	○	7.1.7.3
	OVCE	Override cancellation signal (PMC axis control)	G150#5	○	○	15.1
	OVRA	Analog override command signal (serial spindle)	G072#4	○	○	9.2, 9.15
	OVRB		G076#4	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
P	PBATL	Absolute position detector battery voltage low alarm signal	F172#7	○	○	1.4.2
	PBATZ	Absolute position detector battery voltage zero alarm signal	F172#6	○	○	
	PC1DTA	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	F047#0	○	○	9.2
	PC1DTB		F051#0	○	○	
	PC2SLC	2nd position coder selection signal	G028#7	○	○	9.9
	PECK2	Small–diameter peck drilling in progress signal	F066#5	–	○	11.14
	PN1,PN2,PN4,PN8, PN16	Workpiece number search signal	G009#0 to 4	○	○	15.3
	PORA2A	Signal for approximate spindle orientation with a position coder (serial spindle)	F046#5	○	○	9.2
	PORA2B		F050#5	○	○	
	PRC	Position record signal	G040#6	○	–	14.4.1
	PRGDPL	program screen display mode signal	F053#1	○	○	15.5
	PRTSF	Target parts count reached signal	F062#7	○	○	12.1.11
	PSAR	Spindle polygon speed arrival signal	F063#2	○	–	6.10.2
	PSE1	Master axis not arrival signal	F063#0	○	–	
	PSE2	Polygon synchronous axis not arrival signal	F063#1	○	–	
	PSW01 to PSW16	Position switch signal	F070#0 to F071#7	○	○	1.2.9
	PSYN	Polygon synchronization under way signal	F063#7	○	–	6.9.1
R	R01I to R12I	Spindle motor speed command signal	G032#0 to G033#3	○	○	15.4
	R01I2 to R12I2		G034#0 to G035#3	○	○	
	R01I3 to R12I3		G036#0 to G037#3	○	○	
	R01O to R12O	S12–bit code signal	F036#0 to F037#3	○	○	9.3
	RCFNA	Output switch completion signal (serial spindle)	F046#3	○	○	9.2
	RCFNB		F050#3	○	○	
	RCHA	Power line status check signal (serial spindle)	G071#7	○	○	
	RCHB		G075#7	○	○	
	RCHHGA	High–output MCC status signal while a magnetic sensor (serial spindle)	G072#7	○	○	
	RCHHGB		G076#7	○	○	
	RCHPA	Output switch signal (serial spindle)	F046#2	○	○	
	RCHPB		F050#2	○	○	
	RGSPM	Spindle rotation direction signal	F065#1	–	○	9.10
	RGSPP		F065#0	–	○	
	RGTAP	Rigid tapping signal	G061#0	○	○	
	RGTSP1,RGTSP2	Rigid tapping spindle selection signal	G061#4,#5	○	–	
	RLSOT	Stroke check release signal	G007#7	–	○	2.3.2
	RLSOT3	Stroke check 3 release signal	G007#4	○	○	2.3.3

Group	Symbol	Signal name	Address	T series	M series	Reference item
R	ROTAA	Rotation direction command while changing the orientation stop position signal (serial spindle)	G072#1	○	○	9.2
	ROTAB		G076#1	○	○	
	ROV1,ROV2	Rapid traverse override signal	G014#0,#1	○	○	7.1.7.1
	ROV1E,ROV2E	Rapid traverse override signal(PMC axis control)	G150#0,#1	○	○	15.1
	ROV1O	Software operator's panel signal(ROV1)	F076#4	○	○	12.1.14
	ROV2O	Software operator's panel signal(ROV2)	F076#5	○	○	
	RPALM	Read/punch alarm signal	F053#3	○	○	13.5, 13.3
	RPBSY	Read/punch in–progress signal	F053#2	○	○	
	RPDO	Rapid traversing signal	F002#1	○	○	2.7
	RRW	Reset&rewind signal	G008#6	○	○	5.2
	RSLA	Output switch request signal (serial spindle)	G071#6	○	○	9.2
	RSLB		G075#6	○	○	
	RST	Reset signal	F001#1	○	○	5.2
	RT	Manual rapid traverse selection signal	G019#7	○	○	3.1
	RTAP	Rigid tapping in–progress signal	F076#3	○	○	9.10
	RTE	Manual rapid traverse selection signal (PMC axis control)	G150#6	○	○	15.1
	RTO	Software operator's panel signal(RT)	F077#6	○	○	12.1.14
	RTNT	Rigid tapping retraction start signal	G062#6	–	○	5.11
	RTPT	Rigid tapping retraction completiont signal	F066#1	–	○	
	RTRCT	Retract signal	G066#4	–	○	1.9
	RTRCTF	Retract completion signal	F065#4	–	○	1.9
	RVS	Retrace signal	G007#0	–	○	11.15
	RVSL	Retrace–in–progress signal	F082#2	–	○	
	RWD	Rewinding signal	F000#0	○	○	5.2
S	S00 to S31	Spindle speed code signal	F022 to F025	○	○	8.1
	S1MES	Spindle 1 under measurement signal	F062#3	○	–	14.4
	S2MES	Spindle 2 under measurement signal	F062#4	○	–	
	S2TLS	Spindle measurement select signal	G040#5	○	–	
	SA	Servo ready signal	F000#6	○	○	2.2
	SAR	Spindle speed arrival signal	G029#4	○	○	9.3
	SARA	Speed arrival signal (serial spindle)	F045#3	○	○	9.2
	SARB		F049#3	○	○	
	SBK	Single block signal	G046#1	○	○	5.3.3
	SBKO	Software operator's panel signal(SBK)	F075#3	○	○	12.1.14
	SCLP	Spindle clamp signal	F038#0	○	–	9.7
	SDTA	Speed detection signal (serial spindle)	F045#2	○	○	9.2
	SDTB		F049#2	○	○	
	SF	Spindle speed strobe signal	F007#2	○	○	8.1
	SFIN	Spindle function completion signal	G005#2	○	○	8.4

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SFRA	CW command signal (serial spindle)	G070#5	○	○	9.2
	SFRB		G074#5	○	○	
	SGN	Spindle motor command polarity select signal	G033#5	○	○	15.4
	SGN2		G035#5	○	○	
	SGN3		G037#5	○	○	
	SHA00 to SHA11	Spindle orientation external stop position command signal	G078#0 to G079#3	○	○	9.12
	SHB00 to SHB11		G080#0 to G081#3	○	○	
	SIND	Spindle motor speed command select signal	G033#7	○	○	15.4
	SIND2		G035#7	○	○	
	SIND3		G037#7	○	○	
	SKIP	Skip signal	X004#7	○	○	14.3.1
		Overload torque signal	X004#7	–	○	11.14
	SKIP2 to SKIP6, SKIP7,SKIP8	Skip signal	X004#2 to #6, #0,#1	○	○	14.3.3
	SKIPP	Skip signal	G006#6	○	–	14.3.1
	SLVA	Slave operation command signal (serial spindle)	G073#1	○	○	9.2
	SLVB		G077#1	○	○	
	SLVSA	Slave operation status signal (serial spindle)	F046#4	○	○	
	SLVSB		F050#4	○	○	
	SMZ	Error detect signal	G053#6	○	–	7.2.5.3
	SOCNA	Soft start/stop cancel signal (serial spindle)	G071#4	○	○	9.2
	SOCNB		G075#4	○	○	
	SOR	Spindle orientation signal	G029#5	○	○	9.3
	SOV0 to SOV7	Spindle speed override signal	G030	○	○	
	SPAL	Spindle fluctuation detection alarm signal	F035#0	○	○	9.5
	SPL	Feed hold lamp signal	F000#4	○	○	5.1
	SPO	Software operator's panel signal(*SP)	F075#7	○	○	12.1.14
	SPPHS	Spindle phase synchronous control signal	G038#3	○	○	9.11
	SPSLA	Spindle select signal (serial spindle)	G071#2	○	○	9.2
	SPSLB		G075#2	○	○	
	SPSTP	Spindle stop complete signal	G028#6	○	–	9.7
	SPSYC	Spindle synchronous control signal	G038#2	○	○	9.11
	SRLNI0 to SRLNI3	Group number specification signals	G091#0 to #3	○	○	13.5
	SRLNO0 to SRLNO3	Group number output signals	F178#0 to #3	○	○	
	SRN	Program restart signal	G006#0	○	○	5.7
	SRNMV	Program restart under way signal	F002#4	○	○	
	SRVA	CCW command signal (serial spindle)	G070#4	○	○	9.2
	SRVB		G074#4	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SSIN	Spindle motor command polarity select signal	G033#6	○	○	15.4
	SSIN2		G035#6	○	○	
	SSIN3		G037#6	○	○	
	SSTA	Speed zero signal (serial spindle)	F045#1	○	○	9.2
	SSTB		F049#1	○	○	
	ST	Cycle start lamp signal	G007#2	○	○	5.1
	STL	Cycle start signal	F000#5	○	○	
	STLK	Start lock signal	G007#1	○	–	2.5
	STRD	Input and run simultaneous mode select signal	G058#5	–	○	13.6
	STWD	Output and run simultaneous mode select signal	G058#6	–	○	
	SUCLP	Spindle unclamp signal	F038#1	○	–	9.7
	SVF1 to SVF4	Servo off signal	G126	○	○	1.2.8
	SVSCK1 to SVSCK4	Servo rotational speed check enable signal	G349#0 to #3	○	○	2.10
	SWS1	Spindle selection signals	G027#0	○	○	9.9
	SWS2		G027#1	○	○	
	SWS3		G027#2	○	○	
	SYCAL	Phase error monitor signal	F044#4	○	○	9.11
	SYNC1 to SYNC4	Simple synchronous axis select signal	G138	○	○	1.7
	SYNCJ1 to SYNCJ4	Simple synchronous manual feed axis select signal	G140	–	○	1.7
T	T00 to T31	Tool function code signal	F026 to F029	○	○	8.1
	TAP	Tapping signal	F001#5	○	○	11.6
	TDFSV1 to TDFSV4	Trouble forecast signals	F298#0 to #3	○	○	18.1
	TF	Tool function strobe signal	F007#3	○	○	8.1
	TFIN	Tool function completion signal	G005#3	○	○	8.4
	THRD	Thread cutting signal	F002#3	○	○	6.4.1
	TL01 to TL64	Tool group number select signal	G047#0 to #6	○	–	10.3
	TL01 to TL256		G047#0 to G048#0	–	○	
	TLCH	Tool change signal	F064#0	○	○	
	TLCHB	Tool life arrival notice signal	F064#3	–	○	10.3.2
	TLCHI	Individual tool change signal	F064#2	–	○	10.3
	TLMA	Torque limit signal (serial spindle)	F045#6	○	○	9.2
	TLMB		F049#6	○	○	
	TLMHA	Torque limit command HIGH signal (serial spindle)	G070#1	○	○	
	TLMHB		G074#1	○	○	
	TLMLA	Torque limit command LOW signal (serial spindle)	G070#0	○	○	
	TLMLB		G074#0	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
T	TLNW	New tool select signal	F064#1	○	○	10.3
	TLRST	Tool change reset signal	G048#7	○	○	
	TLRSTI	Individual tool change reset signal	G048#6	–	○	
	TLSKP	Tool skip signal	G048#5	○	○	
	TMRON	General–purpose integrating meter start signal	G053#0	○	○	12.1.11
	TRQL1 to TRQL4	Torque limit reached signal	F114	○	–	14.3.4
	TSA1 to TSA4	Servo rotational speed too–low alarm signal	F349#0 to #3	○	○	2.10
U	UI000 to UI015	Input signal for custom macro	G054,G055	○	○	11.5.1
	UINT	Interrupt signal for custom macro	G053#3	○	○	11.5.2
	UO000 to UO015	Output signal for custom macro	F054,F055	○	○	11.5.1
	UO100 to UO131		F056 to F059	○	○	
W	WOQSM	Workpiece coordinate system shift value write mode select signal	G039#6	○	–	14.4.2
	WOSET	Workpiece coordinate system shift value write signal	G040#7	○	–	
X	XAE	Measuring position reached signal	X004#0	○	○	14.2
Y	YAE		X004#1	–	○	
Z	ZAE		X004#1	○	–	
	ZAE		X004#2	–	○	
	ZP1 to ZP4	Reference position return end signal	F094	○	○	4.1
	ZP21 to ZP24	2nd reference position return end signal	F096	○	○	4.5
	ZP31 to ZP34	3rd reference position return end signal	F098	○	○	
	ZP41 to ZP44	4th reference position return end signal	F100	○	○	
	ZRF1 to ZRF4	Reference position establishment signal	F120	○	○	4.1
	ZRN	Manual reference position return selection signal	G043#7	○	○	
	ZRNO	Software operator's panel signal(ZRN)	F073#4	○	○	12.1.14

A.2.3 List of Signals (In Order of Addresses)

○	:	Available
–	:	Unavailable

Address	Signal name	Symbol	T series	M series	Reference Item
X004#0	Measuring position reached signal	XAE	○	○	14.2
X004#1		YAE	–	○	
X004#1		ZAE	○	–	
X004#2		ZAE	–	○	
X004#2,#4	Manual feed interlock signal for each axis	+MIT1,+MIT2	○	–	2.5
X004#2,#4	Tool offset write signal	+MIT1,+MIT2	○	–	14.4.2
X004#2 to #6,#0,#1	Skip signal	SKIP2 to SKIP6, SKIP7,SKIP8	○	○	14.3.3
X004#3,#5	Manual feed interlock signal for each axis	-MIT1,-MIT2	○	–	2.5
X004#3,#5	Tool offset write signal	-MIT1,-MIT2	○	–	14.4.2
X004#6	Skip signal (PMC axis control)	ESKIP	○	○	15.1
X004#7	Skip signal	SKIP	○	○	14.3.1
X004#7	Overload torque signal	SKIP	–	○	11.14
X008#4	Emergency stop signal	*ESP	○	○	2.1
X009	Reference position return deceleration signal	*DEC1 to *DEC8	○	○	4.1
G000,G001	Data signal for external data input	ED0 to ED15	○	○	15.2
G002#0 to #6	Address signal for external data input	EA0 to EA6	○	○	
G002#7	Read signal for external data input	ESTB	○	○	
G004#3	End signal	FIN	○	○	8.1
G004#4	2nd M function completion signal	MFIN2	○	○	8.4
G004#5	3rd M function completion signal	MFIN3	○	○	
G005#0	Auxiliary function completion signal	MFIN	○	○	
G005#1	External operation function completion signal	EFIN	–	○	
G005#2	Spindle function completion signal	SFIN	○	○	
G005#3	Tool function completion signal	TFIN	○	○	
G005#4	2nd auxiliary function completion signal	BFIN	○	–	
G005#6	Auxiliary function lock signal	AFL	○	○	8.2
G005#7	2nd auxiliary function completion signal	BFIN	–	○	8.4
G006#0	Program restart signal	SRN	○	○	5.7
G006#2	Manual absolute signal	*ABSM	○	○	5.4
G006#4	Override cancel signal	OVC	○	○	7.1.7.3
G006#6	Skip signal	SKIPP	○	–	14.3.1
G007#1	Start lock signal	STLK	○	–	2.5
G007#2	Cycle start signal	ST	○	○	5.1
G007#4	Stroke check 3 release signal	RLSOT3	○	○	2.3.3
G007#5	Follow–up signal	*FLWU	○	○	1.2.7

Address	Signal name	Symbol	T series	M series	Reference Item
G007#6	Stored stroke limit select signal	EXLM	○	○	2.3.2
G007#7	Stroke limit release signal	RLSOT	–	○	
G008#0	Interlock signal	*IT	○	○	2.5
G008#1	Cutting block start interlock signal	*CSL	○	○	
G008#3	Block start interlock signal	*BSL	○	○	
G008#4	Emergency stop signal	*ESP	○	○	2.1
G008#5	Feed hold signal	*SP	○	○	5.1
G008#6	Reset & rewind signal	RRW	○	○	5.2
G008#7	External reset signal	ERS	○	○	
G009#0 to 4	Workpiece number search signal	PN1,PN2,PN4, PN8,PN16	○	○	15.3
G010,G011	Manual feedrate override signal	*JV0 to *JV15	○	○	3.1
G012	Feedrate override signal	*FV0 to *FV7	○	○	7.1.7.2
G014#0,#1	Rapid traverse override signal	ROV1,ROV2	○	○	7.1.7.1
G016#7	F1–digit feed select signal	F1D	–	○	7.1.5
G018#0 to #3	Manual handle feed axis selection signal	HS1A to HS1D	○	○	3.2
G018#4 to #7		HS2A to HS2D	○	○	
G019#0 to #3		HS3A to HS3D	○	○	
G019#4,#5	Manual handle feed amount selection signal (incremental feed signal)	MP1,MP2	○	○	
G019#7	Manual rapid traverse selection signal	RT	○	○	3.1
G023#5	In–position check disable signal	NOINPS	○	○	7.2.5.3
G024#0 to G025#5	Expanded workpiece number search signals	EPN0 to EPN13	○	○	15.3.2
G025#7	Expanded workpiece number search start signal	EPNS	○	○	
G027#0	Spindle selection signal	SWS1	○	○	9.9
G027#1		SWS2	○	○	
G027#2		SWS3	○	○	
G027#3	Stop signal in each spindle	*SSTP1	○	○	9.9
G027#4		*SSTP2	○	○	
G027#5		*SSTP3	○	○	
G027#7	Cs contour control switch signal	CON	○	○	9.8.1
G028#1,#2	Gear selection signal (input)	GR1,GR2	○	○	9.3
G028#4	Spindle unclamp completion signal	*SUCPF	○	–	9.7
G028#5	Spindle clamp completion signal	*SCPF	○	–	
G028#6	Spindle stop complete signal	SPSTP	○	–	
G028#7	2nd position coder selection signal	PC2SLC	○	–	9.9
G029#0	Gear select signal (input)	GR21	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
G029#4	Spindle speed arrival signal	SAR	○	○	9.3
G029#5	Spindle orientation signal	SOR	○	○	
G029#6	Spindle stop signal	*SSTP	○	○	
G030	Spindle speed override signal	SOV0 to SOV7	○	○	
G032#0 to G033#3	Spindle motor speed command signal	R01I to R12I	○	○	15.4
G033#5	Spindle motor command polarity select signal	SGN	○	○	
G033#6		SSIN	○	○	
G033#7	Spindle motor speed command select signal	SIND	○	○	
G034#0 to G035#3	Spindle motor speed command signal	R01I2 to R12I2	○	○	
G035#5	Spindle motor command polarity select signal	SGN2	○	○	
G035#6	Spindle motor command polarity select signal	SSIN2	○	○	
G035#7	Spindle motor speed command select signal	SIND2	○	○	
G036#0 to G037#3	Spindle motor speed command signal	R01I3 to R12I3	○	○	
G037#5	Spindle motor command polarity select signal	SGN3	○	○	
G037#6	Spindle motor command polarity select signal	SSIN3	○	○	
G037#7	Spindle motor speed command select signal	SIND3	○	○	
G038#2	Spindle synchronous control signal	SPSYC	○	○	9.11
G038#3	Spindle phase synchronous control signal	SPPHS	○	○	
G038#6	B–axis unclamp completion signal	*BEUCP	–	○	11.10
G038#7	B–axis clamp completion signal	*BECLP	–	○	
G039#0 to #5	Tool offset number select signal	OFN0 to OFN5	○	–	14.4.2
G039#6	Workpiece coordinate system shift value write mode select signal	WOQSM	○	–	
G039#7	Tool offset value write mode select signal	GOQSM	○	–	
G040#5	Spindle measurement select signal	S2TLS	○	–	14.4
G040#6	Position record signal	PRC	○	–	14.4.1
G040#7	Workpiece coordinate system shift value write signal	WOSET	○	–	14.4.2
G041#0 to #3	Manual handle interrupt axis selection signal	HS1IA to HS1ID	○	○	3.3
G041#4 to #7		HS2IA to HS2ID	○	○	
G042#0 to #3		HS3IA to HS3ID	○	○	
G042#7	Direct operation select signal	DMMC	○	○	15.6
G043#0 to #2	Mode selection signal	MD1,MD2,MD4	○	○	2.6
G043#5	DNC operation select signal	DNCI	○	○	5.9
G043#7	Manual reference position return selection signal	ZRN	○	○	4.1
G044#0,G045	Optional block skip signal	BDT1, BDT2 to BDT9	○	○	5.5
G044#1	All–axis machine lock signal	MLK	○	○	5.3.1
G046#1	Single block signal	SBK	○	○	5.3.3
G046#3 to #6	Memory protect signal	KEY1 to KEY4	○	○	12.2.3
G046#7	Dry run signal	DRN	○	○	5.3.2

Address	Signal name	Symbol	T series	M series	Reference Item
G047#0 to #6	Tool group number select signal	TL01 to TL64	○	–	10.3
G047#0 to G048#0		TL01 to TL256	–	○	
G048#5	Tool skip signal	TLSKP	○	○	
G048#6	Individual tool change reset signal	TLRSTI	–	○	
G048#7	Tool change reset signal	TLRST	○	○	
G049#0 to G050#1	Tool life count override signal	*TLV0 to *TLV9	–	○	
G053#0	General–purpose integrating meter start signal	TMRON	○	○	12.1.11
G053#3	Interrupt signal for custom macro	UINT	○	○	11.5.2
G053#6	Error detect signal	SMZ	○	–	7.2.5.4
G053#7	Chamferring signal	CDZ	○	–	11.8
G054,G055	Input signal for custom macro	UI000 to UI015	○	○	11.5.1
G058#0	External start signal program input	MINP	○	○	13.4
G058#1	External read start signal	EXRD	○	○	13.5, 13.3
G058#2	External read/punch stop signal	EXSTP	○	○	
G058#3	External punch start signal	EXWT	○	○	
G060#7	Tail stock barrier select signal	*TSB	○	–	2.3.4
G061#0	Rigid tapping signal	RGTAP	○	○	9.10
G061#4,#5	Rigid tap spindle select signal	RGTSP1, RGTSP2	○	–	
G062#1	Automatic erase CRT screen display cancel signal	*CRTOF	○	○	12.1.17
G062#6	Rigid tapping retraction start signal	RTNT	–	○	5.11
G063#5	Perpendicular/angular axis control disable signal	NOZAGC	○	○	1.8
G066#0	All–axis VRDY OFF alarm ignore signal	IGNVRY	○	○	2.8
G066#1	External key input mode selection signal	ENBKY	○	○	15.5
G066#4	Retract signal	RTRCT	○	○	1.9
G066#7	Key code read signal	EKSET	○	○	15.5
G067#6	Hard copy stop request signal	HCABT	○	○	13.6
G067#7	Hard copy request signal	HCREQ	○	○	13.6
G070#0	Torque limit command LOW signal (serial spindle)	TLMLA	○	○	9.2
G070#1	Torque limit command HIGH signal (serial spindle)	TLMHA	○	○	
G070#3,#2	Clutch/gear signal (serial spindle)	CTH1A,CTH2A	○	○	
G070#4	CCW command signal (serial spindle)	SRVA	○	○	
G070#5	CW command signal (serial spindle)	SFRA	○	○	
G070#6	Orientation command signal (serial spindle)	ORCMA	○	○	
G070#7	Machine ready completion signal (serial spindle)	MRDYA	○	○	
G071#0	Alarm reset signal (serial spindle)	ARSTA	○	○	
G071#1	Emergency stop signal (serial spindle)	*ESPA	○	○	
G071#2	Spindle select signal (serial spindle)	SPSLA	○	○	
G071#3	Power line switch completion signal (serial spindle)	MCFNA	○	○	
G071#4	Soft start stop cancel signal (serial spindle)	SOCNA	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
G071#5	Signal for controlling velocity integration (serial spindle)	INTGA	○	○	9.2, 9.15
G071#6	Output switch request signal (serial spindle)	RSLA	○	○	9.2
G071#7	Power line status check signal (serial spindle)	RCHA	○	○	
G072#0	Orientation stop position change signal (serial spindle)	INDXA	○	○	9.2, 9.15
G072#1	Rotation direction command while changing the orientation stop position signal (serial spindle)	ROTAA	○	○	
G072#2	Short–distant movement command while changing the orientation stop position signal (serial spindle)	NRROA	○	○	
G072#3	Differential mode command signal (serial spindle)	DEFMDA	○	○	
G072#4	Analog override command signal (serial spindle)	OVRA	○	○	
G072#5	Incremental command external setting type orientation signal (serial spindle)	INCMDA	○	○	
G072#6	Main spindle MCC status signal while changing spindles signal (serial spindle)	MFNHGA	○	○	
G072#7	High–output MCC status signal while a magnetic sensor (serial spindle)	RCHHGA	○	○	
G073#0	Command for spindle orientaion with a magnetic sensor (serial spindle)	MORCMA	○	○	
G073#1	Slave operation command signal (serial spindle)	SLVA	○	○	
G073#2	Motor power stop signal (serial spindle)	MPOFA	○	○	
G073#4	Disconnection detection disbale signal (serial spindle)	DSCNA	○	○	
G074#0	Torque limit command LOW signal (serial spindle)	TLMLB	○	○	9.2
G074#1	Torque limit command HIGH signal (serial spindle)	TLMHB	○	○	
G074#3,#2	Clutch/gear signal (serial spindle)	CTH1B,CTH2B	○	○	
G074#4	CCW command signal (serial spindle)	SRVB	○	○	
G074#5	CW command signal (serial spindle)	SFRB	○	○	
G074#6	Orientation command signal (serial spindle)	ORCMB	○	○	
G074#7	Machine ready completion signal (serial spindle)	MRDYB	○	○	
G075#0	Alarm reset signal (serial spindle)	ARSTB	○	○	
G075#1	Emergency stop signal (serial spindle)	*ESPB	○	○	
G075#2	Spindle select signal (serial spindle)	SPSLB	○	○	
G075#3	Power line switch completion signal (serial spindle)	MCFNB	○	○	
G075#4	Soft start stop cancel signal (serial spindle)	SOCNB	○	○	
G075#5	Signal for controlling velocity integration (serial spindle)	INTGB	○	○	9.2, 9.15
G075#6	Output switch request signal (serial spindle)	RSLB	○	○	9.2
G075#7	Power line status check signal (serial spindle)	RCHB	○	○	
G076#0	Orientation stop position change signal (serial spindle)	INDXB	○	○	9.2, 9.15
G076#1	Rotation direction command while changing the orientation stop position signal (serial spindle)	ROTAB	○	○	
G076#2	Short–distant movement command while changing the orientation stop position signal (serial spindle)	NRROB	○	○	
G076#3	Differential mode command signal (serial spindle)	DEFMDB	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
G076#4	Analog override command signal (serial spindle)	OVRB	○	○	9.2, 9.15
G076#5	Incremental command external setting type orientation signal (serial spindle)	INCMDB	○	○	
G076#6	Main spindle MCC status signal while changing spindles signal (serial spindle)	MFNHGB	○	○	
G076#7	High–output MCC status signal while a magnetic sensor (serial spindle)	RCHHGB	○	○	
G077#0	Command for spindle orientaion with a magnetic sensor (serial spindle)	MORCMB	○	○	
G077#1	Slave operation command signal (serial spindle)	SLVB	○	○	
G077#2	Motor power stop signal (serial spindle)	MPOFB	○	○	
G077#4	Disconnection detection disbale signal (serial spindle)	DSCNB	○	○	
G078#0 to G079#3	Spindle orientation external stop position command signal	SHA00 to SHA11	○	○	9.12
G080#0 to G081#3		SHB00 to SHB11	○	○	
G091#0 to #3	Group number specification signals	SRLNI0 to SRLNI3	○	○	13.5
G092#0	I/O Link confirmation signal	IOLACK	○	○	
G092#1	I/O Link specification signal	IOLS	○	○	
G092#2	Power Mate read/write inprogress signal	BGION	○	○	
G092#3	Power Mate read/write alarm signal	BGIALM	○	○	
G092#4	Power Mate background busy signal	BGEN	○	○	
G096#0 to #6	1% step rapid traverse override signal	*HROV0 to *HROV6	○	○	7.1.7.1
G096#7	1% step rapid traverse override select signal	HROV	○	○	
G098	Key code signal	EKC0 to EKC7	○	○	15.5
G100	Feed axis and direction selection signal	+J1 to +J4	○	○	3.1
G101#0 to #3	External deceleration signal 2	*+ED21 to *+ED24	○	○	7.1.9
G102	Feed axis and direction selection signal	-J1 to -J4	○	○	3.1
G103#0 to #3	External deceleration signal 2	*–ED21 to *–ED24	○	○	7.1.9
G104	Axis direction dependent stored stroke limit switch signal	+EXL1 to +EXL4	○	○	2.3.2
G105		–EXL1 to –EXL4	○	○	
G106	Mirror image signal	MI1 to MI4	○	○	1.2.6
G107#0 to #3	External deceleration signal 3	*+ED31 to *+ED34	○	○	7.1.9
G108	Each–axis machine lock signal	MLK1 to MLK4	○	○	5.3.1
G109#0 to #3	External deceleration signal 3	*–ED31 to *–ED34	○	○	7.1.9
G110	Stroke limit external setting signal	+LM1 to +LM4	–	○	2.3.2
G112		-LM1 to -LM4	–	○	
G114	Overtravel signal	*+L1 to *+L4	○	○	2.3.1
G116		*-L1 to *-L4	○	○	
G118	External deceleration signal	*+ED1 to *+ED4	○	○	7.1.9
G120		*-ED1 to *-ED4	○	○	
G124#0 to #3	Controlled axis detach signals	DTCH1 to DTCH4	○	○	1.2.4
G125	Abnormal load detection ignore signal	IUDD1 to IUDD4	○	○	2.9

Address	Signal name	Symbol	T series	M series	Reference Item
G126	Servo off signal	SVF1 to SVF4	○	○	1.2.8
G127#0 to #3	Cs contour control mode fine acceleration/deceleration function disable signal	CDF1 to CDF4	○	○	9.8.1
G130	Interlock signal for each axis	*IT1 to *IT4	○	○	2.5
G132#0 to #3	Interlock signal for each axis and direction	+MIT1 to +MIT4	–	○	
G134#0 to #3		-MIT1 to -MIT4	–	○	
G136	Control axis select signal (PMC axis control)	EAX1 to EAX4	○	○	15.1
G138	Simple synchronous axis select signal	SYNC1 to SYNC4	○	○	1.7
G140	Simple synchronous manual feed axis select signal	SYNCJ1 to SYNCJ4	–	○	
G142#0	Auxiliary function completion signal (PMC axis control)	EFINA	○	○	15.1
G142#1	Accumulated zero check signal	ELCKZA	○	○	
G142#2	Buffering disable signal (PMC axis control)	EMBUFA	○	○	
G142#3	Block stop signal (PMC axis control)	ESBKA	○	○	
G142#4	Servo off signal (PMC axis control)	ESOFA	○	○	
G142#5	Axis control temporary stop signal (PMC axis control)	ESTPA	○	○	
G142#6	Reset signal (PMC axis control)	ECLRA	○	○	
G142#7	Axis control command read signal (PMC axis control)	EBUFA	○	○	
G143#0 to #6	Axis control command signal (PMC axis control)	EC0A to EC6A	○	○	
G143#7	Block stop disable signal (PMC axis control)	EMSBKA	○	○	
G144,G145	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A	○	○	
G146 to G149	Axis control data signal (PMC axis control)	EID0A to EID31A	○	○	
G150#0,#1	Rapid traverse override signal (PMC axis control)	ROV1E,ROV2E	○	○	
G150#5	Override cancel signal (PMC axis control)	OVCE	○	○	
G150#6	Manual rapid traverse selection signal (PMC axis control)	RTE	○	○	
G150#7	Dry run signal (PMC axis control)	DRNE	○	○	
G151	Feedrate override signal (PMC axis control)	*FV0E to *FV7E	○	○	
G154#0	Auxiliary function completion signal (PMC axis control)	EFINB	○	○	
G154#1	Accumulated zero check signal	ELCKZB	○	○	
G154#2	Buffering disable signal (PMC axis control)	EMBUFB	○	○	
G154#3	Block stop signal (PMC axis control)	ESBKB	○	○	
G154#4	Servo off signal (PMC axis control)	ESOFB	○	○	
G154#5	Axis control temporary stop signal (PMC axis control)	ESTPB	○	○	
G154#6	Reset signal (PMC axis control)	ECLRB	○	○	
G154#7	Axis control command read signal (PMC axis control)	EBUFB	○	○	
G155#0 to #6	Axis control command signal (PMC axis control)	EC0B to EC6B	○	○	
G155#7	Block stop disable signal (PMC axis control)	EMSBKB	○	○	
G156,G157	Axis control feedrate signal (PMC axis control)	EIF0B to EIF15B	○	○	
G158 to G161	Axis control data signal (PMC axis control)	EID0B to EID31B	○	○	
G166#0	Auxiliary function completion signal (PMC axis control)	EFINC	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
G166#1	Accumulated zero check signal	ELCKZC	○	○	15.1
G166#2	Buffering disable signal (PMC axis control)	EMBUFC	○	○	
G166#3	Block stop signal (PMC axis control)	ESBKC	○	○	
G166#4	Servo off signal (PMC axis control)	ESOFC	○	○	
G166#5	Axis control temporary stop signal (PMC axis control)	ESTPC	○	○	
G166#6	Reset signal (PMC axis control)	ECLRC	○	○	
G166#7	Axis control command read signal (PMC axis control)	EBUFC	○	○	
G167#0 to #6	Axis control command signal (PMC axis control)	EC0C to EC6C	○	○	
G167#7	Block stop disable signal (PMC axis control)	EMSBKC	○	○	
G168,G169	Axis control feedrate signal (PMC axis control)	EIF0C to EIF15C	○	○	
G170 to G173	Axis control data signal (PMC axis control)	EID0C to EID31C	○	○	
G178#0	Auxiliary function completion signal (PMC axis control)	EFIND	○	○	
G178#1	Accumulated zero check signal	ELCKZD	○	○	
G178#2	Buffering disable signal (PMC axis control)	EMBUFD	○	○	
G178#3	Block stop signal (PMC axis control)	ESBKD	○	○	
G178#4	Servo off signal (PMC axis control)	ESOFD	○	○	
G178#5	Axis control temporary stop signal (PMC axis control)	ESTPD	○	○	
G178#6	Reset signal (PMC axis control)	ECLRD	○	○	
G178#7	Axis control command read signal (PMC axis control)	EBUFD	○	○	
G179#0 to #6	Axis control command signal (PMC axis control)	EC0D to EC6D	○	○	
G179#7	Block stop disable signal (PMC axis control)	EMSBKD	○	○	
G180,G181	Axis control feedrate signal (PMC axis control)	EIF0D to EIF15D	○	○	
G182 to G185	Axis control data signal (PMC axis control)	EID0D to EID31D	○	○	
G192	Each–axis VRDY OFF alarm ignore signal	IGVRY1 to IGVRY4	○	○	2.8
G198	Position display neglect signal	NPOS1 to NPOS4	○	○	12.1.10
G199#0	Manual handle feed generator selection signals	IOLBH2	○	○	16.1
G199#1	Manual handle feed generator selection signals	IOLBH3	○	○	
G200	Axis control superimposed command signal	EASIP1 to EASIP4	○	○	15.1
G274#4	Cs–axis coordinate establishment request signal	CSFI1	○	○	9.8.2
G349#0 to #3	Servo rotational speed check enable signal	SVSCK1 to SVSCK4	○	○	2.10
G359#0 to #3	Each axis in–position check disable signal	NOINP1 to NOINP4	○	○	7.2.5.3
F000#0	Rewinding signal	RWD	○	○	5.2
F000#4	Feed hold lamp signal	SPL	○	○	5.1
F000#5	Cycle start lamp signal	STL	○	○	5.1
F000#6	Servo ready completion signal	SA	○	○	2.2
F000#7	Automatic operation signal	OP	○	○	5.1
F001#0	Alarm signal	AL	○	○	2.4
F001#1	Resetting signal	RST	○	○	5.2
F001#2	Battery alarm signal	BAL	○	○	2.4

Address	Signal name	Symbol	T series	M series	Reference Item
F001#3	Distribution end signal	DEN	○	○	8.1
F001#4	Spindle enable signal	ENB	○	○	9.3
F001#5	Tapping signal	TAP	○	○	11.6
F001#7	CNC signal	MA	○	○	2.2
F002#0	Inch input signal	INCH	○	○	11.4
F002#1	Rapid traversing signal	RPDO	○	○	2.7
F002#2	Constant surface speed signal	CSS	○	○	9.4
F002#3	Thread cutting signal	THRD	○	○	6.4.1
F002#4	Program restart under way signal	SRNMV	○	○	5.7
F002#6	Cutting feed signal	CUT	○	○	2.7
F002#7	Dry run check signal	MDRN	○	○	5.3.2
F003#0	Incremental feed select check signal	MINC	○	○	2.6
F003#1	Manual handle feed select check signal	MH	○	○	
F003#2	Jog feed select check signal	MJ	○	○	
F003#3	Manual data input select check signal	MMDI	○	○	
F003#4	DNC operation selection confirm signal	MRMT	○	○	5.9
F003#5	Automatic operation select check signal	MMEM	○	○	2.6
F003#6	Memory edit select check signal	MEDT	○	○	
F003#7	TEACH IN select check signal	MTCHIN	○	○	
F004#0,F005	Optional block skip check signal	MBDT1,MBDT2 to MBDT9	○	○	5.5
F004#1	All–axis machine lock check signal	MMLK	○	○	5.3.1
F004#2	Manual absolute check signal	MABSM	○	○	5.4
F004#3	Single block check signal	MSBK	○	○	5.3.3
F004#4	Auxiliary function lock check signal	MAFL	○	○	8.2
F004#5	Manual reference position return selection check signal	MREF	○	○	4.1
F007#0	Miscellaneous function strobe signal	MF	○	○	8.1
F007#1	External operation signal for high–speed interface	EFD	–	○	8.4
F007#2	Spindle–speed function strobe signal	SF	○	○	8.1
F007#3	Tool function strobe signal	TF	○	○	
F007#4	2nd auxiliary function strobe signal	BF	○	–	
F007#7		BF	–	○	
F008#0	External operation signal	EF	–	○	11.7
F008#4	2nd M function strobe signal	MF2	○	○	8.3
F008#5	3rd M function strobe signal	MF3	○	○	
F009#4	Decode M signal	DM30	○	○	8.1
F009#5		DM02	○	○	
F009#6		DM01	○	○	
F009#7		DM00	○	○	
F010 to F013	Miscellaneous function code signal	M00 to M31	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
F014 to F015	2nd M function code signal	M200 to M215	○	○	8.3
F016 to F017	3rd M function code signal	M300 to M315	○	○	
F022 to F025	Spindle speed code signal	S00 to S31	○	○	8.1
F026 to F029	Tool function code signal	T00 to T31	○	○	
F030 to F033	2nd miscellaneous function code signal	B00 to B31	○	○	
F034#0 to #2	Gear selection signal (output)	GR1O,GR2O, GR3O	–	○	9.3
F035#0	Spindle fluctuation detection alarm signal	SPAL	○	○	9.5
F036#0 to F037#3S	12–bit code signal	R01O to R12O	○	○	9.3
F038#0	Spindle clamp signal	SCLP	○	–	9.7
F038#1	Spindle unclamp signal	SUCLP	○	–	
F038#2	Spindle enable signals	ENB2	○	○	9.9
F038#3		ENB3	○	○	
F040,F041	Actual spindle speed signal	AR0 to AR15	○	–	9.6
F044#1	Cs contour control switch completion signal	FSCSL	○	○	9.8.1
F044#2	Spindle synchronous speed control completion signal	FSPSY	○	○	9.11
F044#3	Spindle phase synchronous control completion signal	FSPPH	○	○	
F044#4	Spindle synchronous control alarm signal	SYCAL	○	○	9.11, 9.16
F045#0	Alarm signal (serial spindle)	ALMA	○	○	9.2
F045#1	Speed zero signal (serial spindle)	SSTA	○	○	9.2
F045#2	Speed detection signal (serial spindle)	SDTA	○	○	
F045#3	Speed arrival signal (serial spindle)	SARA	○	○	
F045#4	Load detection signal 1 (serial spindle)	LDT1A	○	○	
F045#5	Load detection signal 2 (serial spindle)	LDT2A	○	○	
F045#6	Torque limit signal (serial spindle)	TLMA	○	○	
F045#7	Orientation completion signal (serial spindle)	ORARA	○	○	
F046#0	Power line switch signal (serial spindle)	CHPA	○	○	
F046#1	Spindle switch completion signal (serial spindle)	CFINA	○	○	
F046#2	Output switch signal (serial spindle)	RCHPA	○	○	
F046#3	Output switch completion signal (serial spindle)	RCFNA	○	○	
F046#4	Slave operation status signal (serial spindle)	SLVSA	○	○	9.2, 9.15
F046#5	Signal for approximate spindle orientation with a position coder (serial spindle)	PORA2A	○	○	
F046#6	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	MORA1A	○	○	
F046#7	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	MORA2A	○	○	
F047#0	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	PC1DTA	○	○	
F047#1	Incremental method orientation signal (serial spindle)	INCSTA	○	○	
F047#4	Motor activation off status signal (serial spindle)	EXOFA	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
F048#4	Cs–axis coordinate establishment status signal	CSPENA	○	○	9.8.2
F049#0	Alarm signal (serial spindle)	ALMB	○	○	9.2
F049#1	Speed zero signal (serial spindle)	SSTB	○	○	
F049#2	Speed detection signal (serial spindle)	SDTB	○	○	
F049#3	Speed arrival signal (serial spindle)	SARB	○	○	
F049#4	Load detection signal 1 (serial spindle)	LDT1B	○	○	
F049#5	Load detection signal 2 (serial spindle)	LDT2B	○	○	
F049#6	Torque limit signal (serial spindle)	TLMB	○	○	
F049#7	Orientation completion signal (serial spindle)	ORARB	○	○	
F050#0	Power line switch signal (serial spindle)	CHPB	○	○	
F050#1	Spindle switch completion signal (serial spindle)	CFINB	○	○	
F050#2	Output switch signal (serial spindle)	RCHPB	○	○	
F050#3	Output switch completion signal (serial spindle)	RCFNB	○	○	
F050#4	Slave operation status signal (serial spindle)	SLVSB	○	○	9.2, 9.15
F050#5	Signal for approximate spindle orientation with a position coder (serial spindle)	PORA2B	○	○	
F050#6	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	MORA1B	○	○	
F050#7	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	MORA2B	○	○	
F051#0	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	PC1DTB	○	○	
F051#1	Incremental method orientation signal (serial spindle)	INCSTB	○	○	
F051#4	Motor activation off status signal (serial spindle)	EXOFB	○	○	
F053#0	Key input disable signal	INHKY	○	○	15.5
F053#1	Program screen display mode signal	PRGDPL	○	○	
F053#2	Read/punch in–progress signal	RPBSY	○	○	13.5, 13.3
F053#3	Read/punch alarm signal	RPALM	○	○	
F053#4	Background busy signal	BGEACT	○	○	
F053#7	Key code read completion signal	EKENB	○	○	15.5
F054,F055	Output signal for custom macro	UO000 to UO015	○	○	11.5.1
F056 to F059		UO100 to UO131	○	○	
F060#0	Read completion signal for external data input	EREND	○	○	15.2
F060#1	Search completion signal for external data input	ESEND	○	○	
F060#2	Search cancel signal for external data input	ESCAN	○	○	
F061#0	B–axis unclamp signal	BUCLP	–	○	11.10
F061#1	B–axis clamp signal	BCLP	–	○	
F061#2	Hard copy stop request acceptance flag	HCAB2	○	○	13.6
F061#3	Hard copy in–progress signal	HCEXE	○	○	13.6
F062#0	AI advanced control mode signal	AICC	–	○	7.1.13

Address	Signal name	Symbol	T series	M series	Reference Item
F062#3	Spindle 1 under measurement signal	S1MES	○	–	14.4
F062#4	Spindle 2 under measurement signal	S2MES	○	–	
F062#7	Target part count reached signal	PRTSF	○	○	12.1.11
F063#7	Polygon synchronization under way signal	PSYN	○	–	6.9.1
F064#0	Tool change signal	TLCH	○	○	10.3
F064#1	New tool select signal	TLNW	○	○	
F064#2	Individual tool change signal	TLCHI	–	○	
F064#3	Tool life arrival notice signal	TLCHB	–	○	10.3.2
F065#0	Spindle rotation direction signal	RGSPP	–	○	9.10
F065#1		RGSPM	–	○	
F065#4	Retract completion signal	RTRCTF	○	○	1.9
F066#0	Lock–ahead control mode signal	G08MD	○	○	7.1.12
F066#1	Rigid tapping retraction completion signal	RTPT	–	○	5.11
F066#5	Small–diameter peck drilling in progress signal	PECK2	–	○	11.14
F070#0 to F071#7	Position switch signal	PSW01 to PSW16	○	○	1.2.9
F072	Software operator's panelgeneral–purpose switch signal	OUT0 to OUT7	○	○	12.1.14
F073#0	Software operator's panel signal (MD1)	MD1O	○	○	
F073#1	Software operator's panel signal (MD2)	MD2O	○	○	
F073#2	Software operator's panel signal (MD4)	MD4O	○	○	
F073#4	Software operator's panel signal (ZRN)	ZRNO	○	○	
F075#2	Software operator's panel signal (BDT)	BDTO	○	○	
F075#3	Software operator's panel signal (SBK)	SBKO	○	○	
F075#4	Software operator's panel signal (MLK)	MLKO	○	○	
F075#5	Software operator's panel signal (DRN)	DRNO	○	○	
F075#6	Software operator's panel signal (KEY1 to KEY4)	KEYO	○	○	
F075#7	Software operator's panel signal (*SP)	SPO	○	○	
F076#0	Software operator's panel signal (MP1)	MP1O	○	○	
F076#1	Software operator's panel signal (MP2)	MP2O	○	○	
F076#3	Rigid tapping mode signal	RTAP	○	○	9.10

Address	Signal name	Symbol	T series	M series	Reference Item
F076#4	Software operator's panel signal (ROV1)	ROV1O	○	○	12.1.14
F076#5	Software operator's panel signal (ROV2)	ROV2O	○	○	
F077#0	Software operator's panel signal (HS1A)	HS1AO	○	○	
F077#1	Software operator's panel signal (HS1B)	HS1BO	○	○	
F077#2	Software operator's panel signal (HS1C)	HS1CO	○	○	
F077#3	Software operator's panel signal (HS1D)	HS1DO	○	○	
F077#6	Software operator's panel signal (RT)	RTO	○	○	
F078	Software operator's panel signal (*FV0 to *FV7)	*FV0O to *FV7O	○	○	
F079,F080	Software operator's panel signal (*JV0 to *JV15)	*JV0O to *JV15O	○	○	
F081#0,#2,#4,#6	Software operator's panel signal (+J1 to +J4)	+J1O to +J4O	○	○	
F081#1,#3,#5,#7	Software operator's panel signal (-J1 to -J4)	-J1O to -J4O	○	○	
F090#0	Servo axis abnormal load detected signal	ABTQSV	○	○	2.9
F090#1	First–spindle abnormal load detected signal	ABTSP1	○	○	
F090#2	Second–spindle abnormal load detected signal	ABTSP2	○	○	
F094	Reference position return end signal	ZP1 to ZP4	○	○	4.1
F096	2nd reference position return end signal	ZP21 to ZP24	○	○	4.5
F098	3rd reference position return end signal	ZP31 to ZP34	○	○	
F100	4th reference position return end signal	ZP41 to ZP44	○	○	
F102	Axis moving signal	MV1 to MV4	○	○	1.2.5
F104	In–position signal	INP1 to INP4	○	○	7.2.5.1
F106	Axis moving direction signal	MVD1 to MVD4	○	○	1.2.5
F108	Mirror image check signal	MMI1 to MMI4	○	○	1.2.6
F110#0 to #3	Controlled axis detach status signals	MDTCH1 to MDTCH4	○	○	1.2.4
F112	Distribution completion signal (PMC axis control)	EADEN1 to EADEN4	○	○	15.1
F114	Torque limit reached signal	TRQL1 to TRQL4	○	–	14.3.4
F120	Reference position establishment signal	ZRF1 to ZRF4	○	○	4.1
F122#0	High–speed skip status signal	HDO0	○	○	14.3.2
F124	Stroke limit reached signals	+OT1 to +OT4	–	○	2.3.2
F124#0 to #3	Overtravel alarm in–progress signal	OTP1 to OTP4	○	○	2.3.1
F126	Stroke limit reached signals	–OT1 to –OT4	–	○	2.3.2

Address	Signal name	Symbol	T series	M series	Reference Item
F129#5	Override 0% signal (PMC axis control)	EOV0	○	○	15.1
F129#7	Control axis selection status signal (PMC axis control)	*EAXSL	○	○	
F130#0	In–position signal (PMC axis control)	EINPA	○	○	
F130#1	Following zero checking signal (PMC axis control)	ECKZA	○	○	
F130#2	Alarm signal (PMC axis control)	EIALA	○	○	
F130#3	Auxiliary function executing signal (PMC axis control)	EDENA	○	○	
F130#4	Axis moving signal (PMC axis control)	EGENA	○	○	
F130#5	Positive–direction overtravel signal (PMC axis control)	EOTPA	○	○	
F130#6	Negative–direction overtravel signal (PMC axis control)	EOTNA	○	○	
F130#7	Axis control command read completion signal (PMC axis control)	EBSYA	○	○	
F131#0	Auxiliary function strobe signal (PMC axis control)	EMFA	○	○	
F131#1	Bufferful signal (PMC axis control)	EABUFA	○	○	
F132,F142	Auxiliary function code signal (PMC axis control)	EM11A to EM48A	○	○	
F133#0	In–position signal (PMC axis control)	EINPB	○	○	
F133#1	Following zero checking signal (PMC axis control)	ECKZB	○	○	
F133#2	Alarm signal (PMC axis control)	EIALB	○	○	
F133#3	Auxiliary function executing signal (PMC axis control)	EDENB	○	○	
F133#4	Axis moving signal (PMC axis control)	EGENB	○	○	

Address	Signal name	Symbol	T series	M series	Referen ce Item
F133#5	Positive–direction overtravel signal (PMC axis control)	EOTPB	○	○	15.1
F133#6	Negative–direction overtravel signal (PMC axis control)	EOTNB	○	○	
F133#7	Axis control command read completion signal (PMC axis control)	EBSYB	○	○	
F134#0	Auxiliary function strobe signal (PMC axis control)	EMFB	○	○	
F134#1	Bufferful signal (PMC axis control)	EABUFB	○	○	
F135,F145	Auxiliary function code signal (PMC axis control)	EM11B to EM48B	○	○	
F136#0	In–position signal (PMC axis control)	EINPC	○	○	
F136#1	Following zero checking signal (PMC axis control)	ECKZC	○	○	
F136#2	Alarm signal (PMC axis control)	EIALC	○	○	
F136#3	Auxiliary function executing signal (PMC axis control)	EDENC	○	○	
F136#4	Axis moving signal (PMC axis control)	EGENC	○	○	
F136#5	Positive–direction overtravel signal (PMC axis control)	EOTPC	○	○	
F136#6	Negative–direction overtravel signal (PMC axis control)	EOTNC	○	○	
F136#7	Axis control command read completion signal (PMC axis control)	EBSYC	○	○	
F137#0	Auxiliary function strobe signal (PMC axis control)	EMFC	○	○	
F137#1	Buffer full signal (PMC axis control)	EABUFC	○	○	
F138,F148	Auxiliary function code signal (PMC axis control)	EM11C to EM48C	○	○	
F139#0	In–position signal (PMC axis control)	EINPD	○	○	
F139#1	Following zero checking signal (PMC axis control)	ECKZD	○	○	
F139#2	Alarm signal (PMC axis control)	EIALD	○	○	
F139#3	Auxiliary function executing signal (PMC axis control)	EDEND	○	○	
F139#4	Axis moving signal (PMC axis control)	EGEND	○	○	
F139#5	Positive–direction overtravel signal (PMC axis control)	EOTPD	○	○	
F139#6	Negative–direction overtravel signal (PMC axis control)	EOTND	○	○	
F139#7	Axis control command read completion signal (PMC axis control)	EBSYD	○	○	
F140#0	Auxiliary function strobe signal (PMC axis control)	EMFD	○	○	
F140#1	Buffer full signal (PMC axis control)	EABUFD	○	○	
F141,F151	Auxiliary function code signal (PMC axis control)	EM11D to EM48D	○	○	
F172#6	Absolute position detector battery voltage zero alarm signal	PBATZ	○	○	1.4.2
F172#7	Absolute position detector battery voltage low alarm signal	PBATL	○	○	
F177#0	Slave I/O Link selection signal	IOLNK	○	○	13.5
F177#1	Slave external read start signal	ERDIO	○	○	
F177#2	Slave read/write stop signal	ESTPIO	○	○	
F177#3	Slave external write start signal	EWTIO	○	○	
F177#4	Slave program selection signal	EPRG	○	○	

Address	Signal name	Symbol	T series	M series	Reference Item
F177#5	Slave macro variable selection signal	EVAR	○	○	13.5
F177#6	Slave parameter selection signal	EPARM	○	○	
F177#7	Slave diagnosis selection signal	EDGN	○	○	
F178#0 to #3	Group number output signals	SRLNO0 to SRLNO3	○	○	
F180	Torque limit reach signals for butt–type reference position setting	CLRCH1 to CLRCH4	○	○	4.6
F182	Controlling signal (PMC axis control)	EACNT1 to EACNT4	○	○	15.1
F274#4	Cs–axis coordinate establishment alarm signal	CSFO1	○	○	9.8.2
F298#0 to #3	Trouble forecast signals	TDFSV1 to TDFSV4	○	○	18.1
F349#0 to #3	Servo rotational speed too–low alarm signal	TSA1 to TSA4	○	○	2.10

Index

Note
Volume 1 : Up to Page 692 / Volume 2 : Page 693 and later

≪Numbers≫

≪A≫

≪B≫

≪C≫

≪D≫

≪E≫

Note
Volume 1 : Up to Page 692 / Volume 2 : Page 693 and later

Note
Volume 1 : Up to Page 692 / Volume 2 : Page 693 and later

≪N≫

≪O≫

≪P≫

≪R≫

≪S≫

Note
Volume 1 : Up to Page 692 / Volume 2 : Page 693 and later

≪T≫

≪V≫

≪W≫

Revision Record

FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C CONNECTION MANUAL (FUNCTION) (B–64113EN–1)

Edition	Date	Contents	Edition	Date	Contents
01	Jun., 2004				