GE Fanuc Automation

Computer Numerical Control Products

Series 0 / 00 / 0-Mate

Connection Manual (Function)

GFZ-61393E-2/02 September 1998

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.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

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 the following NC functions required to enable machine tool builders to design their CNC machine tools.

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 :

Product Name	Abbreviations	Series
FANUC Series 0–MC	0–MC	M series
FANUC Series 0–MD II	0–MD II	
FANUC Series 0–MF	0–MF	
FANUC Series 0–GSC	0–GSC	
FANUC Series 0–GSD II	0–GSD II	
FANUC Series 00–MC	00–MC	
FANUC Series 0–Mate MC	0–Mate–MC	
FANUC Series 0–Mate MF	0–Mate MF	
FANUC Series 0–TC	0–TC	T series
FANUC Series 0–TD II	0–TD II	
FANUC Series 0–TF	0–TF	
FANUC Series 0–TTC	0–TTC	
FANUC Series 0–GCC	0–GCC	
FANUC Series 0–GCD II	0–GCD II	
FANUC Series 00–TC	00–TC	
FANUC Series 00–GCC	00–GCC	
FANUC Series 0–Mate TC	0–Mate TC	

Applicable software series and editions

This manual describes signals used with the following software series and editions. It may not be used for other software series and editions.

Product Name	Series	Edition
FANUC Series 0–MC	0466 0469	Edition 20 or later Edition 01 or later
FANUC Series 0–MD II	0473	Edition 02 or later
FANUC Series 0–MF	0467	Edition 06 or later
FANUC Series 0–GSC	0866	Edition 04 or later
FANUC Series 0–GSD II	0892	Edition 02 or later
FANUC Series 00–MC	0468	Edition 01 or later
FANUC Series 0–Mate MC	0455	Edition 03 or later
FANUC Series 0–Mate MF	0454	Edition 02 or later
FANUC Series 0–TC	0666 0669	Edition 18 or later Edition 01 or later
FANUC Series 0–TD II	0673	Edition 02 or later
FANUC Series 0–TF	0667	Edition 08 or later
FANUC Series 0–TTC	0680 0681 0682	Edition 16 or later Edition 16 or later Edition 16 or later
FANUC Series 0–GCC	0861	Edition 07 or later
FANUC Series 0–GCD II	0882	Edition 02 or later
FANUC Series 00–TC	0668	Edition 01 or later
FANUC Series 00–GCC	0862	Edition 01 or later
FANUC Series 0–Mate TC	0655	Edition 03 or later

Expression of signals

One address accommodates eight signals.

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

	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP	SA	STL	SPL	ZP4	ZP3	ZP2	ZP1

In case the signal address differs between the T and M series, which series the signal address belongs to is indicated above its description.

	#7	#6	#5	#4	#3	#2	#1	#0	
Signal address									(T series)
Signal address									(M series)

Explanation of parameters

Parameters are classified by data type as follows :

Dta type	Valid data range
Bit	0 or 1
Byte	0 to 255
Word	0 to ±32767
2–word	0 to ±99999999

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

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

Data No.	#7	#6	#5	#4	#3	#2	#1	#0
0010	APRS			PRG9		OFFVY	EBCL	ISOT

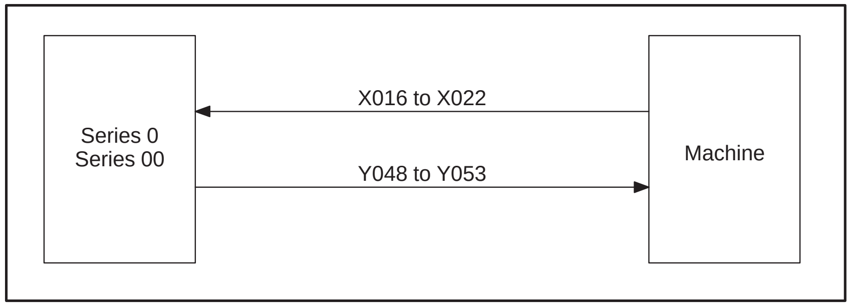
● Notation of parameters other than bit type

Data No.	Data
0518	Data

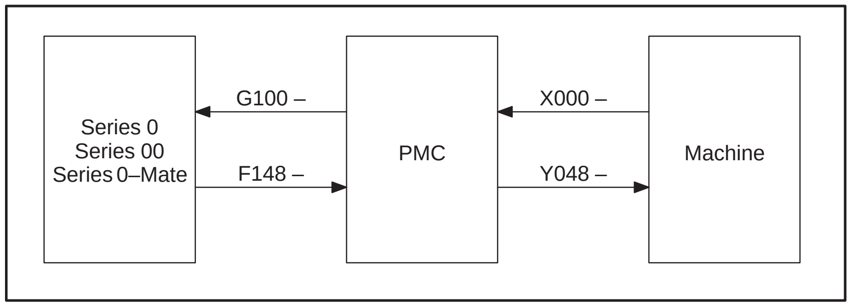
In case the parameter data number differs between the T and M series, which series the data number belongs to is indicated above its description.

Data number	Data	(T series)
Data number	Data	(M series)

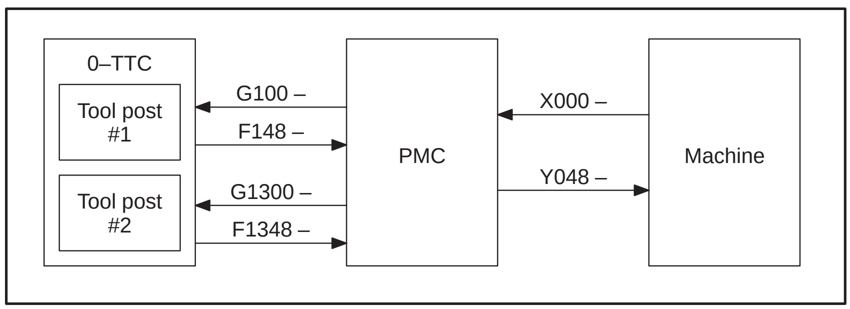
[Machine without PMC]



[Machine with PMC]



[0–TTC]



NOTE

1 When the PMC is not provided, for an explanation of X and Y signals, see the explanation of the G and F signals, respectively, having the addresses obtained by adding 100 to their addresses.
2 What X and Y addresses are usable in a specific unit depends on the type of the I/O card installed in the unit.
3 What F and G addresses are usable in a specific unit depends on the type of the PMC installed in the unit.
4 For the 0–TTC, signal addresses for tool post 2 are obtained by adding 1200 to those for tool post 1. This manual describes only the signals for tool post 1.
5 For the PMC–M, the self–diagnosis data number is obtained by adding 2000 to the signal address.

Manuals related to Series 0/00/0–Mate C and Series 0–D/0–D II

The table below lists manuals related to the Series 0/00/0–Mate C and Series 0–D/0–D II.
In the table, this manual is marked with an asterisk(*).

• Series 0/00/0–Mate C

Table 1 Manuals related to the FANUC Series 0/00/0–Mate C

Manuals name	Specification number	
DESCRIPTIONS	B–61392E	
CONNECTION MANUAL (Hardware)	B–61393E	
CONNECTION MANUAL (Function)	B–61393E–2	*
PROGRAMMING MANUAL (Macro Compiler/Macro Executer)	B–61393E–1	
OPERATOR'S MANUAL (For Lathe)	B–61394E	
OPERATOR'S MANUAL (For MachiningCenter)	B–61404E	
MAINTENANCE MANUAL	B–61395E	
PARAMETER MANUAL (For Lathe)	B–61400E	
PARAMETER MANUAL (For Machining Center)	B–61410E	
DESCRIPTIONS (Remote Buffer)	B–61392EN–1	

• Series 0/0–D II

Table 1 Manuals related to the FANUC Series 0–D/0–D II

Manuals name	Specification number	
DESCRIPTIONS	B–62542EN	
CONNECTION MANUAL (Hardware)	B–61393E	
CONNECTION MANUAL (Function) for Series 0–D	B–62543EN–1	
CONNECTION MANUAL (Function) for Series 0–D II	B–61393E–2	*
OPERATOR'S MANUAL (For Lathe)	B–61394E	
OPERATOR'S MANUAL (For MachiningCenter)	B–61404E	
MAINTENANCE MANUAL	B–61395E	
PARAMETER MANUAL (For Lathe)	B–61400E	
PARAMETER MANUAL (For Machining Center)	B–61410E	

Manuals related to the Servo Motor α series

The table below lists manuals related to the Servo Motor α series.

Table 2 Manuals related to the Servo Motor α series

Document name	Document number	Major contents	Major usage
FANUC AC SERVO MOTOR α series DESCRIPTIONS	B–65142E	• Specification • Characteristics • External dimensions • Connections	• Selection of motor • Connection of motor
FANUC AC SPINDLE MOTOR α series DESCRIPTIONS	B–65152E	• Specification • Characteristics • External dimensions • Connections	
FANUC SERVO AMPLIFIER α series DESCRIPTIONS	B–65162E	• Specifications and functions • Installation • External dimensions and maintenance area • Connections	• Selection of amplifier • Connection of amplifier
FANUC CONTROL MOTOR α series MAINTENANCE MANUAL	B–65165E	• Start up procedure • Troubleshooting • Maintenance of motor	• Start up the system (Hardware) • Troubleshooting • Maintenance of motor
FANUC AC SERVO MOTOR α series PARAMETER MANUAL	B–65150E	• Initial setting • Setting parameters • Description of parameters	• Start up the system (Software) • Turning the system (Parameters)
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL	B–65160E	• Initial setting • Setting parameters • Description of parameters	

Table of Contents

1 AXIS CONTROL

1.1 CONTROLLED AXES

General

The total number of controlled axes and the number of controlled axes that can be used at the same time in a specific unit vary depending on the model and the configuration of its options. Refer to the relevant operator's manual. In manual operation, basically, only one axis can be controlled at a time; the number of axes that can be controlled at the same time can be extended to three by parameter setting, however.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0049				S3JOG				

S3JOG Specifies the number of axes that can be controlled at the same time during manual operation, as follows:
1 : Up to three
0 : One

Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED	The number of the commanded axes exceeded that of simultaneously controlled axes. Correct the program.

Note

NOTE
When the 9" CRT is fitted, the overall position display screen and the position display screen for manual handle interrupt can display up to eight axes. The positions of the 9th and 10th axes are not displayed on these screens when used 0–TTC having nine or more axes.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.2.1	CONTROLLED AXES
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.2.1	CONTROLLED AXES

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

General

The axis names are as follows:

(M series)

Axis number	1	2	3	4
Axis name	X	Y	Z	Parameter (No.0008#2, #3, #4)

(T series)

Axis number	1	2	3	4
Axis name	X	Z	Parameter (No.0030#0)	Parameter (No.0030#4)

(0–TTC)

Axis number	1	2	3	4	5	6
Axis name	X	Z	Parameter (No.0030#0)	Parameter (No.0030#4)	X	Z

(0–GCC/when axis names are changed)

Axis number	1	2	5	6
Axis name	X	Z	Parameter (No.0210)	Parameter (No.0211)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0008				ADW2	ADW1	ADW0			(M series)

ADW0, ADW1, ADW2

Specify the name of the fourth axis as listed below:

ADW2	ADW1	ADW0	Axis name
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W

	#7	#6	#5	#4	#3	#2	#1	#0	
0030				ADW40				ADW30	(T series)

ADW30 Specifies the name of the third axis as follows:
1 : C
0 : B

ADW40 Specifies the name of the fourth axis as follows:
1 : Y
0 : B

	#7	#6	#5	#4	#3	#2	#1	#0	
0069		B3AX	BABS	BAX					(T series)

For G–code system A in the T series, the names of the third axis and fourth axis can be changed by parameter setting as listed below:

B3AX	BABS	BAX	Third axis		Fourth axis	
			ABS	INC	ABS	INC
0	0	0	C	H	Y	V
0	0	1	C	H	Y	B
0	1	1	C	H	B	V
1	0	1	C	B	Y	V
1	1	1	B	H	Y	V

(0–GCC/when axis names are changed)

0210	Axis name of third axis
0211	Axis name of fourth axis

The axis names of the third and fourth axes are set according to the following table:

Axis number	Setting value	Axis name	Setting value
U	85	A	65
V	86	B	66
W	87	C	67

Note

NOTE

1. In 0–TTC, when information (such as the current position) about each axis is displayed on the CRT screen, an axis name may be followed by a subscript to indicate a path number (e.g.,X1 and X2). This is axis name to help the user to easily understand which path an axis belongs to. When writing a program, the user must specify X, Y, Z, U, V, W, A, B, and C without attaching a subscript.
2. If the second auxiliary function is used, it is impossible to use address B as an axis name.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.2.2	NAMES OF AXES
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 into IS–B and IS–C (Tables 1.2.2(a) and 1.2.2 (b)).When selecting IS–C, the option of increment system 1/10 is necessary.

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 X axis at T series. Whether diameter programming or radius programming is used is selected by parameter XRC (No.0019#3).

Parameter

Setting parameter

INCH Specifies the input unit as follows:
0 : Millimeter
1 : Inch

	#7	#6	#5	#4	#3	#2	#1	#0
0001								SCW

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

[Data type] Bit

SCW Specifies the least command increment of linear axes as follows:
0 : Millimeter (metric machine)
1 : Inch (inch machine)

	#7	#6	#5	#4	#3	#2	#1	#0	
0019					XRC				(T series)

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

[Data type] Bit

XRC Specifies whether the X–axis travel distance is specified based on the diameter or radius, as follows:
1 : Radius–based specification
0 : Diameter–based specification

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.2.1	CONTROLLED AXES
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.2.1	CONTROLLED AXES

1.2.3 Specifying the Rotation Axis

General

It is possible to specify whether the fourth axis of the M series unit and the third of fourth axis of the T series unit are a linear or rotation axis, using parameters ADLN (bit 2 of parameter No.0011) and LIN3/LIN4 (bit 2/3 of parameter No.0032), respectively. The basic axes can be specified only as linear, however.
The rollover function can prevent the coordinates of a rotation axis from overflowing. The rollover function is enabled by setting bit 1 of parameter No.0398 (M) and bit 1 of parameter No.0388 (T) to 1 (for the rotation axis).
In a incremental command, the travel distance is determined directly by a value specified in the command. For an absolute command, the coordinates after the tool has moved are values set in parameter No.0860 (M)/No.0788 (T), and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coodinates are closest when parameter No.0398#2 (T) /No.0388#2 (T) is set to 0.
The display of the relative coordinates can be rounded to the travel amount per rotation (when bit 3 of parameter No.0398 (M) or bit 3 of parameter No.0388 (T) is 1).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0011						ADLN			(M series)

ADLN Specifies whether the fourth axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

	#7	#6	#5	#4	#3	#2	#1	#0	
0032					LIN4	LIN3			(T series)

LIN3 Specifies whether the third axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

LIN4 Specifies whether the fourth axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

	#7	#6	#5	#4	#3	#2	#1	#0	
0388					ROCNT	RODRC	ROAXC		(T series)
0398					ROCNT	RODRC	ROAXC		(M series)

[Data type] Bit

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

> **NOTE**
> ROAXC specifies the function only for a rotation axis.

RODRC 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**
> RODRC is valid only when ROAXC is 1.

ROCNT 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 ROCNT is valid only when ROAXC is 1.
> 2 Assign the amount of the shift per one rotation in parameter (No.0860 (M)/No.0788 (T)).

0788	Amount of a shift per one rotation of a rotation axis	(T series)
0860	Amount of a shift per one rotation of a rotation axis	(M series)

> **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 ROAXC = 1.

[Data type] Two–word

[Unit of data]

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.22	ROTARY AXIS ROLL–OVER
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.22	ROTARY AXIS ROLL–OVER

1.2.4 Controlled Axes Detach

General

The PMC can be used to indicate the movement status of each axis.

Signal

Axis–movement–under–way signals MVX to MV4 <F148#4 to #7>

[Classification] Output signal

[Function] These signals indicate that the respective controlled axes are moving. The signals correspond to the controlled axes on a one–to–one basis. A numeral or letter at the end of each signal name represents the controlled axis number.

MV x

X ..... X–axis movement is under way.
Y ..... Y–axis movement is under way.
Z ..... Z–axis movement is under way.
: :
: :

[Output condition] Each signal becomes 1 when:

- The corresponding axis begins to move.
- The corresponding axis is selected as a handle feed axis during manual handle feed mode.

Each signal becomes 0 when:

- Distribution of the move command for the corresponding axis ends.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
F148	MV4	MV3	MVZ	MVX					(T series)
	MV4	MVZ	MVY	MVX					(M series)

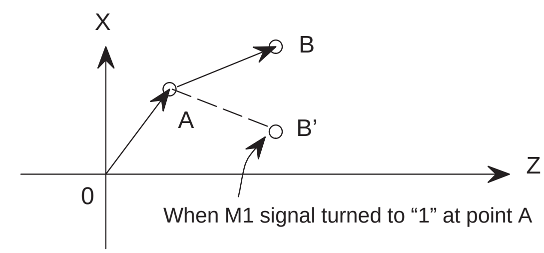
Caution

CAUTION
These signals are output regardless of whether the operation is automatic or manual.

1.2.5
Mirror Image

General

Mirror image can be applied to each axis, either by signals or by setting parameters. 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 #3005 contains the setting data (refer to the operator's manual).

Signal

Mirror image signal MIRX, MIRY, MIR4 <G127#0, #1, #7>(M)

MIX, MIZ <G120#0, G127#1>(T)

[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 or alphabet appended to a signal represents the controlled axis number.

MI x (T series) MIR x (M series)

X ..... Applies mirror image to the X axis.
Y ..... Applies mirror image to the Y axis.
Z ..... Applies mirror image to the Z 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, MMI2, MMI4 <F158#0, #1, #3>(M)

MMI1, MMI2 <F155#0, #1>(T)

[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 n

- 1 ..... Mirror image is applied to the 1st axis
- 2 ..... Mirror image is applied to the 2nd axis
- 4 ..... Mirror image is applied to the 4th 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	
G120								MIX	(T series)
									(M series)
G127							MIZ		(T series)
	MIR4						MIRY	MIRX	(M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
F155							MMI2	MMI1	(T series)
									(M series)
F158									(T series)
					MMI4		MMI2	MMI1	(M series)

Setting parameters (M series)

REVX, REVY, REV4

Turn on or off the mirror image of each axis, as follows:
0 : Off
1 : On

Warning

WARNING
1 When programmable mirror image and ordinary mirror image are specified at the same time, programmable mirror image is applied first.
2 No programmable mirror image affects mirror image check signals MMI1 to MMI4 <F155(T)/F158(M)>.

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 <F155 (T)/F158 (M)>.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.4.8	MIRROR IMAGE
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.4.9	MIRROR IMAGE

1.2.6 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 reset the error counter to zero, assuming a command corresponding to the error has been specified.
Follow–up requires the mechanical handle function and PMC.
Follow–up is always performed during emergency stop or a servo alarm.

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

When signal *FLWU 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 <G104#5>

[Classification] Input signal

[Function] Select whether to perform follow–up when the servo is turned off for those axes.

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

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G104			*FLWU					

Reference item

CONNECTION MANUAL (This manual)	1.2.7	Servo Off (Mechanical handle)

1.2.7 Servo Off (Mechanical Handle)

General

Place the controlled axes in the servo off state; that is, they 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 SVFX to SVF4 <G105#0 to #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 one–to–one basis. A number appended to a signal represents a controlled axis number.

* SVF x

X ..... Servo off for the X axis
Y ..... Servo off for the Y axis
Z ..... Servo off for the Z axis
: :

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

Signal address

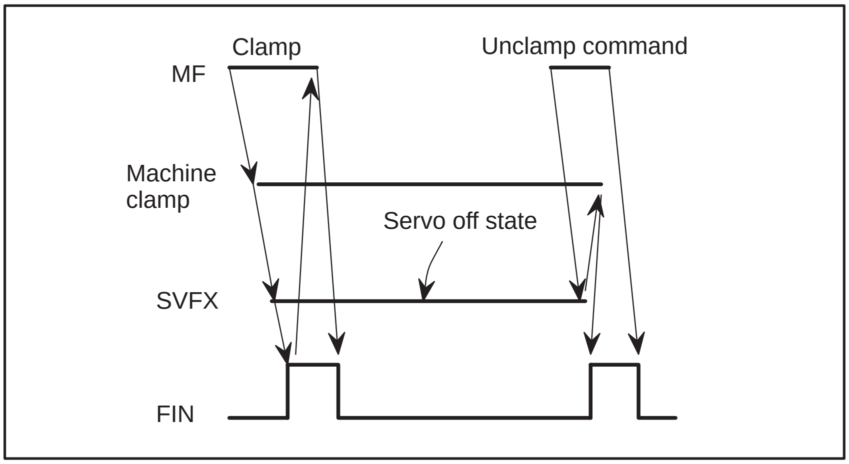
	#7	#6	#5	#4	#3	#2	#1	#0	
G105					SVF4	SVF3	SVFZ	SVFX	(T series)
					SVF4	SVFZ	SVFY	SVFX	(M series)

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)	1.2.6	Follow–up

1.2.8 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 to PSW10 <F190#0 to F191#1>

[Classification] Output signal

[Function] Notifies that the machine coordinates along the controlled axes specified by parameters (0310 to 0319) are within the ranges specified by parameters (0840 to 0849 and 0850 to 0859). Up to ten position switch signals can be output.

[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
F190	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F191							PSW10	PSW09

Parameter

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

0310	Axis corresponding to the first position switch
0311	Axis corresponding to the second position switch
0312	Axis corresponding to the third position switch
0313	Axis corresponding to the fourth position switch
0314	Axis corresponding to the fifth position switch
0315	Axis corresponding to the sixth position switch
0316	Axis corresponding to the seventh position switch
0317	Axis corresponding to the eighth position switch
0318	Axis corresponding to the ninth position switch
0319	Axis corresponding to the tenth position switch

[Data type] Byte

[Valid data range] 1, 2, 3, . . . , Number of control axis

These parameters specify the control–axes numbers corresponding to the first through tenth position switch functions. A corresponding position switch signal is output to PMC when the machine coordinate value of a corresponding axis is within the range that is set using a parameter.

> **NOTE**
> Set 0 for those position switch numbers that are not to be used.

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

● Maximum operation range

0840	Maximum operation range of the first position switch
0841	Maximum operation range of the second position switch
0842	Maximum operation range of the third position switch
0843	Maximum operation range of the fourth position switch
0844	Maximum operation range of the fifth position switch
0845	Maximum operation range of the sixth position switch
0846	Maximum operation range of the seventh position switch
0847	Maximum operation range of the eighth position switch
0848	Maximum operation range of the ninth position switch
0849	Maximum operation range of the tenth position switch

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to +99999999

These parameters set the maximum operation range of the first through tenth position switches.

- **Minimum operation range**

0850	Minimum operation range of the first position switch
0851	Minimum operation range of the second position switch
0852	Minimum operation range of the third position switch
0853	Minimum operation range of the fourth position switch
0854	Minimum operation range of the fifth position switch
0855	Minimum operation range of the sixth position switch
0856	Minimum operation range of the seventh position switch
0857	Minimum operation range of the eighth position switch
0858	Minimum operation range of the ninth position switch
0859	Minimum operation range of the tenth position switch

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] –99999999 to +99999999

These parameters set the minimum operation range of the first through tenth position switches.

1.3 ERROR COMPENSATION

1.3.1 Stored Pitch Error Compensation

General

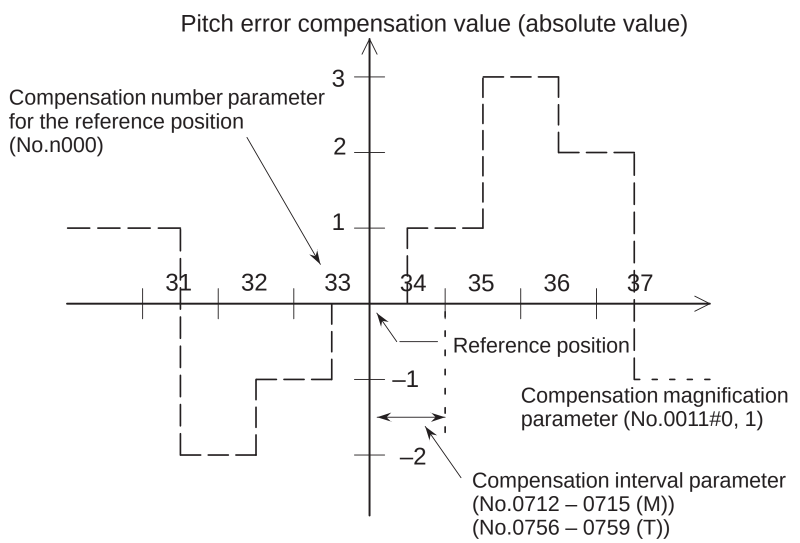
If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.

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

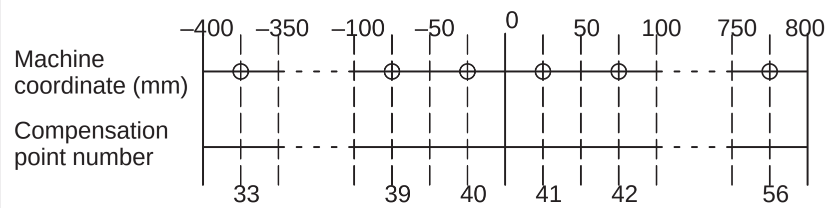
- Number of the pitch error compensation position at the reference position (for each axis): Parameter n000 (n : axis number)
- Pitch error compensation magnification (for each axis): Parameter 0011#0, 1
- Interval of the pitch error compensation positions (for each axis): Parameter 0712 – 0715 (M), 0756 – 0759 (T)

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



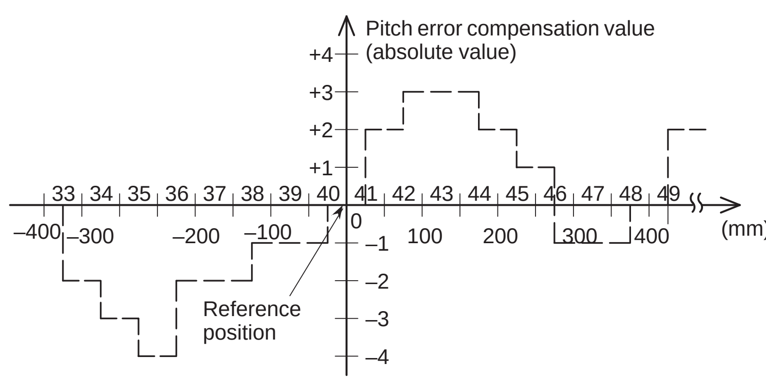
Compensation values are output at the positions indicated by ○ .

Therefore, set the parameters as follows:

Parameter	Setting value
n000 (n : axis number) : Compensation number for the reference position	40
0011#0, 1 : Compensation magnification	0, 0
0712 – 0715 (M), 0756 – 0759 (T) : Interval between pitch error compensation positions	50000

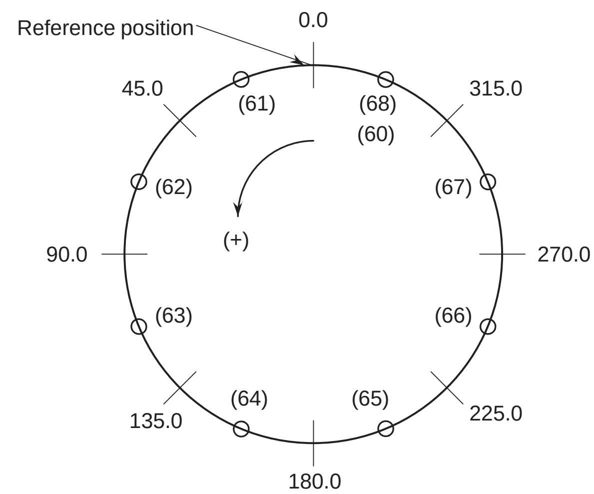
The compensation amount is output at the compensation position No. corresponding to each section between the coordinates.
The following is an example of the compensation amounts.

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



Compensation values are output at the positions indicated by ○ .

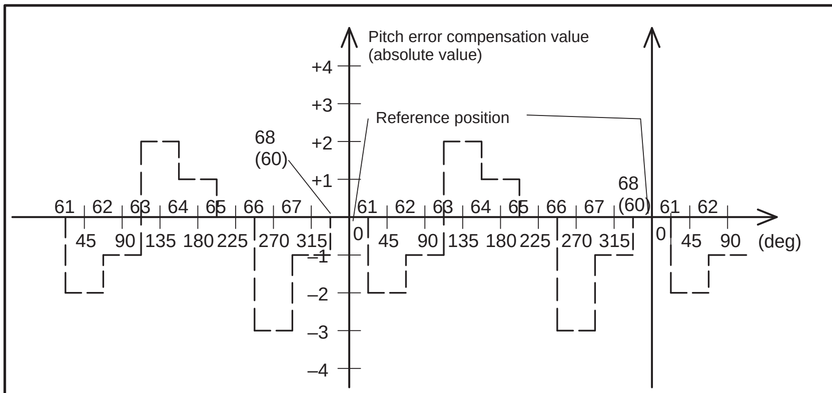
If the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation.
The same value must be set for compensation positions 60 and 68.

Therefore, set the parameters as follows:

Parameter	Setting value
n000 (n : axis number) : Compensation number for the reference position	60
0011#0, 1 : Compensation magnification	0, 0
0712 – 0715 (M), 0756 – 0759 (T) : Interval between pitch error compensation positions	45000

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

n000	Number of the pitch error compensation position for the reference position

(n : axis number)

[Data type] Byte

[Unit of data] Number

[Valid data range] 0 to 127

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

	#7	#6	#5	#4	#3	#2	#1	#0
0011							PML2	PML1

#0 PML1

#1 PML2 Specify the pitch error compensation multiply value as listed below:

#1 PML2	#0 PML1	Multiply
0	0	$\times 1$
0	1	$\times 2$
1	0	$\times 4$
1	1	$\times 8$

0756 – 0759	Interval between pitch error compensation positions	(T series)
0712 – 0715	Interval between pitch error compensation positions	(M series)

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] 0 to 99999999

The pitch error compensation positions are arranged with equally spaced. 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)/1875

Unit: mm, inch, deg

[Example] When the maximum rapid traverse rate is 15000 mm/min, the minimum interval between pitch error compensation positions is 8 mm.

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 1, 2, 4, and 8 in parameters 0011#0, and #1.

2 **Intervals of compensation positions**
The pitch error compensation positions are arranged with equally spaced. Set the space between two adjacent positions for each axis to the parameter (No.n000 (n: axis number)).

3 **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.

4 **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.

1.3.2 Backlash Compensation

General

- **Backlash compensation**

Function for compensating for lost motion on the machine. Set a compensation value in parameter Nos.0535 – 0538, in detection units from 0 to ±2550 pulses.

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

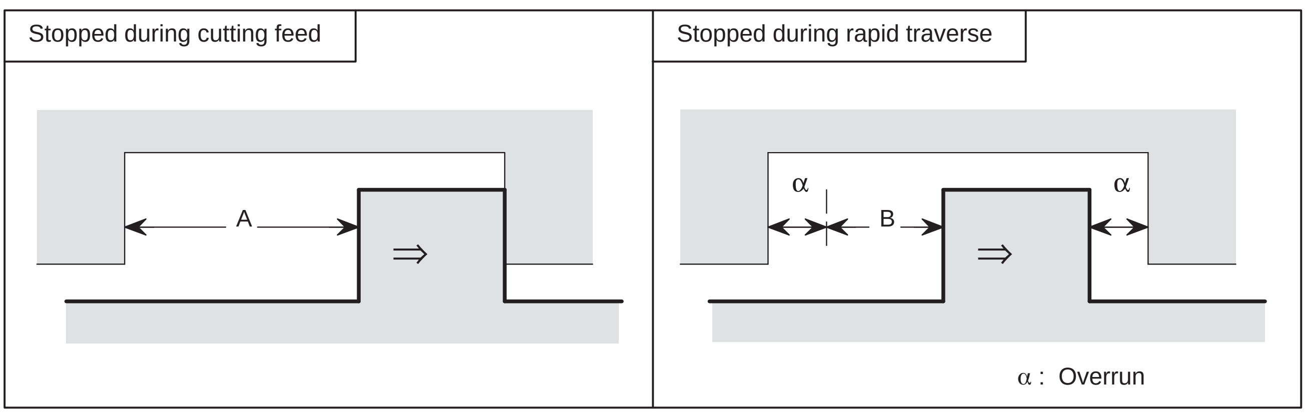
More precise machining can be performed by changing the backlash compensating value depending on the feedrate, the rapid traverse or the cutting feed.

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	$\pm\alpha$	$\pm(-\alpha)$
Opposite direction	$\pm A$	$\pm B$	$\pm(B+\alpha)$	$\pm(B+\alpha)$

· $\alpha = (A✻B)/2$

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



· Assign the measured backlash at cutting feed (A) in parameter Nos.0535 – 0538 and that at rapid traverse (B) in parameter Nos.0686 – 0689 (M)/Nos.0673 – 0676 (T).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0076				ADBLS				

[Data type] Bit

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

0535 – 0538	Backlash compensating value

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to ±2550

Set the backlash compensating value.

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

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

0673 – 0676	Backlash compensating value used for rapid traverse	(T series)
0686 – 0689	Backlash compensating value used for rapid traverse	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to ±2550

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

This parameter is valid when ADBLS, is set to 1.

Caution

CAUTION
The backlash compensation for each 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 Nos.0535 – 0538 irrespective of a rapid traverse and a cutting feed.

Note

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

1.4 SETTINGS RELATED TO SERVO–CONTROLLED AXES

The servo interface of the Series 0 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.8x84 and 8x85 (x : axis number)) n and m (DMR = n/m).

Parameter

POWER OFF	#7	#6	#5	#4	#3	#2	#1	#0	
0004		DMRX			GRDX				
0005		DMRZ			GRDZ				(T series)
		DMRY			GRDY				(M series)
0006		DMR3			GRD3				(T series)
		DMRZ			GRDZ				(M series)
0007		DMR4			GRD4				

GRDx Specify the reference counter size for each axis as listed below:

Setting value				Reference counter size	
#3	**#2**	**#1**	**#0**	**Other than digital servo 0.1μ detector**	**Digital servo 0.1μ detector**
0	0	0	0	1000	10000
0	0	0	1	2000	20000
0	0	1	0	3000	30000
0	0	1	1	4000	40000
0	1	0	0	5000	50000
0	1	0	1	6000	60000
0	1	1	0	7000	70000
0	1	1	1	8000	80000
1	0	0	0	9000	90000
1	0	0	1	10000	100000
1	0	1	0	11000	110000
1	0	1	1	12000	120000
1	1	0	0	13000	130000
1	1	0	1	14000	140000
1	1	1	0	15000	150000
1	1	1	1	16000	160000

DMRx Specify the detection multiply value (DMR) for each axis as follows:

#6	#5	#4	Detection multiplier
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

	#7	#6	#5	#4	#3	#2	#1	#0
0010							OFFVY	

[Data type] Bit

OFFVY 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	
0021			APC8	APC7	APC4	APC3	APCZ	APCX	(T series)
			APC8	APC7	APC4	APCZ	APCY	APCX	(M series)

APCx Specify whether to use absolute pulse coder detector for each axis, as follows:
0 : Do not use
1 : Use

	#7	#6	#5	#4	#3	#2	#1	#0	
0022			ABS8	ABS7	ABS4	ABS3	ABSZ	ABSX	(T series)
			ABS8	ABS7	ABS4	ABSZ	ABSY	ABSX	(M series)

ABSx Specify whether the reference position for the absolute pulse coder of each axis has been established, as follows:
0 : Not established
1 : Established

	#7	#6	#5	#4	#3	#2	#1	#0	
0037			SPTP8	SPTP7	SPTP4	SPTP3	SPTPZ	SPTPX	(T series)
			SPTP8	SPTP7	SPTP4	SPTPZ	SPTPY	SPTPX	(M series)

SPTPx Specify whether to use a separate pulse coder for each axis, as follows:
0 : Do not use
1 : Use

0100 – 0103	Command multiply (CMR)

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

[Data type] Byte

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

Least command increment = detection unit x command multiply

Relationship between the increment system and the least command increment

	Least command increment			
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

The value set in the parameter is obtained as follows:

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

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

Valid data range: 102 to 127

(2) When command multiply is 1 to 48

Set value = 2 $\times$ command multiply
Valid data range: 2 to 96

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

0504 – 0507	Positioning deviation limit in movement

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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.

0508 – 0511	Grid shift

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to $\pm$32767

A grid shift is set.

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the half of 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.

0512 – 0515, 0517	Servo loop gain	(M series)

[Data type] Word

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

[Valid data range] 1 to 9999

Set the loop gain for position control.

When the machine performs linear and circular interpolation (cutting), the same value must be set to parameter 0517. For machines that require only positioning, parameter 0517 must be 0, but the settings of parameters 0512 to 0515 can vary from axis to axis. 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}

0593 – 0596	Positioning deviation limit in the stopped state

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state.

If, in the stopped state, 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).

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 turned on next.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0021			APC8	APC7	APC4	APC3	APCZ	APCX	(T series)
			APC8	APC7	APC4	APCZ	APCY	APCX	(M series)

APCx Specify whether to use absolute pulse coder detector for each axis, as follows:

0 : Do not use

1 : Use

1.4.3 Ignore–the–fourth–axis –signal (M Series)

Signal

Ignore–the–fourth–axis signal 4NG<X004#7> (M series)

[Classification] Input signal

[Function] When a unit with an additional axis is switched on with this signal set to 1, its fourth axis is ignored; that is, it is assumed that the unit has no additional axis. So, it is unnecessary to perform processing related to the additional axis.

CAUTION
This signal is checked instantly the power is switched on. It is no use turning on and off the signal when the power is already on.

NOTE
This signal is valid only when parameter C4NG (bit 1 of parameter No. 0019) = 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004	4NG							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0019							C4NG	

[Data type] Bit

C4NG Specifies whether to enable the ignore–fourth–axis signal (4NG, bit 7 of signal X004), as follows:
0 : Enable
1 : Disable

1.4.4 Cancel–the–Z–axis Command Signal (M Series)

General

Setting this signal to 1 causes the machine to behave in the same manner as when only the Z–axis is locked.

Signal

Cancel–the–Z–axis command signal (Input) ZNG (M series)<G103#6>

[Function] The X– and Y–axis move commands and the M, S, and T functions work normally, but the Z–axis movement of the machine is inhibited, and only the current display is updated.

[Operation] Instantly this signal becomes 1 during Z–axis movement, this function becomes enabled. The signal is valid during both automatic and manual operations.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G103		ZNG						

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.
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.

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.7.1	MACHINE COORDINATE SYSTEM
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.7.1	MACHINE COORDINATE SYSTEM

1.5.2 Workpiece Coordinate System (T/M Series)/ Additional of Workpiece Coordinate System Pair (M Series)

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) Automatic setting
If bit 7 (APRS) of parameter No.0010 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed.

(3) Input using the CRT/MDI panel
Six workpiece coordinate systems can be set beforehand using the CRT/MDI panel.

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) or automatic workpiece coordinate system setting
Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

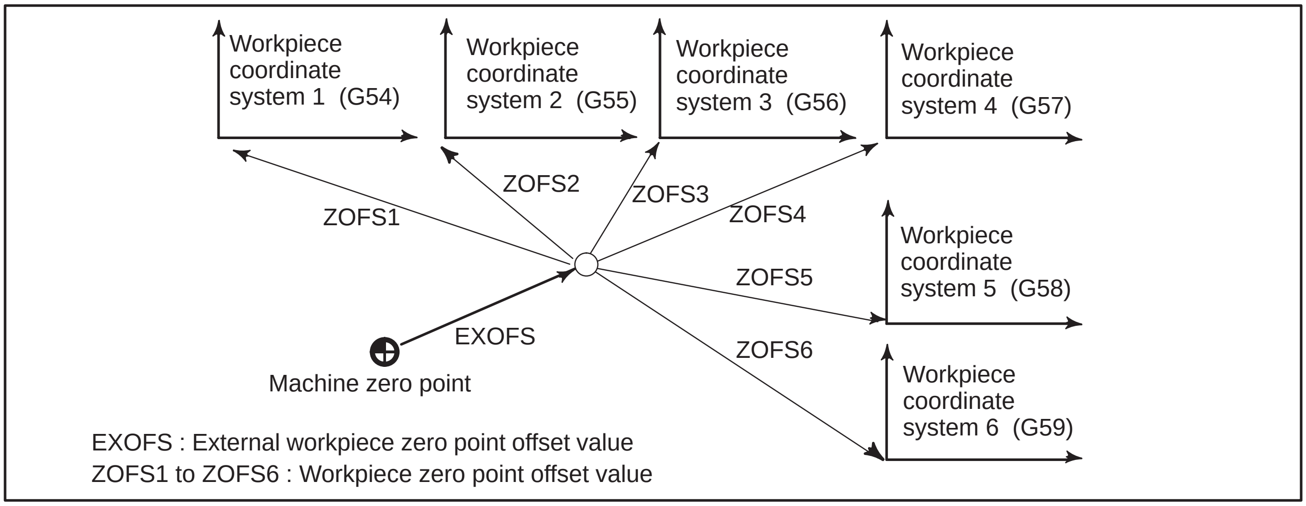
(2) Choosing from six workpiece coordinate systems set using the CRT/MDI panel
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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.

Changing workpiece coordinate system

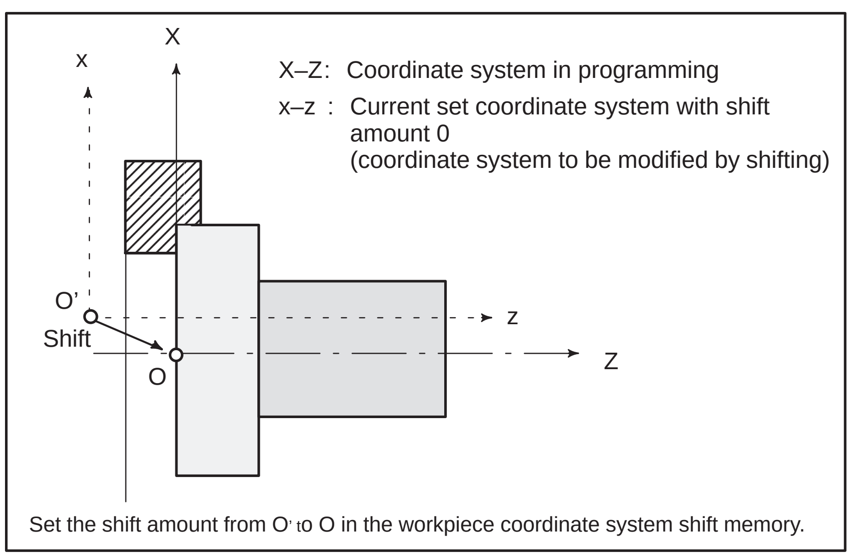
The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.
(1) Inputting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



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 or the automatic coordinate system setting 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	
0010	APRS	WSFT							(T series)
	APRS								(M series)

[Data type] Bit

APRS Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

WSFT Specifies whether to shift the workpiece coordinate system, as follows:
0 : Do not shift
1 : Shift

	#7	#6	#5	#4	#3	#2	#1	#0
0063							PRSTIN	

[Data type] Bit

PRSTIN Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter Nos.0708 – 0711 is used.
1 : For input in mm, the value set in parameter Nos.0708 – 0711 is used, or for input in inches, the value set in parameter Nos.0815 – 0818 is used.

	#7	#6	#5	#4	#3	#2	#1	#0	
0388	WKZSFT								(T series)

[Data type] Bit

WKZSFT 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.

0708 – 0711	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word

[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

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

0751 – 0754	External workpiece zero point offset value	(T series)
0940 – 0943	External workpiece zero point offset value	(M series)

[Data type] Two–word

[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] –7999 to 7999

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. Usually, this is set automatically according to the input from the machine (external data input).

0755 – 0758	Workpiece zero point offset value in workpiece coordinate system1 (G54)	(T series)
0759 – 0762	Workpiece zero point offset value in workpiece coordinate system2 (G55)	(T series)
0763 – 0766	Workpiece zero point offset value in workpiece coordinate system3 (G56)	(T series)
0767 – 0770	Workpiece zero point offset value in workpiece coordinate system4 (G57)	(T series)
0771 – 0774	Workpiece zero point offset value in workpiece coordinate system5 (G58)	(T series)
0775 – 0778	Workpiece zero point offset value in workpiece coordinate system6 (G59)	(T series)

0944 – 0947	Workpiece zero point offset value in workpiece coordinate system1 (G54)	(M series)
0948 – 0951	Workpiece zero point offset value in workpiece coordinate system2 (G55)	(M series)
0952 – 0955	Workpiece zero point offset value in workpiece coordinate system3 (G56)	(M series)
0956 – 0959	Workpiece zero point offset value in workpiece coordinate system4 (G57)	(M series)
0960 – 0963	Workpiece zero point offset value in workpiece coordinate system5 (G58)	(M series)
0964 – 0967	Workpiece zero point offset value in workpiece coordinate system6 (G59)	(M series)

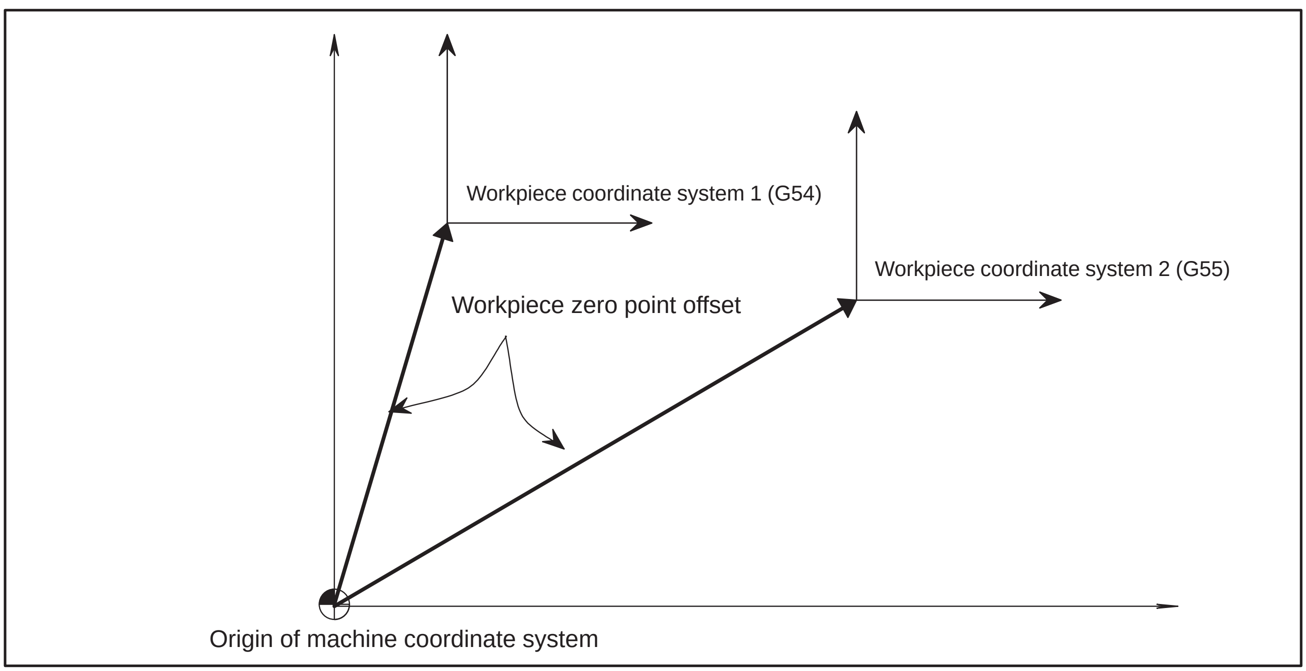
[Data type] Two–word

[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.



0815 – 0818	Coordinate value of the reference position used when automatic coordinate system setting is performed with inch input

[Data type] Two–word

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

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

NOTE
This parameter is valid when PRSTIN in parameter 0063#1 is set to 1.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.7.2	WORKPIECE COORDINATE SYSTEM
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.7.2	WORKPIECE 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 1 (ROACx) of parameter (No.0398 (M) or 0388 (T)) 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.0860 (M) or 0788 (T)). The tool moves in the direction in which the final coordinates are closest when bit 2 (RODRC) of parameter No.0398 (M) or 0388 (T) is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (ROCNT) of parameter No.0398 (M) or 0388 (T) is set to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0011						ADLN			(M series)

ADLN Specifies whether the fourth axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

	#7	#6	#5	#4	#3	#2	#1	#0	
0032					LIN4	LIN3			(T series)

LIN3 Specifies whether the third axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

LIN4 Specifies whether the fourth axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

	#7	#6	#5	#4	#3	#2	#1	#0	
0388					ROCNT	RODRC	ROAXC		(T series)
0398					ROCNT	RODRC	ROAXC		(M series)

[Data type] Bit

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

> **NOTE**
> ROAXC specifies the function only for a rotation axis.

RODRC 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**
> RODRC is valid only when ROAXC is 1.

ROCNT 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 ROCNT is valid only when ROAXC is 1.
> 2 Assign the amount of the shift per one rotation in parameter (No.0860 (M) or 0788 (T)).

0788	Move amount per rotation of rotary axis	(T series)
0860	Move amount per rotation of rotary axis	(M series)

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

[Data type] Two–word

[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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.22	ROTARY AXIS ROLL–OVER
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.22	ROTARY AXIS ROLL–OVER

1.6 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.

- **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.

<M series> The function can synchronize both automatic and manual operations. The fourth axis is used as the slave axis. In the following description, the function is explained separately for the M series and T series.

Signal

<T series and M series>

Signals to select the slave axis for simple synchronous control SYNCX to SYNC4 <G237>

[Classification] Input signal

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

SYNC x

X. ... The X axis becomes the slave axis for synchronization control.
Y. ... The Y axis becomes the slave axis for synchronization control.
Z. ... The Z axis becomes the slave axis for synchronization 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 synchronization control.

The master axis is specified with a parameter.

<M series>

Signal for selecting the manual feed axis for simple synchronous control SYNCJ <G133#6>

[Classification] Input signal

[Function] Synchronization control is performed in jog, handle, or incremental feed mode.

[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 synchronization control.

The master axis is specified with a parameter.

<M series>

Servo axis synchronization alarm signal SYNAL <F192#7>

[Classification] Output signal

[Function] This signal is used to post notification of the occurrence of a synchronization error.

[Operation] :This signal is output when the difference in the positional deviation between the master axis and slave axis exceeds the value set in parameter 475. At the same time, alarm 213 is issued.

Signal address

T series

	#7	#6	#5	#4	#3	#2	#1	#0
G237					SYNC4	SYNC3	SYNCZ	SYNCX

M series

	#7	#6	#5	#4	#3	#2	#1	#0
G133		SYNCJ						
G237					SYNC4			
F192	SYNAL							

Parameter

T series

0281	Axes synchronized with the X and Z axes in synchronization control
0282	Axes synchronized with the 3rd and 4th axes in synchronization control

[Valid data range] 0 to 32

[Unit of data] An axis synchronized with each axis in servo axis synchronization control is set according to the table given below. Set the number of an axis to be synchronized in each digit of the parameters. Set the axis to be synchronized with the X/third axis in the unit's position of each parameter, and set the axis to be synchronized with the Z/fourth axis in the ten's position.

Setting value	Synchronized with
0	X axis
1	Z axis
2	3rd axis
3	4th axis

Ex. No. 0281 = 32

X axis is synchronized with the 3rd axis.
The Z axis is synchronized with the 4th axis.

Ex. No. 0282 = 10

The 3rd axis is synchronized with the X axis.
The 4th axis is synchronized with the Z axis.

M series

	#7	#6	#5	#4	#3	#2	#1	#0
0075							SYNM1	SYNM0

SYNM0, SYNM1 The master axis, in simple synchronous control, is set according to the table below.

#1 SYNM1	#0 SYNM0	Master axis in simple synchronous control
0	0	Simple synchronous control is not applied.
0	1	The master axis is the X axis. (The 4th axis is synchronized with the X axis.)
1	0	The master axis is the Y axis. (The 4th axis is synchronized with the Y axis.)
1	1	The master axis is the Z axis. (The 4th axis is synchronized with the Z axis.)

0475	Limit of the difference between the amount of positioning deviation of the master and slave axes

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Alarm and message

T series

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

M series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	One of the following errors occurred during synchronous operation (simple synchronization control): (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 0475.

Caution

CAUTION

1 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.

2 Pitch error compensation and backlash compensation are executed separately for the master and slave axes.

Reference item

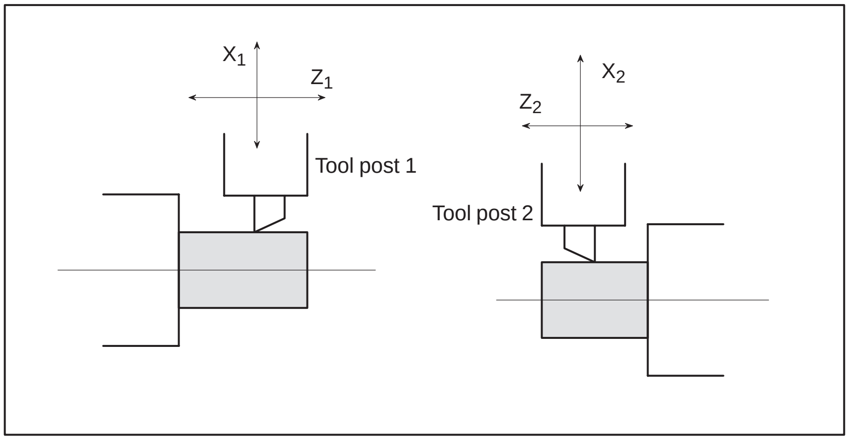
OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.21	SIMPLE SYNCHRONOUS CON-TROL

1.7 AXIS RECOMPOSITION (0–TTC)

General

With the 0–TTC, movement along those axes (X_1, Z_1, C_1, Y_1) of tool post 1 is usually specified using programmed commands for tool post 1, while movement along the axes (X_2, Z_2, C_2) of tool post 2 is specified using programmed commands for tool post 2.

Separate control for each tool post



The axis recomposition function allows synchronization control between the tool posts and within a single post, as well as composite control between the tool posts.

Synchronization control

Movement along an arbitrary axis of a tool post is synchronized with another axis of the other tool post.

Example: Movement along the Z_1 axis is synchronized with movement along the Z_2 axis.

Movement along an arbitrary axis of a tool post is synchronized with another axis of the same tool post.

Example: Movement along the Z_1 axis is synchronized with movement along the Y_1 axis.

Composite control

Move commands for arbitrary axes of the tool posts are switched.

Example: Commands for the X_1 axis and X_2 axis are switched.

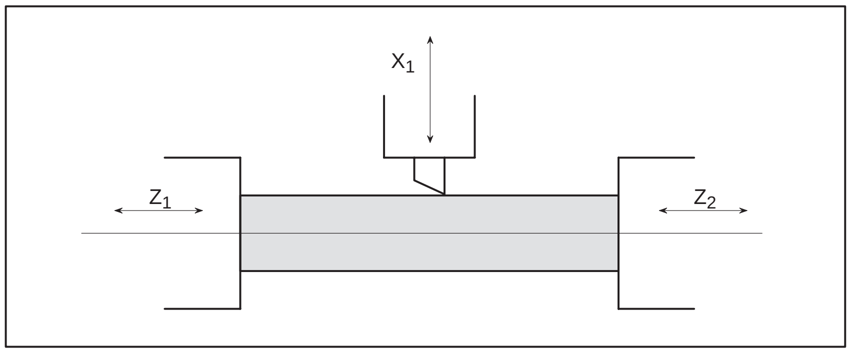
→ Performing movement along the X_2 axis and Z_1 axis by using programmed commands for tool post 1
Performing movement along the X_1 axis and Z_2 axis by using programmed commands for tool post 2

Synchronization control

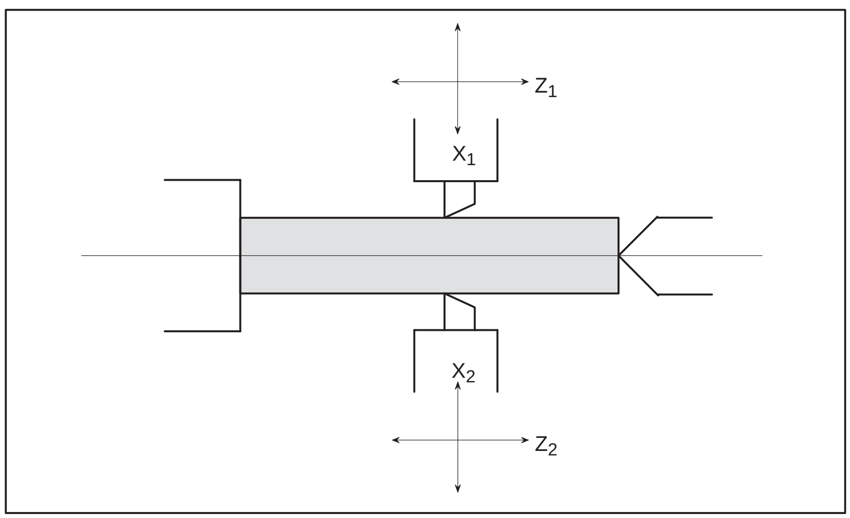
Movement along an axis of one tool post can be synchronized with movement along an axis of the other tool post. When a move command is specified for an axis (synchronization master axis), the same move command is also specified for the corresponding axis (synchronization slave axis) so that the same movement is performed along these axes. When this function is used with the parking function, which places a specific axis in the halt state by ignoring a move command for that axis, the following control operations are enabled:

(1) Synchronizing movement along an arbitrary axis on a tool post with movement along an arbitrary axis on the other tool post (Both master and slave moved)

Example 1: Synchronize the Z_2 axis with the Z_1 axis. (Machining with both ends of a workpiece held)



Example 2: Synchronize the X_2 and Z_2 axes with the X_1 and Z_1 axes. (Balance cut)



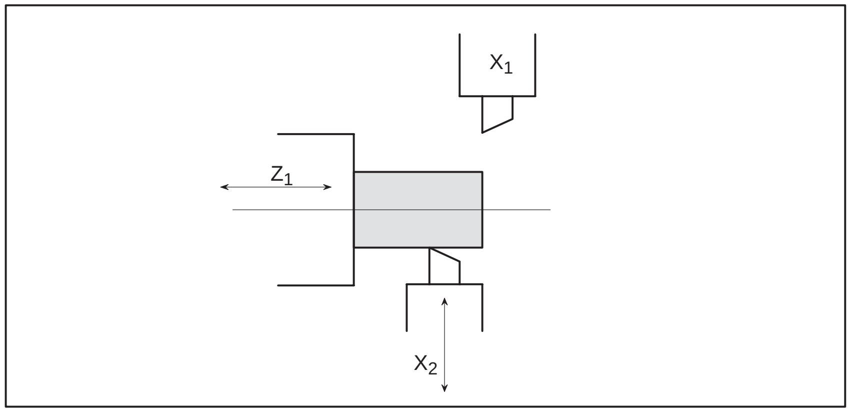
(2) Using a move command for an arbitrary axis on a tool post to perform movement along an arbitrary axis on the other tool post. At this time, no movement is performed along that axis for which the move command is specified. (Master parked, slave moved)

(3) Updating the coordinate on an arbitrary axis of a tool post based on the distance traveled along an arbitrary axis of the other tool post. At this time, no movement is performed along that axis for which the coordinate is updated. (Master moved, slave parked)

Using methods (2) and (3) allows one motor to be controlled by both tool post 1 and tool post 2.

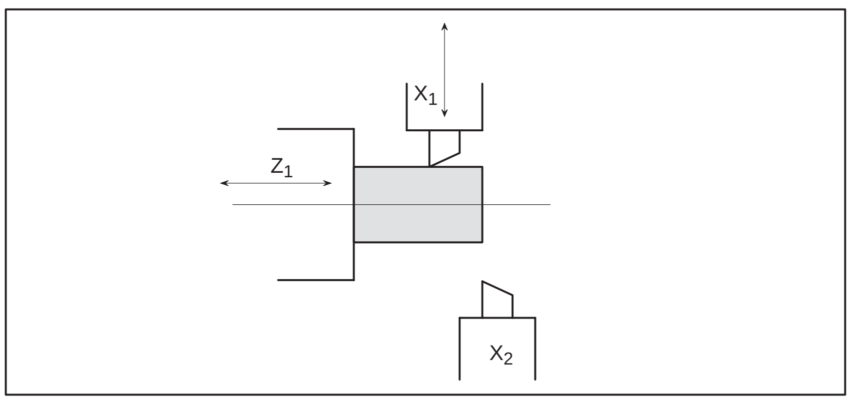
Example 3: Share one motor between the Z_1 axis and Z_2 axis (where, the motor is assumed to be connected to the Z_1 axis).

- **Master parking**



Movement along the X_2 and Z_1 axes is performed using a programmed command for tool post 2. (The Z_1 axis is synchronized with the Z_2 axis). At this time, the Z_2 axis is placed in the parking state. The coordinate systems for the Z_1 and Z_2 axes are updated respectively.

- **Slave parking**



Movement along the X_1 and Z_1 axes is performed using a programmed command for tool post 1. (The Z_2 axis is synchronized with the Z_1 axis.) At this time, the Z_2 axis is placed in the parking state. For the Z_2 axis, only its coordinate system is updated.

The coordinate systems on the Z_1 axis and Z_2 axis are each updated constantly. Therefore, as soon as the synchronization state is changed, a move command can be executed without setting the coordinate system again.

In a special case, the fourth axis of tool post 2 (Y_2 axis) can be used as the synchronization master axis that is always parked, and the seventh axis of tool post 1 can be used as the synchronization slave axis. Then, Y_2 axis control is enabled.

CAUTION

1 The synchronization control mentioned here is used to specify the same move command to the two servo processing systems simultaneously. At this time, the synchronization error compensation function, which always detects the deviation between the two servo motors and compensates one servo motor to reduce the deviation, is not applied. Synchronization error detection can, however, be performed by parameter setting. If a synchronization error is detected, cancel synchronization control immediately, and turn off the servo ready signal.

2 When synchronization control is canceled during automatic operation, do not specify a move command nor coordinate system setting for the synchronization slave axis within the two blocks including the currently executed block (within three blocks when tool–nose radius compensation is being performed).

NOTE

Parking prevents move commands from being issued to the servo processing system, thus preventing the updating of all coordinates. However, absolute and relative coordinates can be updated by parameter setting.

Composite control

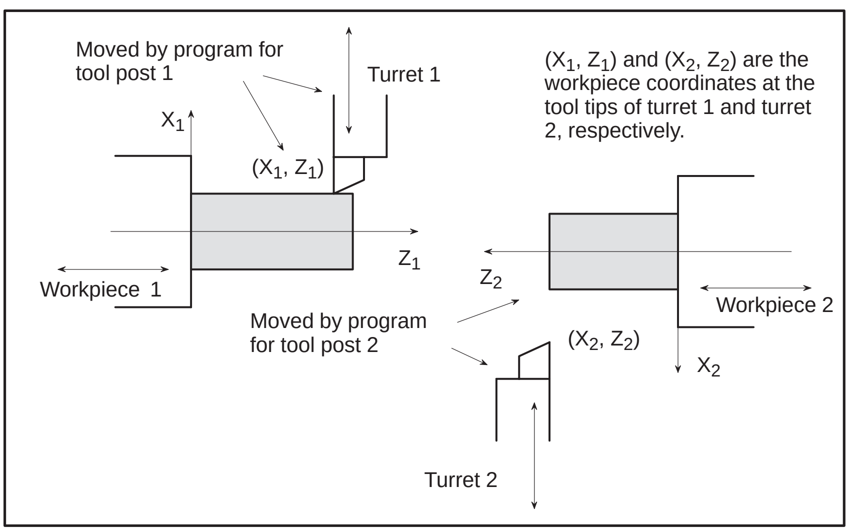
Move commands can be switched between arbitrary axes on the tool posts to perform movement along each axis.

Example of composite control

In a machine having the X_1 and Z_1 axes of tool post 1 and the X_2 and Z_2 axes of tool post 2, switching between the X_1 axis and X_2 axis is explained below. Here, assume that the move commands for the Z_1 and Z_2 axes allow a workpiece to move along these axes.

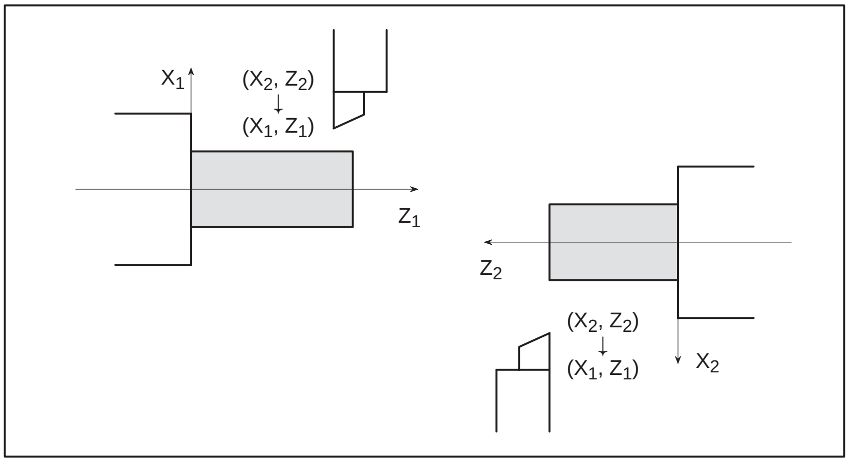
Coordinate systems in separate control mode

Turret 1 and workpiece 1 belong to tool post 1 and are moved according to programmed commands issued for tool post 1. Turret 2 and workpiece 2 belong to tool post 2, and are moved according to programmed commands issued for tool post 2.



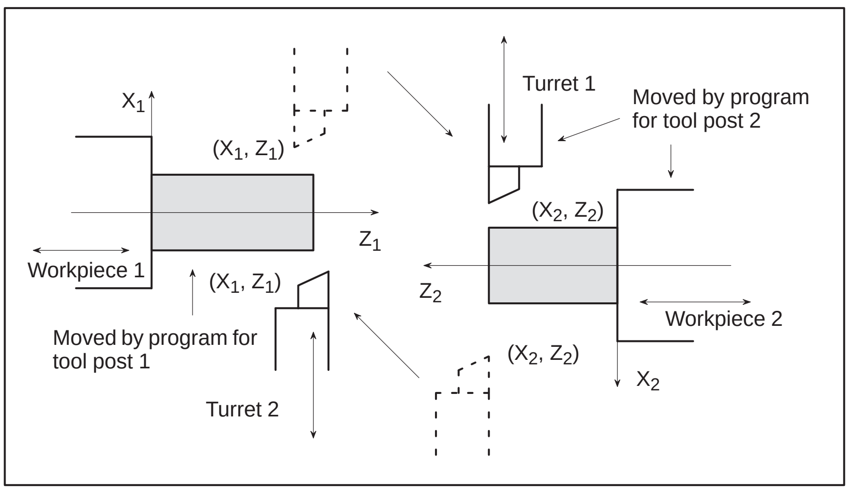
Change from separate control to composite control

The control mode is changed to composite control so that turret 2 is controlled with tool post 1, and turret 1 is controlled with tool post 2. Set the coordinates of the tool tip of turret 2 in the workpiece coordinate system for tool post 1, and set the coordinates of the tool tip of turret 1 in the workpiece coordinate system for tool post 2. The coordinate systems for composite control can be set automatically by parameter setting. In automatic setting, each workpiece coordinate system is calculated from the machine coordinates and the workpiece coordinates at the reference position. (Whether the coordinate systems are to be set automatically is determined by parameter setting.)



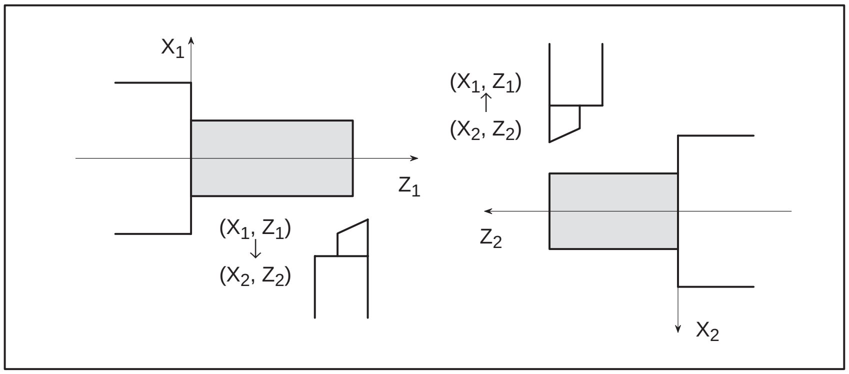
Coordinate systems in composite control mode

Turret 2 and workpiece 1 belong to tool post 1, and are moved by programmed commands issued for tool post 1. Turret 1 and workpiece 2 belong to tool post 2, and are moved by programmed commands issued for tool post 2.



Change from composite control to separate control

The control mode is changed to separate control so that turret 1 is controlled with tool post 1, and turret 2 is controlled with tool post 2. Set the coordinates of the tool tip of turret 1 in the workpiece coordinate system for tool post 1, and set the coordinates of the tool tip of turret 2 in the workpiece coordinate system for tool post 2. The coordinate systems for separate control can be set automatically by parameter setting. In automatic setting, each workpiece coordinate system is calculated from the machine coordinates and the workpiece coordinates at the reference position. (Whether the coordinate systems are to be set automatically is determined by parameter setting.)

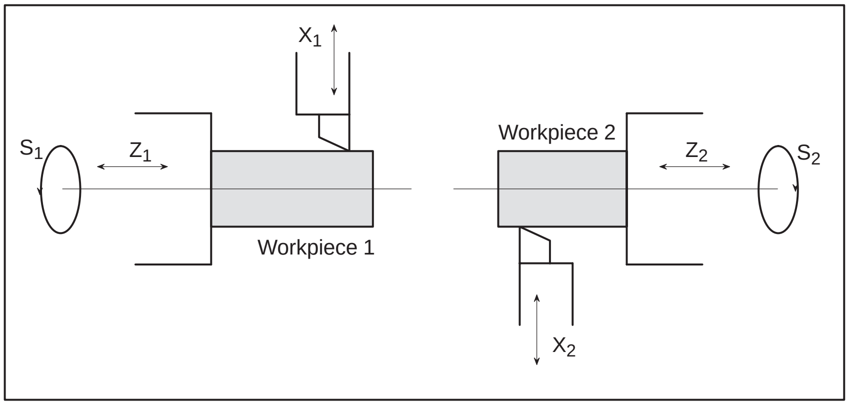


Spindle control

Speed commands for the spindles and feed–per–rotation operations based on the feedback pulses sent from the position coder are not exchanged by this composite control function.

- S command output: Output to the spindle connected to each path. If command switching is necessary, change the analog command voltage by using the SLSPA and SLSBP signals.
- Feed per rotation, threading, spindle fluctuation detection: The feedback pulses from the position coder connected to each path are used. Pulse switching cannot be performed in the CNC. If pulse switching is required, a pulse switching circuit must be added to the machine.
- Constant surface speed control: The spindle connected to each path is controlled by calculating the spindle speed from the absolute coordinate on the X axis of the path.
- Cs axis control: This composite control function cannot perform Cs axis switching.

Example: Performing separate control and composite control with a machine having the following configuration:



- Separate control: System X_1–Z_1, S_1 is controlled by a program for tool post 1
  System X_2–Z_2, S_2 is controlled by a program for tool post 2

  Workpiece 1 is machined using the program for tool post 1, while workpiece 2 is machined using the program for tool post 2.
- Composite control 1 (switching between X_1 and X2):
  System X_2–Z_1, S_1 is controlled by a program for tool post 1.
  System X_1–Z_2, S_2 is controlled by a program for tool post 2.

  Workpiece 1 is machined using the program for tool post 1, while workpiece 2 is machined using the program for tool post 2. Feed per rotation, threading, and constant surface speed control are enabled.
- Composite control 2 (switching between Z_1 and Z_2):
  System X_1–Z2, S_1 is controlled by a program for tool post 1.
  System X_2–Z_1, S_2 is controlled by a program for tool post 2.

  Workpiece 1 is machined using the program for tool post 2, while workpiece 2 is machined using the program for tool post 1. Feed per rotation, threading, and constant surface speed control are disabled.

In the configuration shown above, composite control 1 is used.

Tool offset in composite control

When switching between separate control and composite control is performed, the already–set offset value and tool–tip radius compensation are left as is. Therefore, after the control mode is changed, a T code must be specified to set appropriate offset values.

Move command after switching between separate control and composite control

After performing switching between separate control and composite control during automatic operation, never specify the setting of a coordinate system nor a move command for the switched axes within the two blocks including the currently executed block (or within three blocks when tool–nose radius compensation is being performed).

Example: Starting composite control to switch between the X_1 axis and X_2 axis at the N200 block

N190 ;
N200 M55;(This M code command starts composite control.)
N210 ;
N220 ;
N230 ;

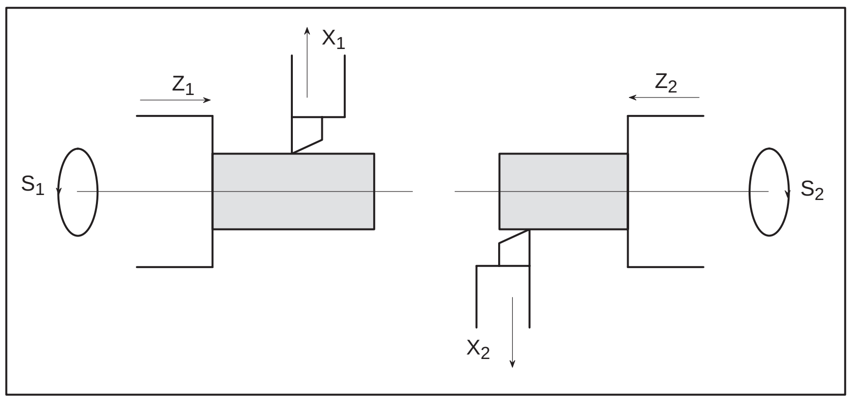
In the above example, no move command can be specified for the X axis in the N210 block (also in the N220 block when tool–nose radius compensation is being performed). If M55 is an M code that is not buffered, however, coordinate system setting or a move command for the X axis can be specified in the N210 and subsequent blocks.

Examples

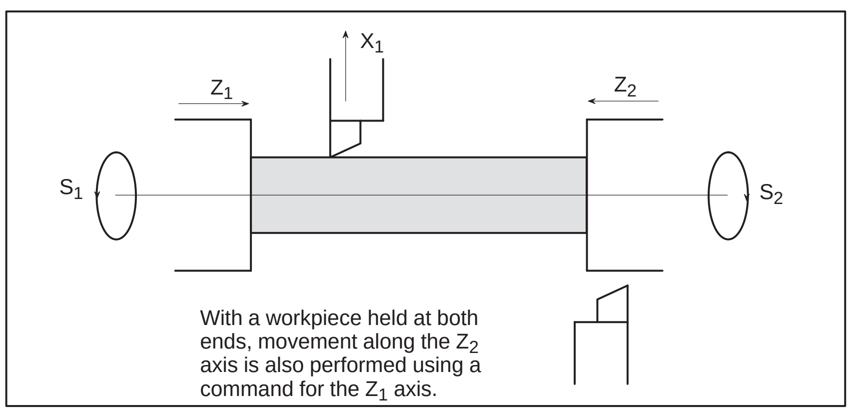
Performing separate control and Z_1–Z_2 axes synchronization control

(1) Machine configuration

(i) When separate control is applied



(ii) When Z_1–Z_2 axes synchronization control is applied



(2) Parameter setting

	Tool post 1								Tool post 2							
	#7	#6	#5	#4	#3	#2	#1	#0	#7	#6	#5	#4	#3	#2	#1	#0
0380			0	0	0	0	0	0			0	0	0	0	1	0
0381			0	0	0	0	0	0			0	0	0	0	1	0
0382			0	0	0	0	0	0			0	0	0	0	0	0

	Tool post 1	Tool post 2
0320	0	0
0321	0	2
0322	0	0
0323	0	0
0324	0	0
0325	0	0
0686	0	0
0687	0	100 to 1000
0688	0	0
0689	0	0
0690	0	0
0691	0	0

- To synchronize the Z_2 axis with the Z_1 axis, set 2 in parameter 0321 for tool post 2.
- Since the positive direction of the Z_1 axis and that of the Z_2 axis are opposite to one another, mirror image is applied to perform synchronization. For this purpose, set SMR2 to 1 for tool post 2.
- Since movement along the Z_1 axis and movement along the Z_2 axis must be equal, synchronization error detection is performed. Set SER2 to 1 for tool post 2. Set a value of about 100 to 1000 as the synchronization error limit in parameter 0687 for tool post 2. (This value varies depending on the machine.)
- During synchronization, the difference in positional deviation between the Z_1 axis and Z_2 axis is indicated in parameter 0693 for tool post 2.

(3) Signal operation

- To start synchronous movement along the Z_1 axis and Z_2 axis, set the SYN2S signal to 1.
- To cancel synchronization, set the SYN2S signal to 0.
- When an emergency stop, NC reset, or alarm occurs, also set SYN2S to 0.
- Leave all signals other than SYN2S set to 0.

(4) Sample program

Tool post 1	Tool post 2	
:	:	Perform machining separately on each tool post.
Z80. ;	Z150. ;	Move the workpiece and chuck to a predetermined position.
M200 ;	M200 ;	Wait until movement is completed.
M61 ;		Clamp the workpiece and start synchronization.
M3 S800 ;		Turn the spindle clockwise.
Z–25. ;		Movement along the Z_1 axis
:		Machining through movement along the X_1 and Z_1 axes
M62 ;		Cancel synchronization and unclamp the workpiece.
M201 ;	M201 ;	Wait until synchronization is released.
;	;	Dummy block (specifying no move command)
: :		Perform machining separately for each tool post.

Where, the following is assumed:

M61 is an M code that is used to clamp the workpiece and cause the SYN2S signal to become 1. M62 is an M code that is used to turn the SYN2S signal to 0 and unclamp the workpiece.

(5) Note

NOTE

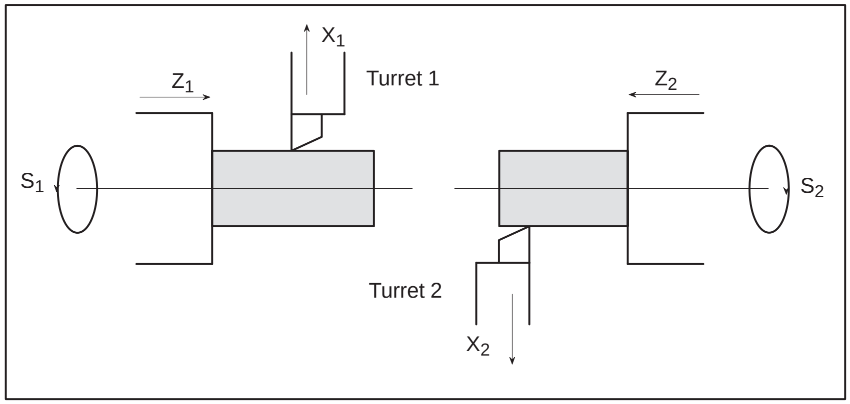
1 Operation to match the speeds of spindles S_1 and S_2 is required. For example, a spindle command for tool post 1 is output to both S_1 and S_2.

2 When synchronous movement is performed with the master axis and slave axis clamped mechanically as shown in the above example, a torque limit should be applied to the slave axis (the Z_2 axis in the example) in some cases.

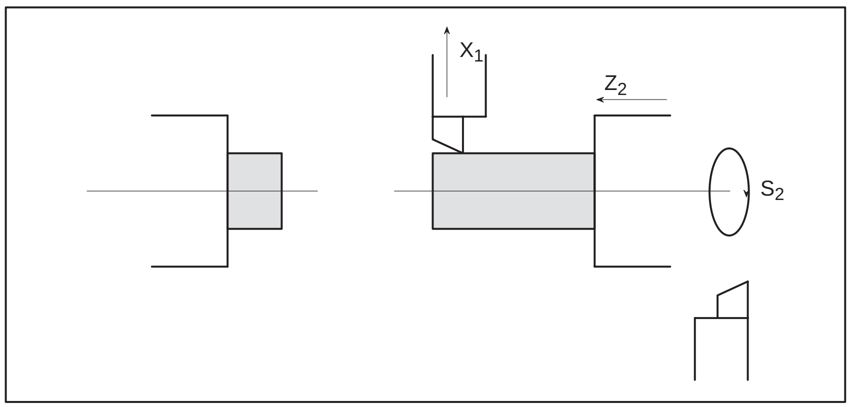
Example of performing separate control and X_1–Z_2 axes interpolation

(1) Machine configuration

(i) When separate control is applied



(ii) When X_1–Z_2 axes interpolation is performed



X_1–Z_2 axes interpolation may be performed using either of the following two methods:

(a) The X_2–Z_2 axes are controlled using a program for tool post 2, and the X_1 axis is synchronized with the X_2 axis. The X_2 axis of tool post 1 is parked. Axis control is not performed for tool post 1.

(b) Composite control is performed to exchange the X_1 and X_2 axes between the tool posts. Axis control is not performed for tool post 1.

In the following, the method of using synchronization control and that of using composite control will be explained separately.

(a) When synchronization control is applied

(a.2) Parameter setting

	Tool post 1 #7	#6	#5	#4	#3	#2	#1	#0	Tool post 2 #7	#6	#5	#4	#3	#2	#1	#0
0380			0	0	0	0	0	0			0	0	0	0	0	0
0381			0	0	0	0	0	0			0	0	0	0	0	0
0382			0	0	0	0	0	0			0	0	0	0	0	0

	Tool post 1	Tool post 2
0320	1	0
0321	0	0
0322	0	0
0323	0	0
0324	0	0
0325	0	0
0686	0	0
0687	0	0
0688	0	0
0689	0	0
0690	0	0
0691	0	0

- To synchronize the X_1 axis with the X_2 axis, set 1 in parameter 0320 for tool post 1.
- For both the X_1 and X_2 axes, movement in the positive direction of the coordinate is always away from the workpiece center. So, mirror image is not required.
- Since the X_2 axis is parked, synchronization error detection is not performed.
- During synchronization, the difference in positional deviation between the X_2 axis and X_1 axis is indicated in parameter 0692 for tool post 1.

(a.3) Signal operation

- To start synchronous movement along the X_2 and X_1 axes, set the SYN1M and PK1S signals to 1.
- To cancel synchronization, set the SYN1M and PK1S signals to 0.
- When an emergency stop, NC reset, or alarm occurs, also set SYN1M and PK1S to 0.
- Leave the signals other than SYN1M and PK1S set to 0.

(a.4) Sample program

Tool post 1	Tool post 2	
:	:	Perform machining separately on each tool post.
Z0 ;	Z20. ;	Move each workpiece to a predetermined position.
X120. ;	X120. ;	Movement along each X axis to the synchronization start position ($X_1 = X_2$)
M200 ;	M200 ;	Wait until movement has been completed.
	M55 ;	Start synchronization between the X_2 and X_1 axes and parking of the X_2 axis.
	T0212 ;	Set an offset for turret 1.
	S1000 M4 ;	
	G0 X30. Z55.;	Machining through movement along the X_1 axis and Z_2 axis
	G1 F0.2 W–15. ;	
	M56 ;	Cancel synchronization and parking.
M201 ;	M201 ;	Wait until synchronization has been released.
;	;	Dummy block (specifying no move command)
:	:	Perform machining separately for each tool post.

Where, the following is assumed:

M55 is the M code that is used to start the control of turret 1 using a program for tool post 2. M56 is the M code that is used to cancel the control of turret 1 using the program for tool post 2.

(a.5) Note

NOTE

During synchronization control for the X axes, no move command can be issued from tool post 1 to the X_1 axis. However, movement along the Z_1 axis is specified from tool post 1.

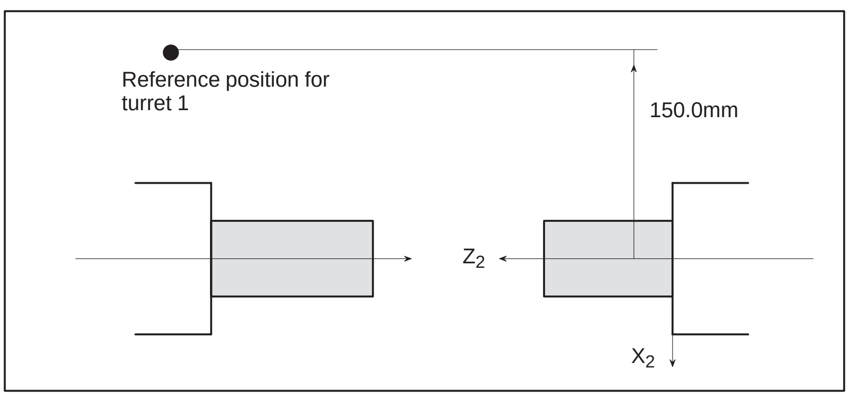
(b) When composite control is applied

(b.2) Parameter setting

	Tool post 1 #7	#6	#5	#4	#3	#2	#1	#0	Tool post 2 #7	#6	#5	#4	#3	#2	#1	#0
0383			0	0	0	0	0	0			0	0	0	0	0	1
0384			0	0	0	0	0	0			0	0	0	0	0	1
0385			0	0	0	0	0	1			0	0	0	0	0	0

	Tool post 1	Tool post 2
0326	0	1
0327	0	0
0328	0	0
0329	0	0
0860	0	–150000
0861	0	0
0862	0	0
0863	0	0
0864	0	0

- To perform composite control to exchange the X_1 and X_2 axes, set 1 in parameter 0326 for tool post 2.
- The direction of the coordinates on the X_1 axis is opposite to that of the coordinates on the X_2 axis, so set 1 in MCD1 for tool post 2.
- To automatically set the position of turret 1 in the workpiece coordinate system for tool post 2 at the start of composite control, set 1 in MPM1 for tool post 2.
- To automatically set the position of turret 1 in the workpiece coordinate system for tool post 1 at the end of composite control, set 1 in MPM1 for tool post 1.
- When the X coordinate of the reference position for turret 1 is –150.0 mm in the workpiece coordinate system for tool post 2, as shown in the figure below, set –150000 in MPRM1 for tool post 2 to perform automatic coordinate system setting.



(b.3) Signal operation

- To start composite control for the X_2 and X_1 axes, set the MIX1 signal to 1.
- To cancel composite control, set the MIX1 signal to 0.
- When an emergency stop, NC reset, or alarm occurs, also set MIX1 to 0.
- Leave all signals other than MIX1 set to 0.

(b.4) Sample program

Tool post 1	Tool post 2	
:	:	Perform machining separately on each tool post.
Z0 ;	Z20. ;	Move each workpiece to a predetermined position.
	X120. ;	Movement along the X_2 axis to an appropriate position to prevent interference
M200 ;	M200 ;	Wait until movement has been completed.
	M55 ;	Start composite control for the X_2 and X_1 axes. (The position of turret 1 is set in the workpiece coordinate system for tool post 2.)
	;	Dummy block (specifying no move command)
	M0212 ;	Set an offset for turret 1.
	S1000 M4 ;	
	G0 U10. W–20.;	Machining through movement along X_1 axis and Z_2 axis
	G1 F0.2 W–15. ;	
	M56 ;	Cancel composite control. (The position of turret 1 is set in the workpiece coordinate system for tool post 1.)
M201 ;	M201 ;	Wait until composite control is released.
;	;	Dummy block (specifying no move command)
:	:	Perform machining separately on each tool post.

Where, the following is assumed:

M55 is the M code that is used to start the control of turret 1 using a program for tool post 2. M56 is the M code that is used to cancel control of turret 1 using the program for tool post 2.

(b.5) Note

NOTE

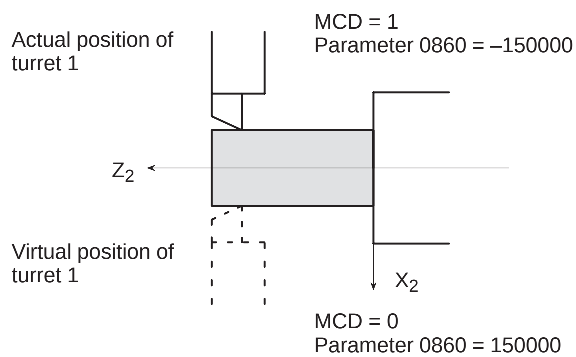
1 The automatic setting of a coordinate system is not always needed at the start and end of composite control. When automatic setting is not performed, specify the setting of an appropriate coordinate system in a program.

2 During X–axis synchronization control, move commands for the X axes can be issued to tool post 1 to perform movement along the X_2 axis.

3 In the parameter setting explained previously, turret 1 is positioned on the negative side of the X coordinate in the workpiece coordinate system for tool post 2. Therefore, to move turret 1 toward the workpiece center, for example, specify U+10. To move the turret away from the center, specify U–10. Note that the specified sign is opposite to that for normal operation. If this specification method proves to be inconvenient, set the parameters as follows:

MCD1 = 0
Parameter 0860 = 150000

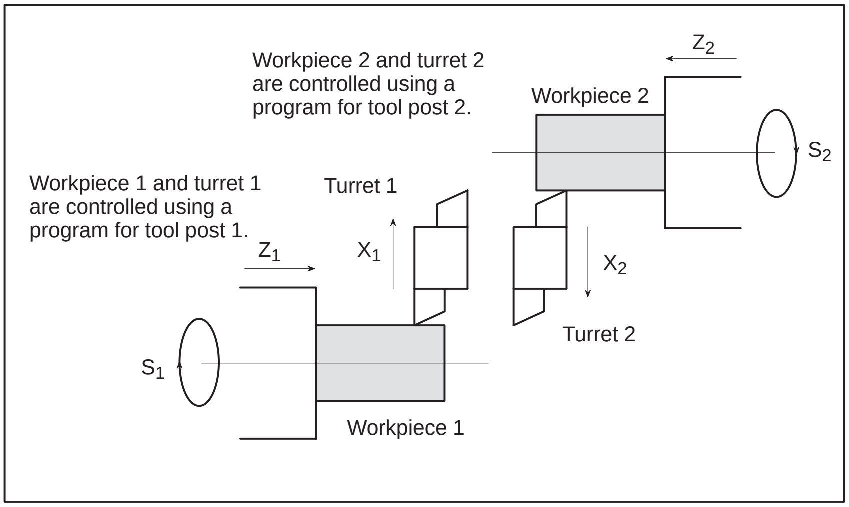
Then, turret 1 is set as if it were on the positive side of the X coordinate.



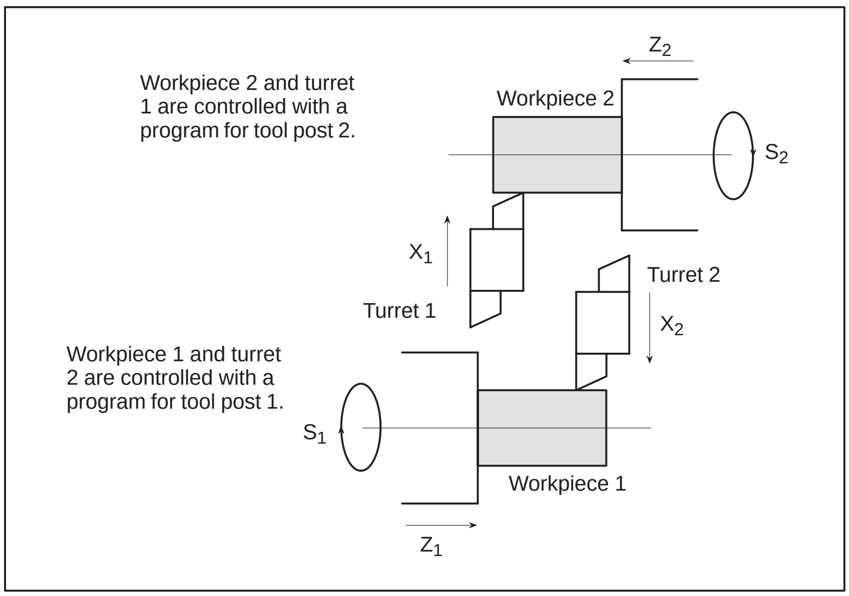
Example of performing separate control and X_1–Z_2, X_2–Z_1 composite control

(1) Machine configuration

(i) When separate control is applied



(ii) When X_1–Z_2 axes interpolation is performed

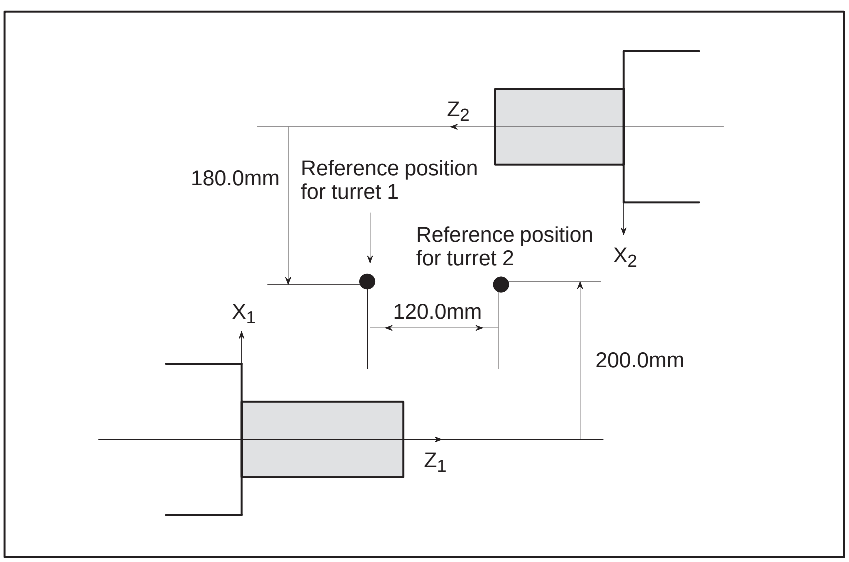


(2) Parameter setting

	Tool post 1 #7	#6	#5	#4	#3	#2	#1	#0	Tool post 2 #7	#6	#5	#4	#3	#2	#1	#0
0383			0	0	0	0	0	1			0	0	0	0	0	1
0384			0	0	0	0	0	1			0	0	0	0	0	1
0385			0	0	0	0	0	1			0	0	0	0	0	1

	Tool post 1	Tool post 2
0326	0	1
0327	0	0
0328	0	0
0329	0	0
0860	200000	180000
0861	0	0
0862	0	0
0863	0	0
0864	0	0

- To perform composite control to exchange the X_1 axis and X_2 axis, set 1 in parameter 0326 for tool post 2.
- The direction of the coordinates on the X_1 axis is opposite to that of the coordinates on the X_2 axis, so set 1 in MCD1 for tool post 2.
- In the workpiece coordinate system for each tool post, to automatically set the position of the turret of the other tool post at the start of composite control, set 1 in MPM1 for tool posts 1 and 2.
- In the workpiece coordinate system for each tool post, to automatically set the position of the turret of the tool post at the end of composite control, set 1 in MPS1 for tool posts 1 and 2.
- In the relationship between the workpiece coordinate systems for the tool posts and the reference positions shown below, to perform automatic coordinate system setting, set 200000 in parameter 0860 for tool post 1, and set 180000 in parameter 0860 for tool post 2.



(3) Signal operation

- To start composite control for the X_2 axis and X_1 axis, set the MIX1 signal to 1.
- To cancel composite control, set the MIX1 signal to 0.
- When an emergency stop, NC reset, or an alarm occurs, set MIX1 to 0.
- Leave all signals other than MIX1 set to 0.

(4) Sample program

Tool post 1	Tool post 2	
:	:	Perform machining separately on each tool post.
M350 ;	M350;	Wait for the start of composite control.
	M55 ;	Start composite control for the X_1 and X_2 axes.
M351 ;	M351 ;	Completion of the start of composite control
;	;	Dummy block (specifying no move command) Select the tool to be used for composite control, and set an offset.
T0313	T0212 ;	Select the tool to be used for composite control, and set an offset.
G50 W120. ;	G50 W120. ;	Shift the Z–axis workpiece coordinate system.
S1000 M4 ;	S1500 W120. ;	
G0 X20. Z15	G0 X15. Z30. ;	Machining under composite control
G1 F0.5 W–8. ;	G1 F0.1 W–5. ;	
M360 ;	M360 ;	Wait for the end of composite control.
	M56 ;	End of composite control
M361 ;	M361 ;	Completion of the end of composite control
;	;	Dummy block (specifying no move command)
G50 W–120. ;	G50 W–120. ;	Shift the Z–axis workpiece coordinate system.
:	:	Perform machining separately on each tool post.

Where, the following is assumed:

M55 is the M code that is used to start composite control. M56 is the M code that is used to cancel composite control.

(5) Note

> **NOTE**
> Automatic coordinate system setting is not always needed at the start and end of composite control. When automatic setting is not performed, specify the setting of an appropriate coordinate system in a program.

Others

- The above description covered single synchronization control or composite control operation. In actual operation, more than one synchronization control and composite control operation can be selected and executed simultaneously. First, set all the required parameters. Then, select the desired synchronization or composite control operations by using signals. At this time, be careful to ensure that no axis is subject to more than one synchronization or composite control operation at any one time.
- For each synchronization or composite control operation, only one axis pair can be set. When multiple axis combinations are required, use the programmable parameter input function (G10) to change the parameter setting in a program. Before a parameter setting can be changed, synchronization control and composite control must be canceled.

Example: Changing the parameter settings so that the Z_2 axis is synchronized with the Y_1 axis
(The following program is executed for tool post 2 to set 4 in parameter 0321 for tool post 2:)

:	
G10 L50 ;	Start parameter setting.
N0321 P4 ;	Set 4 in parameter 0321.
G11 ;	End parameter setting.
:	

Blocks G10 to G11 must be executed when neither synchronization control nor composite control is being performed for the Z_2 axis and Y_1 axis.

Signal

Synchronization control start signals (tool post 1) SYN1M to SYN7M <G237#0 to G237#4>

[Classification] Input signal

[Function] These signals start synchronization control using the corresponding axis, the first to the seventh axis, of tool post 1 as the slave axis.

[Output condition] Synchronization control is performed so that the axis specified with signal SYN1M to SYN7M is used as the slave axis for the master axis set in parameter 320 to 323.

Synchronization control start signals (tool post 2) SYN1S to SYN4S <G1437#0 to G1437#3>

[Classification] Input signal

[Function] These signals start synchronization control using the corresponding axis, the first to the fourth axis, of tool post 2 as the slave axis.

[Output condition] Synchronization control is performed so that the axis specified with signal SYN1S to SYN4S is used as the slave axis for the master axis set in parameter 320 to 323.
Even when signal SYN1M to SYN7M or SYN1S to SYN4S is turned on, it is ignored unless an axis number is set in corresponding parameter SAX1 to SAX7.

Composite control start signals MIX1 to MIX4 <G1437#4 to G1437#7>

[Classification] Input signal

[Function] Composite control is performed to switch move commands between each of the first to fourth axes of tool post 2 and the corresponding axis of tool post 1.

[Output condition] Move commands are switched between the axis specified with parameters 326 to 329 and the axis specified with signal MIX1 to MIX4.
Even when signal MIX1 to MIX4 is turned on, it is ignored unless an axis number is set in corresponding parameter 326 to 329.

Synchronization control parking signals (tool post 1) PK1M to PK7M <G238#0 to G238#4>

[Classification] Input signal

[Function] These signals place the corresponding axis of tool post 1 in the parking state.

[Output condition] Movement along an axis of tool post 1 under synchronization control is stopped, and the axis is parked.

Synchronization control parking signals (tool post 2) PK1S to PK4S <G1438#0 to G1438#3>

[Classification] Input signal

[Function] These signals place the corresponding axis of tool post 2 in the parking state.

[Output condition] Movement along an axis of tool post 2 under synchronization control is stopped, and the axis is parked.
The parking signals are valid only while synchronization control is applied. If a parking signal is turned on while synchronization control is not applied, it is ignored.

Axis recomposition signals (tool post 1) SYN1OM to SYN7OM <F189#0 to F189#4>

[Classification] Output signal

[Function] These signals post notification that each of the first to seventh axes of tool post 1 is under synchronization or composite control.

Axis recomposition signals (tool post 2) SYN1OS to SYN4OS <G1389#0 to G1389#3>

[Classification] Output signal

[Function] These signals post notification that each of the first to fourth axes of tool post 2 is under synchronization or composite control.
When starting or ending synchronization or composite control, check that the corresponding signal SYN□O□has been changed correctly.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G237				SYN7M	SYN4M	SYN3M	SYN2M	SYN1M
G238				PK7M	PK4M	PK3M	PK2M	PK1M
G1437	MIX4	MIX3	MIX2	MIX1	SYN4S	SYN3S	SYN2S	SYN1S
G1438					PK4S	PK3S	PK2S	PK1S
F189				SYN7OM	SYN4OM	SYN3OM	SYN2OM	SYN1OM
F1389					SYN4OS	SYN3OS	SYN2OS	SYN1OS

Parameter

The following parameters are provided for each of tool posts 1 and 2. Parameters are provided for each of the first to sixth axes. For tool post 2, the first to fourth axes can be used. Parameters related to the following must be set:

Axis used as the slave in synchronization control

Axis to be switched in composite control

Parameters related to synchronization control

	#7	#6	#5	#4	#3	#2	#1	#0
0380	NRST	SPERR	SMR6	SMR5	SMR4	SMR3	SMR2	SMR1

[Data type] Bit

SMR1 to SMR6 1 : Uses the mirror image function for synchronization control. (Movement along the master axis and movement along the slave axis are made in opposing directions.)

0 : Does not use the mirror image function for synchronization control. (Movement along the master axis and movement along the slave axis are made in the same direction.)

SPERR 1 : The synchronization error is the sum of the difference in the positional deviation and the difference in acceleration/deceleration between the master axis and slave axis.

0 : The synchronization error is the difference in the positional deviation between the master axis and slave axis. (The same value is set for both of tool posts 1 and 2.)

When the acceleration/deceleration time constant for the master axis differs from that for the slave axis, this bit must be set to 1.

NRST 1 : A reset operation does not cancel the axis recomposition function.

0 : A reset operation cancels the axis recomposition function.
(The same value is set for both of tool posts 1 and 2.)

	#7	#6	#5	#4	#3	#2	#1	#0
0381				SER6	SER4	SER3	SER2	SER1

[Data type] Bit

SER1 to SER6 1 : When movement is made along both the master and slave axes in synchronization mode, the positional deviations for the corresponding axes are compared. If the difference is equal to or greater than the set value, an alarm is issued. (Synchronization error detection)
If one of the axes is parked or placed in the machine lock state, this check is not performed.
0 : The positional deviation check explained above is not made in synchronization mode.

When the master and slave axes are on the same tool post, synchronization error detection cannot be performed.

	#7	#6	#5	#4	#3	#2	#1	#0
0382				PKU6	PKU4	PKU3	PKU2	PKU1

[Data type] Bit

PKU1 to PKU6 1 : When parking is performed, the absolute/relative coordinates are updated. The machine coordinates are not updated.
0 : When parking is performed, none of the absolute, relative, machine coordinates are updated.

0320	Master axis with which the X axis is synchronized
0321	Master axis with which the Z axis is synchronized
0322	Master axis with which the third axis is synchronized
0323	Master axis with which the fourth axis is synchronized

[Data type] Byte

[Valid data range] 0 to 4, 200 to 204

Set the master axis with which a corresponding axis is synchronized when the axis recomposition function is used.
<<When synchronization control is applied between the tool posts>>
Set the axis number of the corresponding synchronization master axis in the parameter of the synchronization slave axis. An axis number not greater than the maximum number of controlled axes cannot be set.

Example: When synchronizing the Z2 axis with the Z1 axis

	Tool post 1	Tool post 2
No.320	0	0
No.321	0	0
No.322	0	0
No.323	0	0

<<When synchronization control is applied with one tool post>>
In the parameter of the synchronization slave axis, set a value obtained from (the axis number of the corresponding master axis + 200). A number not greater than the maximum number of controlled axes cannot be set.

Example: When synchronizing the Y1 axis with the Z1 axis

	Tool post 1	Tool post 2
No.320	0	0
No.321	0	0
No.322	0	0
No.323	202	0

NOTE
1 When 0 is set in the parameter of an axis, that axis is not used as the slave axis synchronized with another axis.
2 When the same number is set in more than one parameter, multiple slave axes can be synchronized with a single master axis.

0686	Limit on the difference in the positional deviation during synchronization control where the X axis is used as the slave axis
0687	Limit on the difference in the positional deviation during synchronization control where the Z axis is used as the slave axis
0688	Limit on the difference in the positional deviation during synchronization control where the third axis is used as the slave axis
0689	Limit on the difference in the positional deviation during synchronization control where the fourth axis is used as the slave axis
0690	Limit on the difference in the positional deviation during synchronization control where the seventh axis is used as the slave axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

When synchronization control is applied using each axis as a slave axis, the limit on the difference in positional deviation from the corresponding master axis is set in these parameters.

NOTE
1 These parameters are valid when bit 0 (ERRX) to bit 5 (ERR8) of parameter 0381 are set to 1.
2 These parameters are valid with the 0–TTC.

0692	Difference in the positional deviation during synchronization control where the X axis is used as the slave axis
0693	Difference in the positional deviation during synchronization control where the Z axis is used as the slave axis
0694	Difference in the positional deviation during synchronization control where the third axis is used as the slave axis
0695	Difference in the positional deviation during synchronization control where the fourth axis is used as the slave axis
0696	Difference in the positional deviation during synchronization control where the seventh axis is used as the slave axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] –32768 to 32767

When synchronization control is applied using each axis as a slave axis, the difference in the positional deviation from the corresponding master axis is set in these parameters.

(Difference in the positional deviation) =
(Amount of positional deviation for the master axis) ±
(Amount of positional deviation for the slave axis)

+: Mirror image ON
–: Mirror image OFF

Parameters related to composite control

	#7	#6	#5	#4	#3	#2	#1	#0
0383				MPM6	MPM4	MPM3	MPM2	MPM1

[Data type] Bit

MPM1 to MPM6
1 : Automatically sets the workpiece coordinate system as the coordinate system for composite control at the start of composite control.
0 : Does not automatically set the workpiece coordinate system at the start of composite control.

When the workpiece coordinate system is automatically set at the start of composite control, the workpiece coordinate system is calculated from the current machine coordinate and the composite control workpiece coordinate of the reference position on each axis, which is set in parameters 0860 to 0865.

	#7	#6	#5	#4	#3	#2	#1	#0
0384				MPS6	MPS4	MPS3	MPS2	MPS1

[Data type] Bit

MPS1 to MPS6 1 : Automatically sets the workpiece coordinate system as the coordinate system for separate control at the end of composite control.

0 : Does not automatically set the workpiece coordinate system at the end of composite control.

When the workpiece coordinate system is automatically set at the end of composite control, the workpiece coordinate system is calculated from the current machine coordinate and the workpiece coordinate of the reference position on each axis, which is set in parameters 0708 to 0711, 0825, and 0826.

	#7	#6	#5	#4	#3	#2	#1	#0
0385				MCD7	MCD4	MCD3	MCD2	MCD1

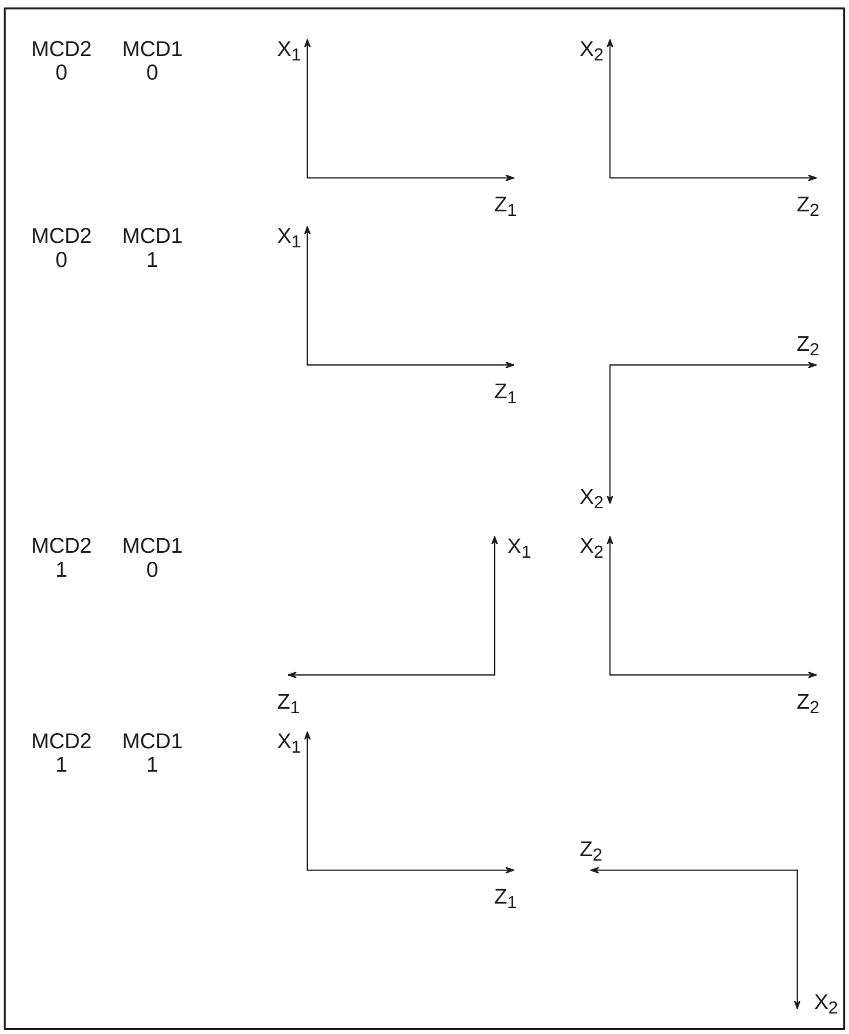
[Data type] Bit

MCD1 to MCD7 1 : The direction of the coordinate system for the corresponding axis is reversed, so mirror image is used for composite control. (Movement along an axis of tool post 1 and movement along an axis of tool post 2 are made in opposite directions to one another.)

0 : The direction of the coordinate system for the corresponding axis is the same, so mirror image is not used for composite control. (Movement along an axis of tool post 1 and movement along an axis of tool post 2 are made in the same direction.)

The relationship between the coordinate systems of the axes to be switched in composite control is set. This setting determines the direction of movement along the corresponding axis. This setting is also used when automatic coordinate system setting is performed when composite control switching is applied.

Example: When the X axes and Z axes of tool posts 1 and 2 have the relationships shown, MCD1 and MCD2 are set as follows (when switching between the X axes and between the Z axes is performed):



0326	Axis of tool post 1 with which composite control is performed for the X axis of tool post 2
0327	Axis of tool post 1 with which composite control is performed for the Z axis of tool post 2
0328	Axis of tool post 1 with which composite control is performed for the third axis of tool post 2
0329	Axis of tool post 1 with which composite control is performed for the fourth axis of tool post 2

[Valid data range] 0 to 4

When the axis recomposition function is applied with the 0–TTC, these parameters set the axis of tool post 1 with which composite control is to be performed for each axis of tool post 2.

Example 1: When composite control is applied to switch between the X1 axis and X2 axis

	Tool post 1	Tool post 2
No.326	0	1
No.327	0	0
No.328	0	0
No.329	0	0

Example 2: When composite control is applied to switch between the Y1 axis and X2 axis

	Tool post 1	Tool post 2
No.326	0	4
No.327	0	0
No.328	0	0
No.329	0	0

NOTE

1 These parameters are set only for tool post 2.
2 When 0 is set in the parameter for an axis, that axis is not subject to control switching in composite control.
3 The same number can be set in two or more parameters. However, the axes corresponding to these parameters are not placed under composite control simultaneously.

0860	Coordinate of the reference position for the axis corresponding to the X axis in the coordinate system in composite control
0861	Coordinate of the reference position for the axis corresponding to the Z axis in the coordinate system in composite control
0862	Coordinate of the reference position for the axis corresponding to the third axis in the coordinate system in composite control
0863	Coordinate of the reference position for the axis corresponding to the fourth axis in the coordinate system in composite control
0864	Coordinate of the reference position for the axis corresponding to the seventh axis in the coordinate system in composite control

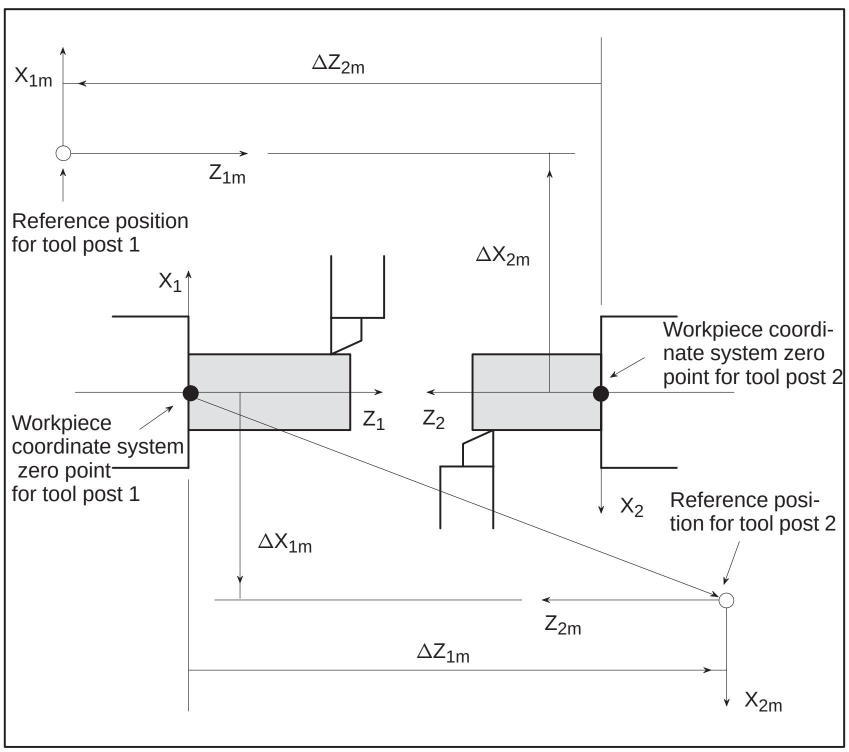
[Data type] Two–word

[Unit of data] Least input increment

[Valid data range] 0 to ±99999999

The coordinate of the reference position for the corresponding axis in the coordinate system in composite control can be set for each axis sequentially.

> **NOTE**
> These parameters are valid when bits 0 (MPMX) to 5 (MPM7) of parameter 0383 are all set to 1.



In the workpiece coordinate system for tool post 1, the reference position for tool post 2 is at (ΔX_{1m}, ΔZ_{1m}). In the workpiece coordinate system for tool post 2, the reference position for tool post 1 is at (ΔX_{2m}, ΔZ_{2m}).

In this case, set ΔX_{1m} in parameter 860 for tool post 1, and set ΔX_{2m} in parameter 860 for tool post 2.

If parameter MPMX is set to 1 at the start of composite control, a workpiece coordinate system is set so that the following expressions are satisfied:

X_1 = (parameter 860 for tool post 1) $\pm$ (X_2 machine coordinate)

When MCD1 for tool post 1 = 0: +
When MCD1 for tool post 1 = 1: –

X_2 = (parameter 860 for tool post 2) $\pm$ (X_1 machine coordinate)

When MCD1 for tool post 2 = 0: +
When MCD1 for tool post 2 = 1: –

If parameter MPS1 is set to 1 at the end of composite control, a workpiece coordinate system is set so that the following expressions are satisfied:

X_1 = (PRSX of parameter 708 for tool post 1) +
(X_1 machine coordinate)

X_2 = (PRSX of parameter 708 for tool post 2) +
(X_2 machine coordinate)

Alarm and message

PS Alarm

No.	Contents
225	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number patameter setting. 2 When there is a mistake in control commanded. Modify the program or the patameter.
226	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the patameter.
229	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system over load. 2 The above condition occured in CMC devices (hardware) and synchro–state could not be kept. (This alarm is not generated in normal use conditions.)

Servo Alarm

No.	Contents
407	The difference in synchronous axis position deviation exceeded the set value.

If any of the above alarms is issued, the axis recomposition function is released for all axes.

Note

Cautions and notes relating to both synchronization control and composite control

CAUTION

1 Before synchronization control or composite control is started or terminated, the axes to be controlled must be stopped.
2 The corresponding axes under synchronization control and composite control must have the same least input increment, detection unit, and diameter/radius specification. If these settings do not match, the amounts of travel along the axes will not be equal.
3 While synchronization control or composite control is being performed, do not change the related parameter settings and parking signals.

NOTE

1 Multiple axes can be placed in the synchronization state or composite control state simultaneously. However, an axis cannot be synchronized with more than one axis at the same time. An axis currently used by switching in composite control cannot be synchronized with another axis. In addition, duplicate switching is not allowed.
2 When an emergency stop, servo–off state, or servo alarm has been released, perform reference position return and coordinate system setting before starting synchronization control or composite control.
3 When synchronization control or composite control is started, reference position return for the target axes must have been completed after power–on, or the reference positions for the axes must have been established by using an absolute pulse coder.
4 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are performed independently, regardless of the axis recomposition function.
5 The axis recomposition function cannot be selected at the same time as the simple synchronous control or balance cut function.

Cautions and notes related only to synchronization control

CAUTION
During synchronization control, never issue move commands for the synchronization axis on the synchronization slave side.

NOTE
1 Whenever possible, set the same acceleration/deceleration time constant and servo–related parameters for the corresponding axes placed under synchronization control.
2 When operation which changes only the workpiece coordinate system (while the machine remains stationary), such as a workpiece coordinate system setting/shift and geometry offset command, is performed on the synchronization master side, the workpiece coordinate system on the synchronization slave side is not affected.
3 When a wear offset command or tool–nose radius compensation is executed on the synchronization master side, the tool path on the synchronization slave side is shifted by the offset, but the offset amount is not set. (No offset vector is created.)

Limitations imposed during synchronization control and composite control

Function	During synchronization control	During composite control
Acceleration/deceleration control	Acceleration/deceleration of the same type is performed for the synchronization axes. A separate time constant is used for each axis.	The acceleration/deceleration type of the specified path is used. A separate time constant is used for each axis.
Linear acceleration/deceleration after interpolation	Disabled for all axes during synchronization control.	Disabled for all axes during composite control.
Feedrate clamp	Clamped on the synchronization master side.	Clamped on the specified path side.
Reference position return	Enabled unless the synchronization master axis is parked. However, the synchronization slave axis is not placed in the reference position return completion state.	Enabled only for those axes which are not affected by composite control.
Reference position return check	Enabled only for the synchronization master axis.	Enabled only for those axes which are not affected by composite control.
PMC axis control	Enabled for all axes except the synchronization slave axis.	Enabled.
Polar coordinate interpolation Circular interpolation	Enabled.	Switching between separate control and composite control must be performed in cancel mode.
Handle interrupt	Interrupt can be performed separately regardless of synchronization.	Enabled only for those axes that are not affected by composite control.
X–axis mirror image	Each signal is enabled.	Signals for the specified path are enabled.
Machine lock	Each signal is enabled.	Signals for the specified path are enabled.
Interlock	For the synchronization slave axis, those signals on the synchronization master side are enabled.	Signals for the specified path are enabled.
Override	For the synchronization slave axis, those signals on the synchronization master side are enabled.	Signals for the specified path are enabled.
External deceleration	For the synchronization slave axis, those signals on the synchronization master side are enabled.	Signals for the specified path are enabled.
Skip function	Disabled for the synchronization slave axis.	Enabled only for those axes which are not affected by composite control.
Automatic tool compensation	Disabled for the synchronization slave axis.	Enabled only for those axes which are not affected by composite control.
Tool center	Disabled for the synchronization slave axis.	Enabled only for those axes which are not affected by composite control.
Follow–up	Disabled during synchronization.	Disabled during composite control.
Program restart	Disabled for programs containing the specification of synchronization control.	Disabled for programs containing the specification of composite control.
Cs axis	Synchronization control is disabled.	Composite control is disabled.

Reading coordinates during synchronization control and composite control

During synchronization control or composite control, system variable position information in custom macros, or the current coordinates from the PMC window, are read as follows:

Position information type	During synchronization control	During composite control
Absolute coordinates	Can be read.	Can be read(*1).
Machine coordinates	Can be read.	Can be read.
End point of each block	Can be read only for the master.	Can be read(*1).
Skip signal position	Can be read only for the master.	Cannot be read

*1 Coordinates in the coordinate system used for composite control are read. The relationship with the machine coordinate system differs from that used for separate control.

Canceling synchronization control and composite control

Synchronization control and composite control are canceled when the synchronization signal/composite control signal is turned off. In addition, they are canceled when the following are encountered:

(1) Emergency stop

(2) Reset

(3) Servo alarm

(4) Servo–off

(5) Overtravel

(6) Alarms related to synchronization control or composite control

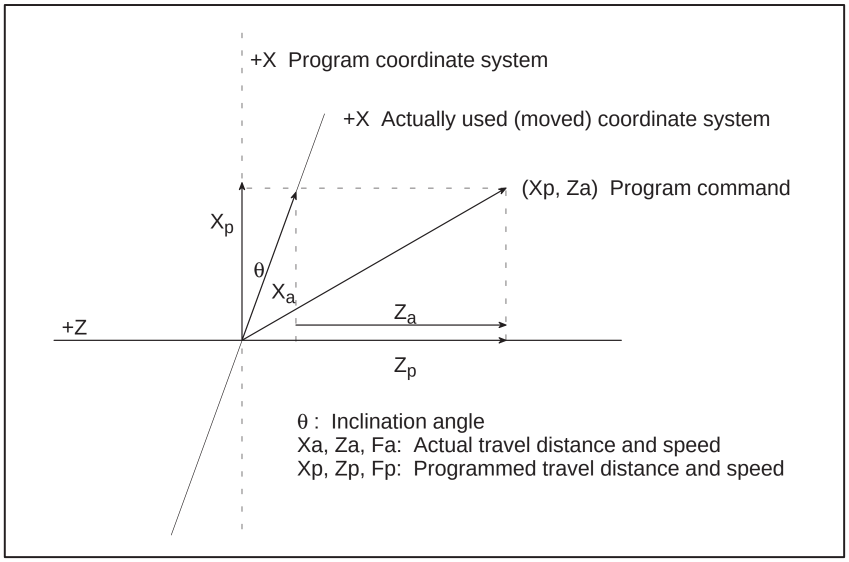
(7) P/S000 alarm

If any of the above occurs, even on one tool post, the synchronization/composite control state is canceled for all axes. If any of the above occurs on one tool post during the execution of the axis recomposition function, the other tool post is automatically placed in the feed hold state (when automatic operation is being performed) or the interlock state (when manual operation is being performed).

1.8 ANGULAR AXIS CONTROL (0–GCC, 0–GSC)

General

In 0–GCC, when the angular axis makes an angle other than 90° with the X axis, the Z axis control function controls the distance traveled along each axis according to the inclination angle. A program, when created, assumes that the X axis and Z axis intersect at right angles. However, the actual distance traveled is controlled according to an inclination angle.



For the 0–GSC, this control is applied between the Y– and Z–axes. In that case, replace ”X” in the above description with ”Y.”

The travel distance for each axis is controlled according to the following expressions.
For the 0–GCC
The X–axis travel distance specified based on the diameter is obtained by:
$Xa = Xp/2\cos\theta$
For the Z–axis travel distance is obtained using the following expression, which is corrected by the inclination of the X–axis:
$Za = Zp - 1/2 \times Xp \times \tan\theta$
The feedrate, which is always specified based on the radius, is obtained as follows:
X–axis rate component
$Fa = Fp/\cos\theta$
For the 0–GSC
The Y–axis travel distance is obtained by:
$Ya = Yp/\cos\theta$
For the Z–axis travel distance is obtained using the following expression, which is corrected by the inclination of the Y–axis:
$Za = Zp - Yp \times \tan\theta$
The feedrate is obtained as follows:
Y–axis rate component
$Fa = Fp/\cos\theta$

The NOZAGC signal can be used to prevent Z–axis movement from being activated due to compensation. No update occurs, so the current position can be maintained. (The machine behaves as if it were in a machine lock state.)

Method of use

Parameter AGLST enables or disables the inclined axis control function. If the function is enabled, the distance traveled along each axis is controlled according to an inclination angle (set at parameter).
Parameter ZRTM1 enables X axis manual reference point return only with a distance along the X axis.

Machine position display

(1) Absolute/Relative position display
The position is displayed using the programmed rectangular coordinate system.

(2) 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 inclination angle operation.

Signal

Angular axis control–related Z–axis compensation movement signal NOZAGC<G133#6> (0–GCC) NOZAGC<G237#5> (0–GSC)

[Function] Prevents compensation–based Z–axis movement during inclined–axis control.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G133		NOZAGC							(0–GCC)
G237			NOZAGC						(G–GSC)

Parameter

		#7	#6	#5	#4	#3	#2	#1	#0	
0036							ZRTM1		AGLST	(0–GCC)
0077				AGLST	ZRTM1					(G–GSC)

[Data type] Bit

ZRTM1 0 : The machine tool is moved along the Z axis during manual reference position return along the X axis (Y axis) under angular axis control.
1 : The machine tool is not moved along the Z axis during manual reference position return along the X axis (Y axis) under angular axis control.

AGLST 0 : Does not perform angular axis control.
1 : Performs inclined axis control.

0755	Inclination angle for X axis control	(0–GCC)
0837	Inclination angle for Y axis control	(0–GSC)

[Valid data range] 20000 to 60000, –20000 to –60000

[Unit of data] 0.001 degree

Warning

WARNING
After inclined axis control parameter setting, be sure to perform manual reference point return operation.

Note

NOTE
1 If Z–axis movement is expected during a manual reference position return on the X–axis, perform an X–axis reference position return first.
2 If an inclination angle close to 0° or ±90° is set, an error can occur. A range from ±20° to ±60° should be used.
3 Before a Z axis reference point return check (G27) can be made, X axis reference point return operation must be completed.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.23	ANGULAR AXIS CONTROL (0–GSC, 0–GSD)
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.23	ANGULAR AXIS CONTROL (0–GCC/00–GCC/0–GCD)

1.9 POSITION SIGNAL OUTPUT (T SERIES)

General

The entire stroke of each axis of the machine is divided into up to 256 areas. The current area of the machine position on each axis is output to the PMC as an eight–bit position signal.

Signal

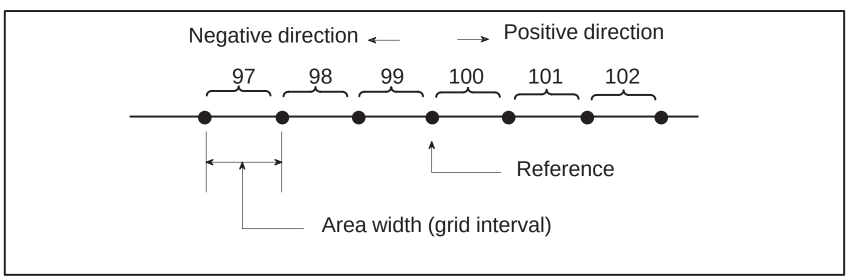
Position signal output signal PX0 to PX7, PZ0 to PZ7 (T series) <F165, F167>

[Classification] Output signal

[Function] (1) The entire stroke of each axis of the machine is divided by the grid interval specified in parameter 0713 or 0714. When each region divided by the grid is referred to as an area, the width of the area (grid interval) formed by this function, which can specify up to 256 areas, must satisfy the following relationship:

$$\frac{\text{(Whole stroke)}}{\text{(Area width)}} \leqq 256$$

(2) The individual areas are assigned numbers as follows: The number of the area in which reference position return is completed is specified in parameter 0113 or 0114; The number increases by one for each successive area of the grid in the positive direction; The number decreases by one for each successive area of the grid in the negative direction.



In the example shown above, the area number for the reference position is 100. Area 255 (eight output bits set to one) comes next to area 0 (eight output bits set to zero) in the negative direction.

(3) The CNC outputs the machine position to the PMC by the area number (8–bit binary code signal) at intervals of 16 ms. The signal is output even during movement. The PMC monitors the signal constantly (LSB: PX0, PZ0; MSB: PX7, PZ7) to keep track of the machine position even during axial movement.

(4) Since the CNC outputs the signal at 16–ms intervals, the area width (grid interval) must be specified as shown below so that no area is missed even during rapid traverse:

$$\text{(Grid interval)} \geqq \frac{R \times 10^6}{60} \times 0.016$$

(When a 1–μm increment system is used)
R: Rapid traverse rate (m/min)

For example, when the rapid traverse rate is 24 m/min, the grid interval must be at least 6,400, calculated as shown below:

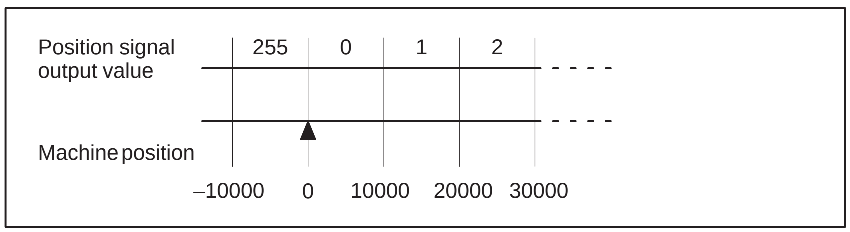
$$\frac{24 \times 10^6}{60} \times 0.016 = 6{,}400$$

This value is based on the output condition of the CNC. If the signal monitoring period of the PMC is long, the grid interval must be increased accordingly. This function can be used to turn a control signal on or off at a particular position during movement of the machine.

Example of parameter setting

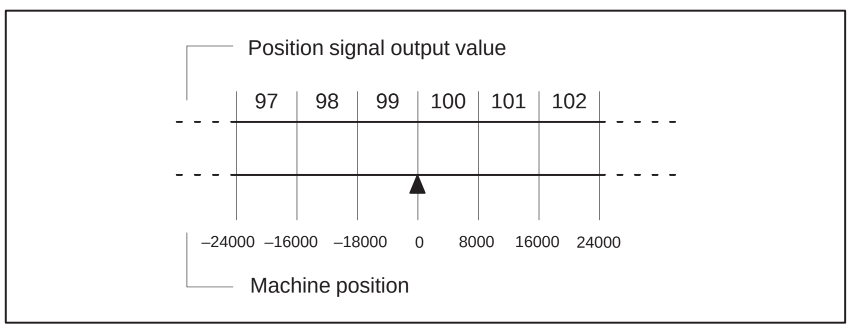
(i) Grid width = 10,000

Grid number at the reference position = 0



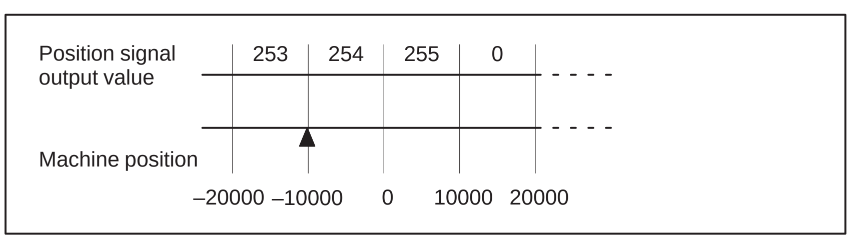
(ii) Grid width = 8,000

Grid number at the reference position = 100



(iii) Grid width = 10,000

Grid number at the reference position = 255



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0

Parameter

0113	X–axis grid number of position signal output
0114	Z–axis grid number of position signal output

[Data type] byte

[Valid data range] 0 to 255

[Operation] Specify the grid number of the axial reference position for the position signal output function.

0713	X–axis grid width of position signal output
0714	Z–axis grid width of position signal output

[Data type] 2–word

[Unit of data]

Setting value	IS–A	IS–B	IS–C
Millimeter machine [mm]	0.01	0.001	0.0001
Inch machine [inch]	0.001	0.0001	0.00001

[Valid data range] 0 to 99999999

[Operation] Specify the grid width of each axis.

> **NOTE**
> When zero is specified, position signal output is not performed.

1.10 Cf AXIS CONTROL (T SERIES)

General

This function controls the third axis connected to a servo motor as an NC axis.

Specification

As the basic specifications of the Cf axis are the same as those for the first and second axes, this section focuses on the differences.

Attribute: This function can be set for linear and rotation axes, using parameter LIN3 (bit 2 of parameter No.0032). Which axis of the basic coordinate system becomes the Cf axis is specified using parameter No.0279.

Command address:	Absolute command	C (*)
	Incremental command	H

> **NOTE**
> (*) It is necessary to set parameter ADW (bit 0 of parameter No.0030) to 1. It is also necessary to set parameter No. ADRC (bit 4 of parameter No.0029) to 1 if the chamfering corner R and direct drawing dimension input functions are used.

Signal

If this function is unavailable, bit 2 of G118 is GR1, bit 2 is GR2, and bit 7 is DRN. If this function is available, bit 2 of G118 is changed to +3, bit 3 is to –3, and bit 7 is to H3. So, when using GR1, GR2, or DRN, set parameter ADDCF (bit 5 of parameter No.0031) to 1, so that these signals are assigned to G123.

C–axis–off signal COFF<G123#0>

[Classification] Input signal

[Function] Informs the NC that the third axis is connected as the spindle.

[Operation] When the signal becomes 1, the relative coordinates are cleared to 0. If a reference position return has been completed, the reference–position–return–completed signal (ZP3, bit 2 of F148) for the third axis becomes 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G123								COFF	(T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0029				ADRC				DSP3

DSP3 Specifies whether to display the current position of the third axis, as follows:
1 : Display
0 : Do not display

ADRC Specifies the address of chamfering corner R as follows:
1 : I/K rather than C is used. C/R for the address for direct drawing dimension input is prefixed with a comma as follows: ,C/,R
0 : C/R is used. C/R is used also for direct drawing dimension input.

	#7	#6	#5	#4	#3	#2	#1	#0
0030								ADW30

ADW30 Specifies the name of the third axis as follows:
1 : C
0 : B

	#7	#6	#5	#4	#3	#2	#1	#0
0031	CNRST	ESFC	ADDCF					

ADDCF Specifies the PMC address for signals GR1, GR2, and DRN as follows:
1 : G123#2 GR1, G123#3 GR2, G123#7 DRN
0 : G118#2 GR1, G118#3 GR2, G118#7 DRN

ESFC Specifies whether to enable the feedback pulse from the position detector during the turning mode in Cf axis control.
1 : Enable
0 : Disable

CNRST Specifies whether to clear the Cf axis relative coordinates at a reference position return.
1 : Clear
0 : Do not clear

	#7	#6	#5	#4	#3	#2	#1	#0
0032	ROT10					LIN3		

LIN3 Specifies whether the third axis is a linear or rotation axis, as follows:
1 : Linear axis
0 : Rotation axis

ROT10 Specifies the measurement unit of parameters for the upper limit to the cutting feedrate (parameter No.0527), rapid traverse F0 rate (parameter No.0533), and the FL rate for a reference position return (parameter No.0534) during inch output, as follows:
1 : 1 degrees/min
0 : 0.1 degrees/min

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.

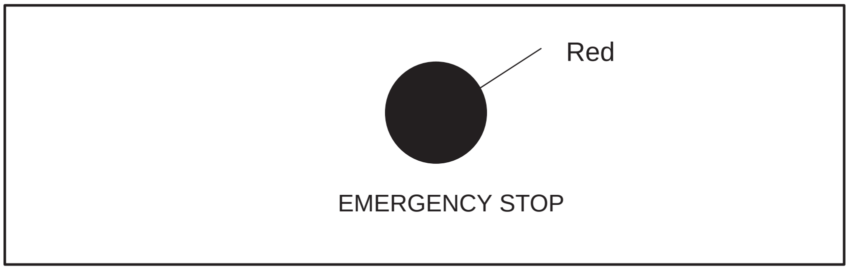


Fig. 2.1 (a) EMERGENCY STOP

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<X021#4,G121#4>

[Classification] Input signal

[Function] Outputting 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 software limit 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) below).

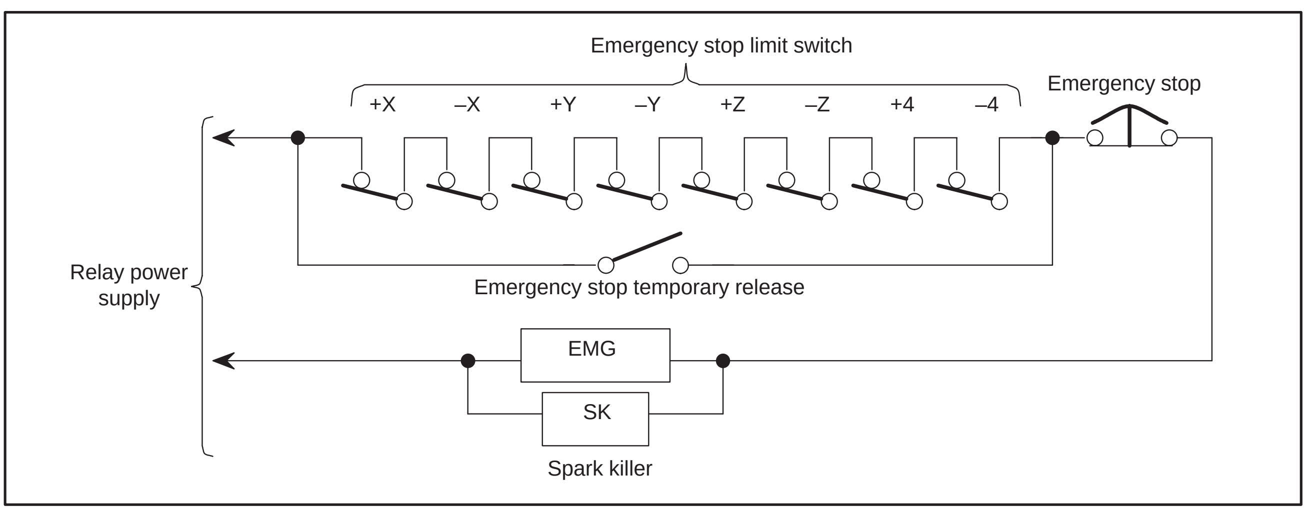


Fig. 2.1 (b) Connection of Emergency Stop Limit Switch

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 Descriptions.”

WARNING

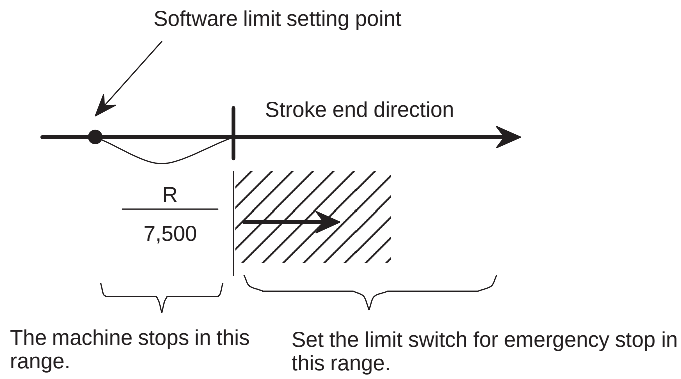
Software limit setting point and operating point of limit switch for emergency stop

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.0700 – 0703 and 0704 – 0707) 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
X021				*ESP				
G121				*ESP				

Reference item

AC SERVO MOTOR series DESCRIPTIONS	B–65002E
AC SERVO MOTOR α series DESCRIPTIONS	B–65142E

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<F149#7>

[Classification] Output signal

[Function] The CNC ready signal reports 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 <F148#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 sent and apply the brake when this signal is not sent.
Time chart of this signal is as follows:

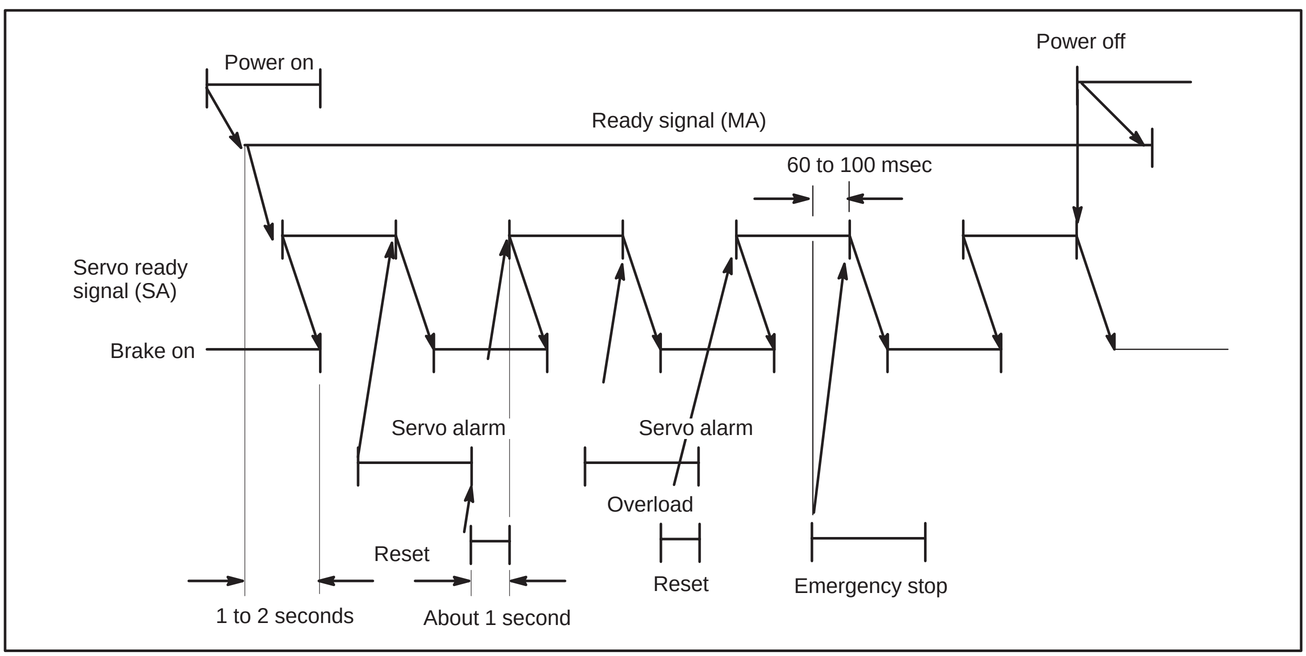


Fig.2.2 Time Chart for Servo Ready Signal

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F148		SA						
F149	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 because of working the limit switch and an OVER TRAVEL is displayed.

Signal

Overtravel signal *+LX to *–LZ <X020#0 to #5>(M) *+LZ<X018#5>(T)

[Classification] Input signal

[Function] Indicates that the control axis has reached its stroke limit. There are signals for every direction in every control axis. The +/– in the signal name indicates the direction and the alphabet corresponds to the control axis.

+ L x

X X axis is at stroke limit.
Y Y axis is at stroke limit.
Z Z axis is at stroke limit.
: :

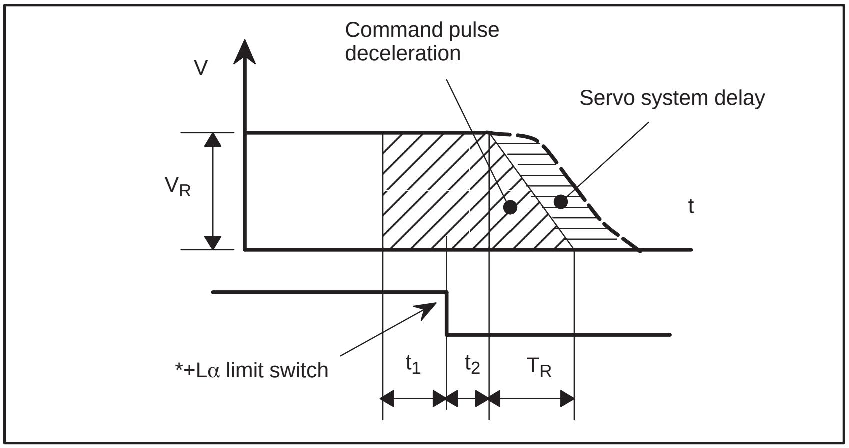
+ Has moved to and reached a + direction.
– Has moved to and reached a – direction.

[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 movement 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))

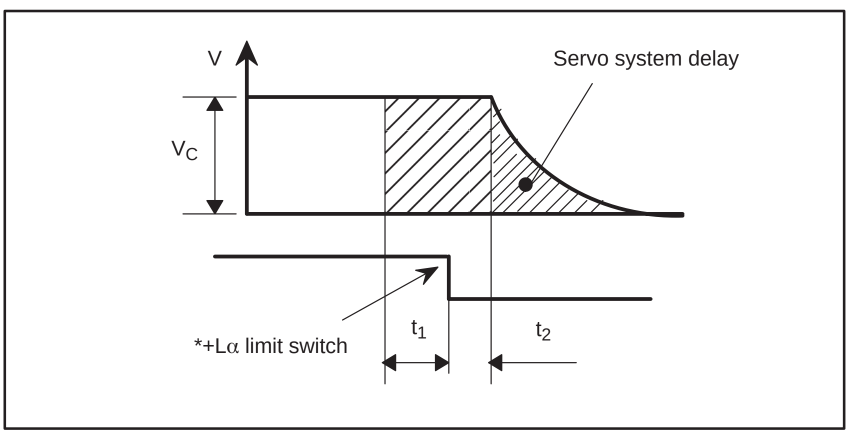
t_2: Receiver delay time 30ms

T_R: Rapid traverse acceleration/deceleration time constant (ms)

T_S: Servo system time constant (ms)

NOTE
Servo system time constant T_S 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**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
X018			*+LZ						(T series)
X020			*–LZ	*–LY	*–LX	*+LZ	*+LY	*+LX	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0015						COTZ			(T series)

[Data type] Bit

COTZ The overtravel limit signal is:
0 : Checked
1 : Not checked

> **WARNING**
> For safety, usually set 0 to check the overtravel signal.

	#7	#6	#5	#4	#3	#2	#1	#0	
0057			HOT3						(M series)

HOT3 The overtravel limit signal is:
0 : Not checked
1 : Checked

> **WARNING**
> For safety, usually set 1 to check the overtravel signal.

Alarm and message

M series

Number	Message	Description
5n4	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 4) + side hardware OT.
5n5	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 4) – side hardware OT.

T series

Number	Message	Description
520	OVER TRAVEL : +Z axis	Exceeded the Z axis + side hardware OT.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.6.2	OVERTRAVEL
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.6.2	OVERTRAVEL

2.3.2 Stored Stroke Check 1

General

When the tool exceeds a stored stroke check, 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 can be moved in the reverse direction from which the tool came.

Parameters (Nos.0700 – 0707 or 0743 – 0750 (M)/0770 – 0777 (T)) set boundary. Outside the area of the set checks is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

Signal

Stored stroke check select signal EXLM2 <G129#6>

[Classification] Input signal

[Function] Selects stroke check 1 (parameter Nos.0700 to 0707) or stroke check 2 (parameter Nos.0743 to 0750 (M)/0770 to 0777 (T)).

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

– Checks stroke check 1 on the basis of parameter Nos.0743 to 0750 (M)/0770 to 0777 (T), instead of parameter Nos.0700 to 0707.

Stroke check external setting signals +LMX to –LMZ <G129#0 to #5> (M series)

[Classification] Input signal

[Function] Change the values of the parameters governing the stroke check (0700 to 0707).

[Operation] When these signals are set to 1, the control unit operates as follows:
– Change the stored checks, set with parameter Nos. 0700 to 0707, to the machine coordinates when the signals are input.

Stroke check release signal RLSOT <G129#7> (M series)

[Classification] Input signal

[Function] Selects whether the software limit check are checked.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Does not check the sofware limit check.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G129		EXLM2							(T series)
G129	RLSOT	EXLM2	–LMZ	–LMY	–LMX	+LMZ	+LMY	+LMZ	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0015				LM2					(T series)
0020				LM2					(M series)

[Data type] Bit

LM2 The EXLM2 signal for switching stored stroke check 1 and 2
0: Disabled
1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
0065					PSOT			

[Data type] Bit

PSOT Checking of stored stroke limit during the time from power–on to the manual position reference return
0: The stroke limit is checked.
1: The stroke limit is not checked

	#7	#6	#5	#4	#3	#2	#1	#0
0076	OTRFOM							

[Data type] Bit

OTRFOM When a command that exceeds a stored stroke check is issued
0: An alarm is generated after the stroke check is exceeded.
1: An alarm is generated before the stroke check is exceeded.

0700 – 0703	Coordinate value I of stored stroke check 1 in the positive direction on each axis
0704 – 0707	Coordinate value I of stored stroke check 1 in the negative direction on each axis

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

The coordinate values of stored stroke checks 1 in the positive and negative directions are set for each axis in the machine coordinate system. The outside area of the two checks set in the parameters is inhibited.

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:

parameters 0700 – 0703 < parameters 0704 – 0707

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.

0743 – 0746	Coordinate value II of stored stroke check 1 in the positive direction on each axis	(T series)
0747 – 0750	Coordinate value II of stored stroke check 1 in the negative direction on each axis	(T series)
0770 – 0773	Coordinate value II of stored stroke check 1 in the positive direction on each axis	(M series)
0774 – 0777	Coordinate value II of stored stroke check 1 in the negative direction on each axis	(M series)

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set 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 EXLM2 is ON, stroke checks are checked with parameters 0700 to 0707, not with parameters 1320 and 1321. The area outside that set by parameters 1326 and 1327 is inhibited.

NOTE

The EXLM2 signal is enabled only when parameter LM2 (No.0020#4 (M)/No.0015#4 (T)), is set to 1.

Alarm and message

Number	Message	Description
5n0	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 4) + side stored stroke check I.
5n1	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 4) – side stored stroke check I.

Caution

CAUTION
In setting a forbidden area, if two points to be set are the same, all area is forbidden in check 1.

Note

NOTE
1 Parameter PSOT (bit 6 of No.0065) 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 For the 0–TTC, set a forbidden area for each path.
3 Parameter OTRFOM (bit 7 of No.0076) 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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.6.3	STROKE CHECK
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.6.3	STROKE CHECK

2.3.3 Stored Stroke check 3, 4

General

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 3,and stored stroke check 4. (Stored stroke check 4 is performed in T series only.)

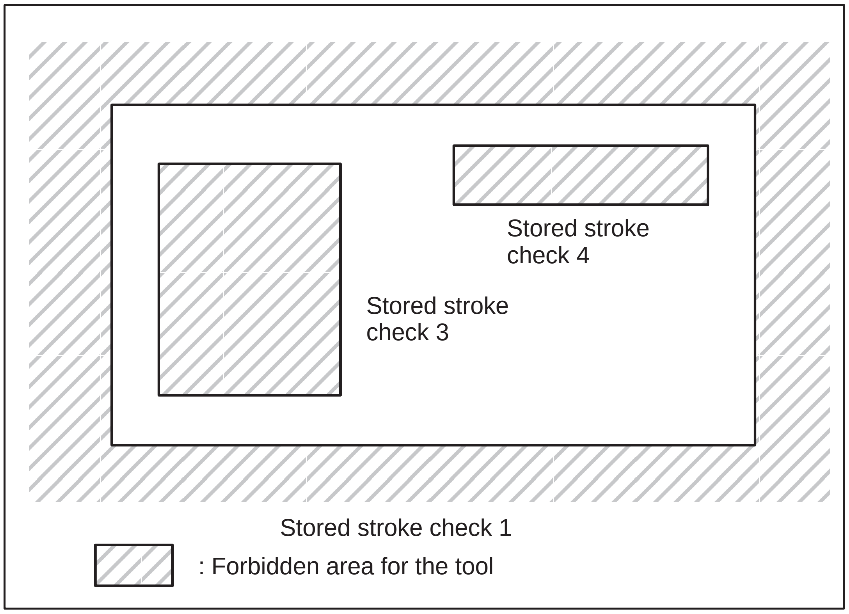


Fig. 2.3.3 (a) Stroke check (T series)

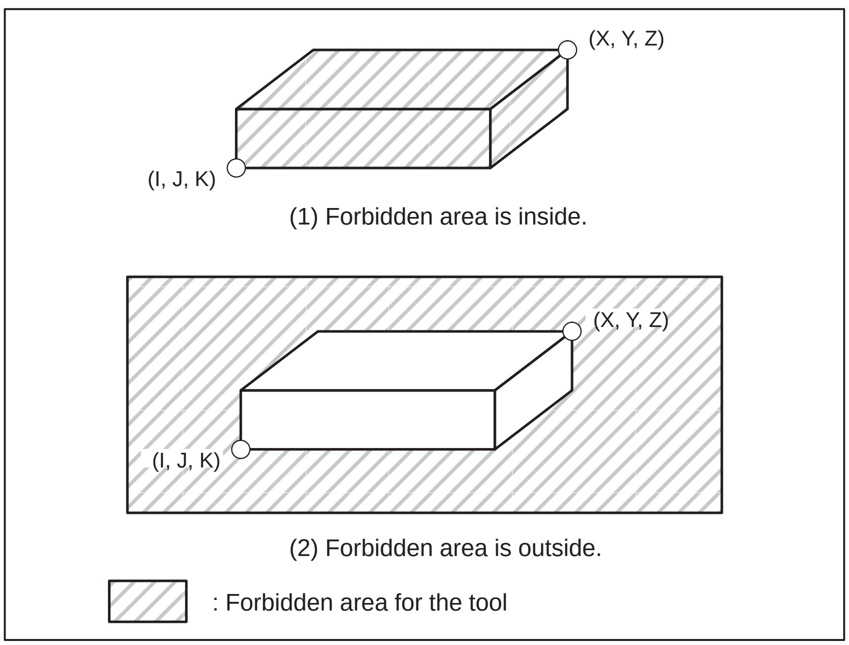


Fig. 2.3.3 (b) Stroke check (M series)

When the tool exceeds a stored stroke check, 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 can be moved in the reverse direction from which the tool came.

Stored stroke check 3

Parameters (Nos. 0804–0809 (M)/0747–0754 (T)) or commands set these boundaries. Inside or outside the area of the check can be set as the forbidden area. Parameter INOUT (No. 0024#4) selects either inside or outside as the forbidden area.
In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

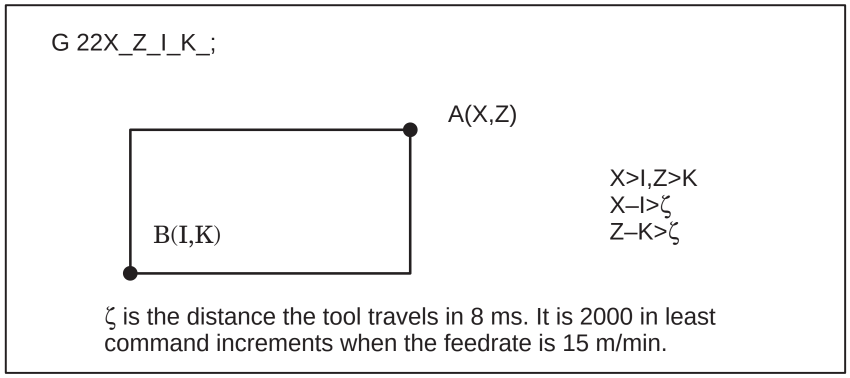


Fig. 2.3.3 (c) Creating or changing the forbidden area using a program (T series)

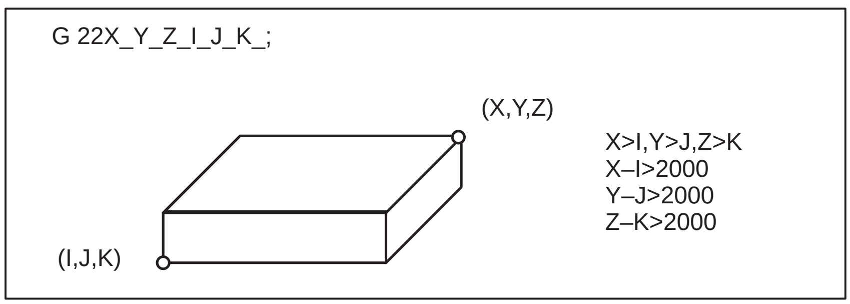


Fig. 2.3.3 (d) Creating or changing the forbidden area using a program (M series)

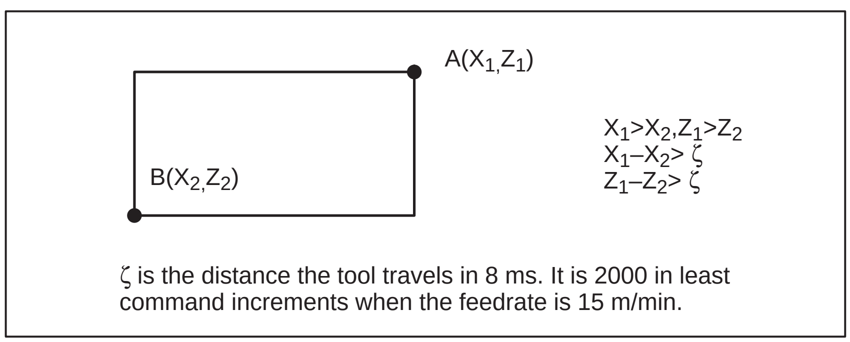


Fig. 2.3.3 (e) Creating or changing the forbidden area using a parameters (T series)

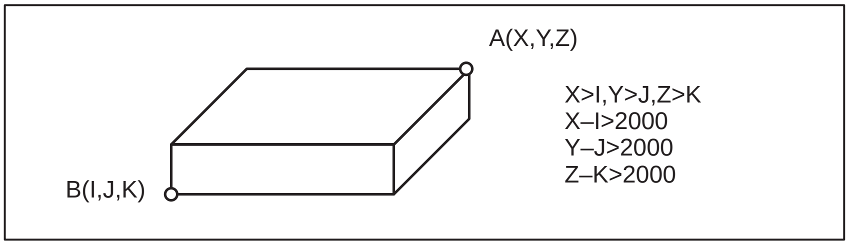


Fig. 2.3.3 (f) Creating or changing the forbidden area using a parameters (M series)

When you set the forbidden area X_1,Z_1,X_2,and Z_2 through parameters Nos.0804 to 0809 (M)/Nos.0747 to 0754 (T), the data should be specified by the distance from the reference position in the least command increment (output increment).
If set the forbidden area XZIK by a G22 command, specify the data by the distance from the reference position in 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 4**

Set the boundary with parameters Nos. 0760 to 0767. The area inside the boundary becomes the forbidden area. (T series)

- **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 top 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.

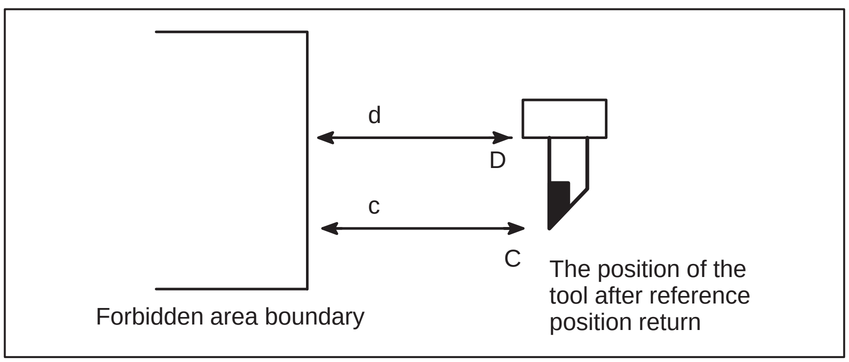


Fig. 2.3.3 (g) Setting the forbidden area (T series)

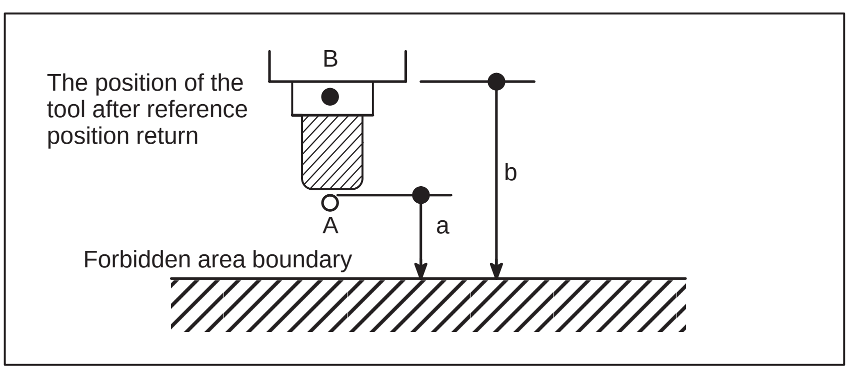


Fig. 2.3.3 (h) Setting the forbidden area (M series)

- **Forbidden area over–lapping**

Area can be set in piles.

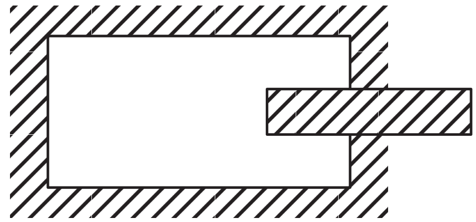


Fig. 2.3.3 (i) Setting the forbidden area overlapping (T series)

Unnecessary checks should be set beyond the machine stroke.

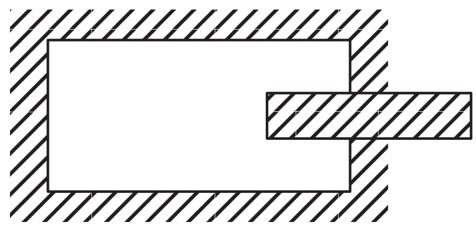


Fig. 2.3.3 (j) Setting the forbidden area overlapping (M series)

- **Effective time for a forbidden area**

Parameter PSOT (bit 3 of No. 0065) 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 3).

- **Releasing the alarms**

When the tool has become unmovable in the forbidden area, push the emergency stop button to release the forbidden condition and move the tool out of the forbidden area in the G23 mode; then, if the setting is wrong, correct it and perform the reference position return again.

- **Change from G23 to G22 in a forbidden area**

When G23 is switched to G22 in the forbidden area, the following results.
(1)When the forbidden area is inside, an alarm is informed in the next move.
(2)When the forbidden area is outside, an alarm is informed immediately.

- **Creating the forbidden area for the 0–TTC**

For the 0–TTC, set a forbidden area for each tool post.

Parameter

		#7	#6	#5	#4	#3	#2	#1	#0
0024					INOUT				

[Data type] Bit

INOUT The area inside or outside of the third stored stroke check is set as an inhibition area.
0: Inside
1: Outside

		#7	#6	#5	#4	#3	#2	#1	#0
0065						PSOT			

[Data type] Bit

PSOT Checking of stored stroke limit during the time from power–on to the manual position reference return
0: The stroke limit is checked.
1: The stroke limit is not checked

		#7	#6	#5	#4	#3	#2	#1	#0
0076		OTRFOM							

[Data type] Bit

OTRFOM When a command that exceeds a stored stroke limit is issued
0: An alarm is generated after the stroke limit is exceeded.
1: An alarm is generated before the stroke limit is exceeded.

0747–0750	Coordinate value of stored stroke check 3 in the positive direction on each axis	(T series)
0751–0754	Coordinate value of stored stroke check 3 in the negative direction on each axis	(T series)
0804–0806	Coordinate value of stored stroke check 3 in the positive direction on each axis	(M series)
0807–0809	Coordinate value of stored stroke check 3 in the negative direction on each axis	(M series)

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke checks 2 in the positive and negative directions for each axis in the machine coordinate system. INOUT, #4 of parameter 0024, 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.

0760–0763	Coordinate value of stored stored check 4 in the positive direction on each axis
0764–0767	Coordinate value of stored stroke check 4 in the negatice direction on each axis

[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
Rotation axis	0.01	0.001	0.0001	deg

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke checks 3 in the positive and negative directions for each axis in the machine coordinate system. The area inside the checks set in the parameters is forbidden.

Alarm and message

Number	Message	Description
5n2	OVER TRAVEL : +n	Exceeded the n–th axis + side stored stroke check 3. (Parameter Nos. 0747–750 (T)/ 0804–0806 (M))
5n3	OVER TRAVEL : –n	Exceeded the n–th axis – side stored stroke check 3. (Parameter Nos. 0751–0754 (T)/ 0807–0809 (M))
5n4	OVER TRAVEL : +n	Exceeded the n–th axis + side stored stroke check 4. (Parameter Nos. 0760–0763 (T))
5n5	OVER TRAVEL : –n	Exceeded the n–th axis – side stored stroke check 4. (Parameter Nos. 0764–0767 (T))

NOTE
Over travel alrams (No. 5n4–5n5) occur, in the T series

Warning

WARNING
In setting a forbidden area, if the two points to be set are the same, the area is as follows:
(1)When the forbidden area is check 1, all areas are forbidden areas.
(2)When the forbidden area is check 3 or check 4, all areas are movable areas.

Note

NOTE
Parameter OTRFOM (bit 7 of No. 0076) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

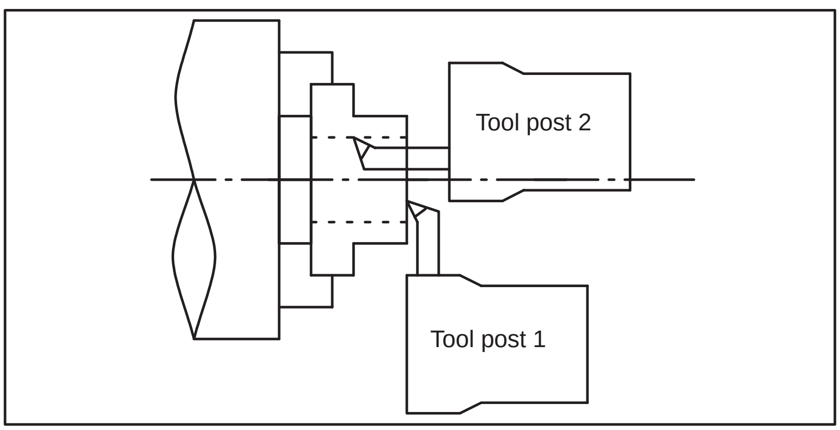
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.6.3	STROKE CHECK
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.6.3	STROKE CHECK

2.3.4 Tool Post Interference Check (0–TTC)

General

When two tool posts machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the two tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur.
The function "tool post interference check" is available which can decelerate and stop the two tool posts before the tool posts interfere with each other due to an incorrect command.



The contours of the two tool posts are checked to determine whether or not an interference occurs.

Signal

Tool post interference check signal BOFF <F180#6>

[Classification] Output signal

[Function] Indicates whether the tool post interference check function is being performed.

[Output condition] This signal goes “0” when:

(i) All requisites for the tool post interference check function are satisfied.

This signal goes “1” when:

(i) Any of the requisites for the tool post interference check function are not satisfied.

> **NOTE**
> For details of the requisites for the tool post interference check function, refer to the operator’s manual for Lathe (B–61394E).

Tool post interference alarm signal TAL <F180#7>

[Classification] Output signal

[Function] Indicates that the tool post interference alarm is activated.

[Output condition] This signal goes “1” when:

(i) The control unit judges that the two tool posts will interfere with each other during the execution of the tool post interference check function.

This signal goes “0” when:

(i) The control unit judges that the two tool posts will not interfere with each other during the execution of the tool post interference function.

(ii) When the tool post interference check function is not being performed.

NOTE

1 During the execution of the interference check function, if the control unit judges that the two tool posts will interfere with each other, it stops both tool posts by slowing them down, and then enters the alarm state. The CNC then sets the TAL signal "1" to indicate that an interference alarm has occurred.

2 If the interference alarm is activated, switch the operation mode to the manual mode, manually withdraw the tool posts to where they do not interfere each other, then release the alarm status by resetting the control unit.
As the result of manually withdrawing the tool posts, the TIALM signal goes "0" when the control unit judges that the tool posts are separated enough not to interfere with each other any more. When manually withdrawing the interfering tool posts, the TIALM signal is effective in identifying how far the tool posts must be separated from each other. This is because it is easy for the operator to check at which point the signal goes "0".

3 If an interference alarm occurs, the axis being moved and its moving direction are stored, and the axis cannot be moved in the stored direction until the alarm is released by resetting the control unit. This prevents the axis from interfering any further by prohibiting movement in the the direction that caused the interference.

Signal address

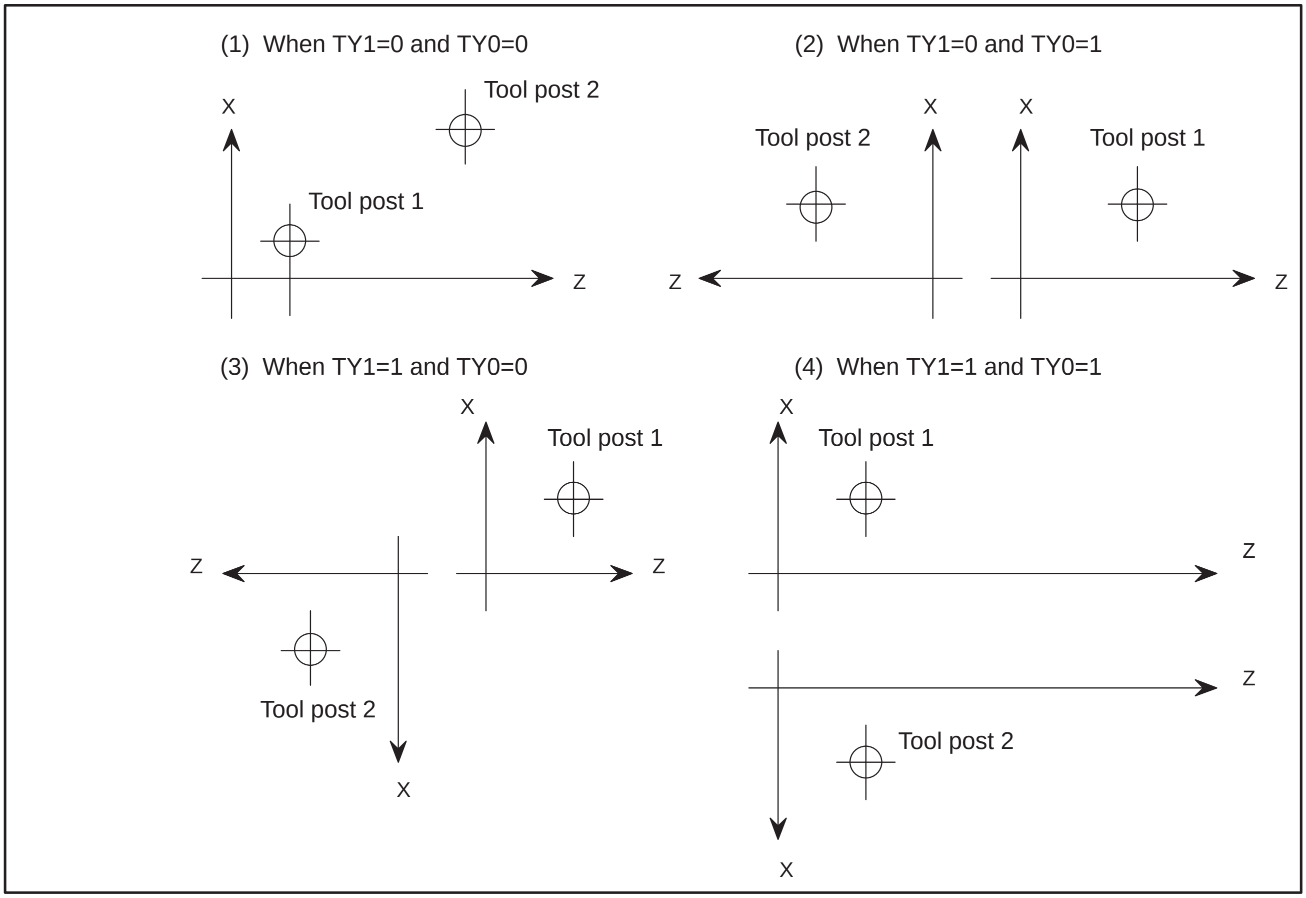
	#7	#6	#5	#4	#3	#2	#1	#0
F180	TAL	BOFF						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0048			ZCLR	IFE	IFM	ITO	TY1	TY0

[Data type] Bit

TY0, TY1 This parameter specifies the relationship between the coordinate systems of the two tool posts.



ITO When offset number 0 is specified by the T code,
- 0: Checking interference between tool posts is stopped until an offset number other than 0 is specified by the next T code.
- 1: Checking interference between tool posts is continued according to the previously specified offset number.

IFM Specifies whether interference between tool posts is checked in the manual operation mode.
- 0: Not checked
- 1: Checked

IFE Specifies whether interference between tool posts is checked.
- 0: Checked
- 1: Not checked

ZCLR Specifies whether interference along the Z axis is checked while checking interference between tool posts.
- 0: Checked
- 1: Not checked (Only interference along the X axis is checked.)

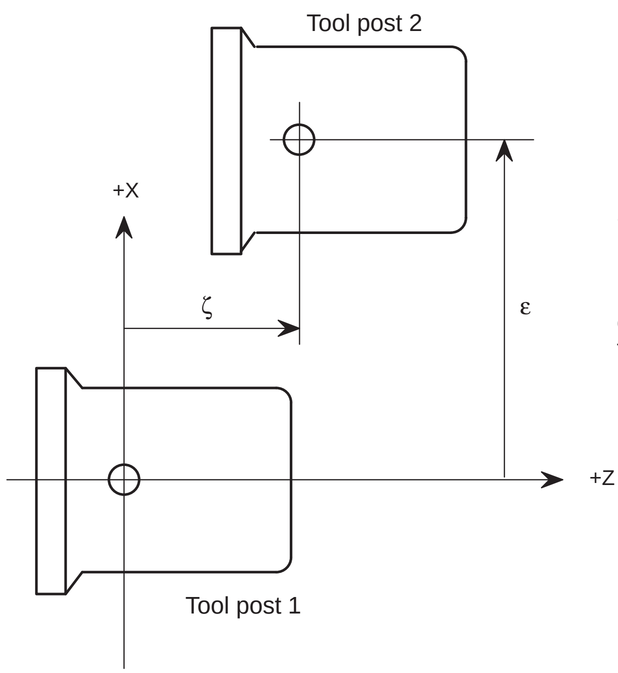
0768	Distance along the X axis between the reference positions of tool posts 1 and 2
0769	Distance along the Z axis between the reference positions of tool posts 1 and 2

[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 X and Z values (ε, ζ) of the reference position of tool post 2 in parameters 0768 and 0769, respectively, in the Z–X plane coordinate system. However, note that the reference position of tool post 1 is plotted at the zero point.

WARNING

After the parameter values are changed, perform manual reference position return for individual tool posts. Otherwise, data on the positional relationship between the tool posts stored in memory will not be updated to the new parameter values.

Alarm and message

Number	Message	Description
169	ILLEGAL TOOL GEOMETRY DATA	Incorrect tool figure data in interference check.
590 592	INTERFERENCE : +X INTERFERENCE : +Z	An interference alarm has generated when X or Z axis is moving in the positive direction.
591 593	INTERFERENCE : – X INTERFERENCE : – Z	An interference alarm has generated when X or Z axis is moving in the negative direction.

Warning

WARNING

1 When an alarm is raised, the CNC system and machine system stop with some delay in time. So an actual stop position can be closer to the other tool post beyond an interference forbidden position specified using tool shape data. So, for safety, tool shape data a little larger than the actual shape should be set. The extra distance, L, required for this purpose is calculated from a rapid traverse feedrate as follows

$$L = (\text{Rapid traverse rate}) \times \frac{1}{7500}$$

For example, when a rapid traverse feedrate of 15 m/min is used, L=2mm.

2 When parameters or tool shape data (contact forbidden area) are set for the interference check function, check that the interference forbidden area is correctly set by moving the tool posts to foul each other in several directions in manual mode (interference check enabled with a parameter).

Reference item

OPERATOR’S MANUAL (For Lathe) (B–61394EN)	II.24.3	TOOL POST INTERFERENCE CHECK

2.4 ALARM SIGNAL

General

When an alarm is triggered in the CNC, the alarm is indicated on the CRT 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<F149#0>

[Classification] Output signal

[Function] The alarm signal reports that the CNC is in an alarm state.

There are the following alarms. The following alarms are 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.

Absolute pulse coder battery alarm signal BAL1 to BAL4, BAL7, BAL8 <F156#0 to F156#5> (T series) <F159#0 to F159#5> (M series)

[Classification] Output signal

[Function] Indicates when the battery voltage for the absolute pulse coder is below the specified value.

[Output condition] The signal is set to 1 when:

– The battery voltage of the absolute pulse coder has fallen to below the specified level.

The signal is set to 0 when:

– The battery voltage of the absolute pulse coder has risen to the specified level or higher.

Battery alarm signal BAL<F149#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	
F149						BAL		AL	
F156			BAL8	BAL7	BAL4	BAL3	BAL2	BAL1	(T series)
F159			BAL8	BAL7	BAL4	BAL3	BAL2	BAL1	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0064			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

This signal disables machine movement along axes. When this signal is input during movement along axes, the tool movement is decelerated, then stopped.

Signal

Start lock signal STLK<G120#1>(T series)

[Classification] Input signal

[Function] This signal disables machine movement along axes in 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, or T 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 (STL is “1”, SPL is “0”). When the STLK signal turns to “0”, operation restarts. (Figs.2.5 (a), (b)).

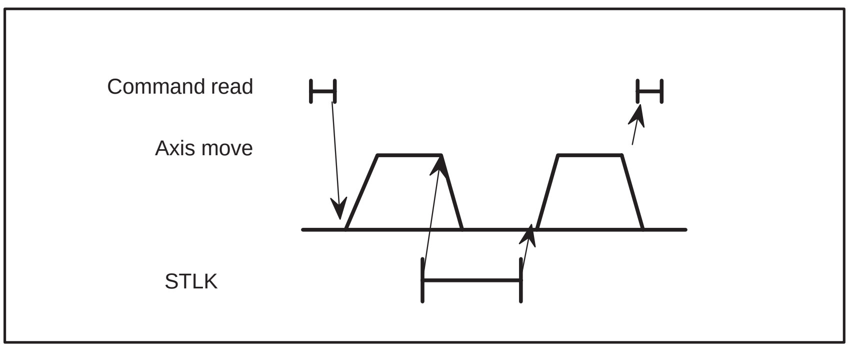


Fig.2.5 (a) Block containing only axis move command

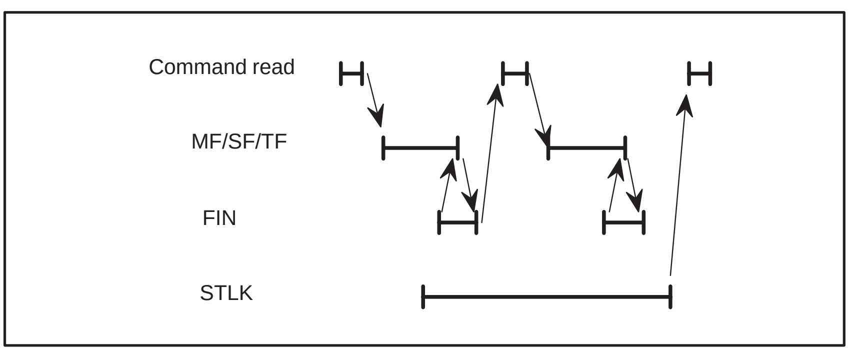


Fig.2.5 (b) Block containing auxiliary functions only

Interlock signal *ILK<G117#0>(M series)

[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 *ILK signal is "0", the axis movement is decelerated and stopped. In automatic operation, blocks containing M, S or T 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 *ILK signal turns to "1", operation resumes (Figs.2.5.2(c), (d)).

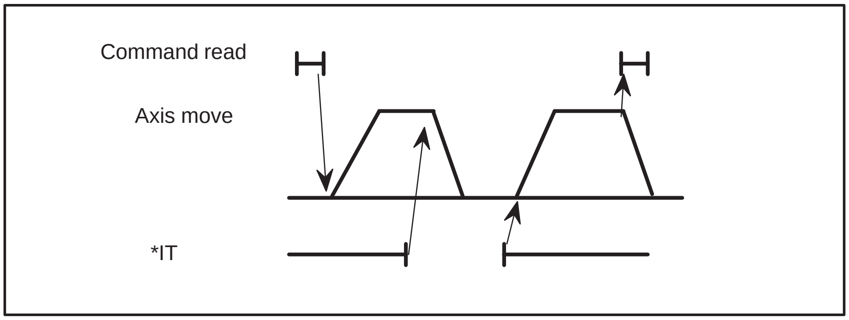


Fig.2.5 (c) Block containing only axis move command (manual and automatic operation)

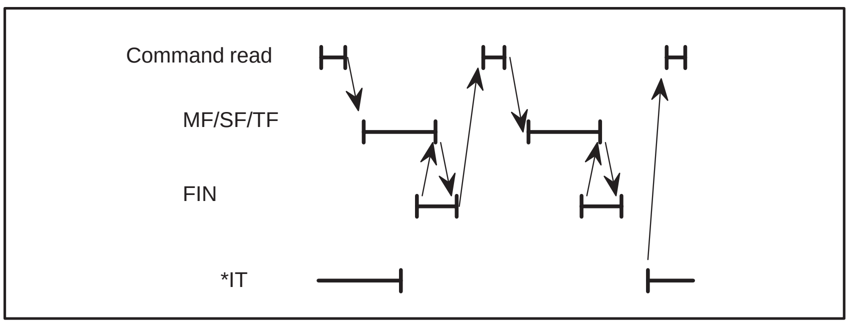


Fig.2.5 (d) Block containing auxiliary functions only (automatic operation)

NOTE

The overtravel amount of the motor after turning *ILK to "0" is represented by the following formula.

$$Q_{max} = F_m \times \frac{1}{60} \times \left(\frac{T_c}{1000} + \frac{T_s}{1000} + \frac{A}{1000} \right)$$

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 at standard interlock signal A = 16ms at high–speed interlock signal

Interlock signal for each axis *ITX to *IT4 <G128#0 to #3>(M series) ITX to IT4 <G128#0 to #3>(T series)

[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 x

X ..... Interlock for the X axis
Y ..... Interlock for the Y axis
Z ..... Interlock for the Z 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 (AUTO 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), all axes movements are prevented.

If a moving axis is interlocked, all axes stop moving after being decelerated, and they start moving again when it is released from being interlocked.

This function is effective during dry run.

Interlock signal for each axis and direction *+MITX, *–MITX, *+MITY, *–MITY, *+MITZ, *–MITZ, *+MIT4, *–MIT4 <G142>(M series) +MIT1, –MIT1, +MIT2, –MIT2 <X008#2 to X008#5> (T series)

[Classification] Input signal

[Function] This function allows a directional interlock for each axis. Together with the high-speed interlock, it is possible to release the interlock of an axis only in the direction in which no axis/directional interlock is applied.

[Operation] When the axis/directional interlock signal becomes “0” for M series or “1” for T series, CNC applies interlock only in the corresponding axial direction. However, during automatic operation, all axes will stop.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
X008			–MIT2	+MIT2	–MIT1	+MIT1			(T series)
G117								*ILK	(M series)
G120							STLK		(T series)
G128					IT4	IT3	ITZ	ITX	(T series)
					*IT4	*ITZ	*ITY	*ITX	(M series)
G142	*–MIT4	*–MITZ	*–MITY	*–MITX	*+MIT4	*+MITZ	*+MITY	*+MITX	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0008	EILK								(T series)

EILK Specifies whether a start lock is applied to each axis separately or in common, as follows:
1 : Separately (ITX, bit 0 of G0128 to IT4, bit 3)
0 : In common (STLK, bit 1 of G0120)

	#7	#6	#5	#4	#3	#2	#1	#0	
0008	EILK								(M series)

EILK Specifies whether an interlock signal is applied to each axis separately or in common, as follows:
1 : Separately
0 : All axes in common or Z–axis alone (Either is selected according to the setting of the ZILK bit (bit 1 of parameter 0012).)

	#7	#6	#5	#4	#3	#2	#1	#0	
0012							ZILK		(M series)

ZILK Specifies whether an interlock is applied to only the Z–axis or all axes in common, as follows:
1 : Z–axis only
0 : All axes in common

	#7	#6	#5	#4	#3	#2	#1	#0	
0015						RILK			(M series)

RILK Specifies whether to enable the high–speed interlock signal (*RILK, bit 5 of X008), as follows:
1 : Enable
0 : Disable

	#7	#6	#5	#4	#3	#2	#1	#0	
0024	EDILK								(T series)

EDILK Specifies whether to enable axial interlock signals (+MIT1, bit 2 of X008 to –MIT2, bit 5), as follows:
1 : Enable
0 : Disable

	#7	#6	#5	#4	#3	#2	#1	#0	
0049							RDIK	DILK	(M series)

DILK Specifies whether to enable axial interlock signals (*+MIT, bit 0 of G142 to *–MIT, bit 7), as follows:
1 : Enable
0 : Disable

RDIK Specifies when the high–speed interlock signal (*RILK, bit 5 of X008) is to be enabled, as follows:
1 : Disable when the axial interlock signals (*+MIT, bit 0 of G142 to *–MIT, bit 7)
0 : Always enable

Note

NOTE
The interlock signal for each axis and direction (T series) is supported regardless of whether the tool compensation measurement value direct input B function 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 seven modes -- memory edit (EDIT), memory operation (MEM), manual data input (MDI), manual handle/incremental feed (HANDLE/STEP), manual continuous feed (JOG), TEACH IN JOG, and in addition, DNC operation mode can be selected by combining the (AUTO) 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 MD1, MD2, MD4 <G122#0 to #2> DNCI<G127#5> ZRN<G120#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 than 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.
“Faulty mode switching” means for example:

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 mode switching, only MD2 should change from 0 to 1. If a transient signal status change occurs in a signal other than MD2 during mode switching, however, another mode (manual continuous feed mode, for example) is 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 intends to switch the mode to the memory edit mode, the CNC is, instead, placed in the feed hold state.

	Mode	Signal status				
		MD4	MD2	MD1	DNCI	ZRN
1	Memory edit (EDIT)	0	1	1	0	0
2	Memory operation (AUTO)	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	Jog feed (JOG)	1	0	1	0	0
6	TEACH IN HANDLE	1	1	1	0	0
7	TEACH IN JOG	1	1	0	0	0
8	DNC operation	0	0	1	1	0
9	Manual reference position return (ZRN)	1	0	1	0	1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G120	ZRN							
G122						MD4	MD2	MD1
G127			DNCI					

Note

NOTE

Precautions on modes and mode switching

1 In the MDI mode, the STL signal turns to "0" and the CNC stops at the end of execution of the commands input from the CRT/MDI panel, but the SPL signal does not turn to "1". Therefore, another command can be input from the manual data input unit under this state.

2 When the manual handle function is not provided, incremental feed is enabled in the HANDLE/INC mode. When the manual handle function is provided, only manual handle feed is enabled in that mode.

3 Manual operation in TEACH IN JOG and TEACH IN HANDLE mode.

a) When parameter JHD no.0013#0 is set to "0" so that jog operation and handle feed operation are performed with separate modes:
In TEACH IN JOG mode, jog operation can be done.
In TEACH IN HANDLE mode, handle feed can be done when optional manual handle feed function is provided, and incremental feed can be done when handle feed function is not provided.

b) When parameter JHD no.0013#0 is set to "1" so that jog operation and handle feed operation are performed with the same mode:
In TEACH IN JOG mode, handle feed and jog feed can be done when optional manual handle feed function is provided, but jog feed only when it is not included.
In TEACH IN HANDLE mode, handle feed and jog feed can be done when optional manual handle feed function is provided, but incremental feed only when it is not included.
The program can be edited in both TEACH IN JOG and TEACH IN HANDLE modes.

4 When the CNC is operating in the MEM mode and is switched to MDI, the unit automatically switches to single block operation, then enters the MDI mode at the end of the executing block. The STL signal turns to "0" at this time, but the SPL signal does not turn to "1" (see Fig.2.6 (a)).

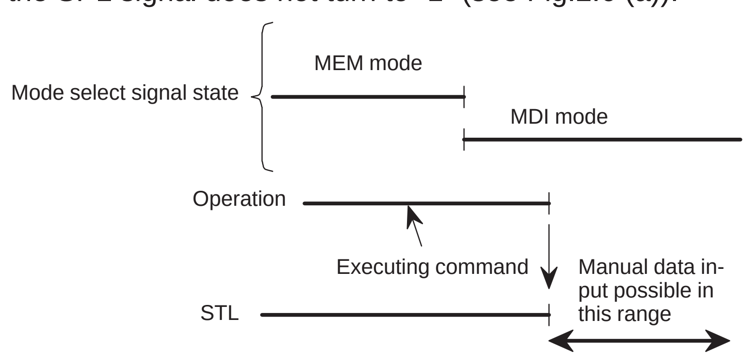


Fig.2.6 (a)

NOTE

5 When the CNC is operating in the MDI mode and is switched to MEM mode, the executing command is finished before the CNC switches to the MEM mode (Fig. 2.6 (b)).

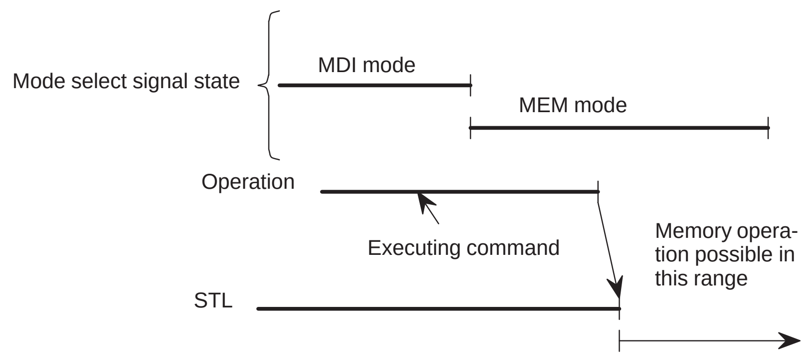


Fig.2.6 (b)

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. Manual handle feed or incremental feed by axis direction select signal is possible under this state. Since the MEM mode or MDI mode commands are held, operation can be restarted by the cycle start signal 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 automatic operation, it can be restarted only by the mode in use before the operation was stopped (Fig.2.6 (c)).

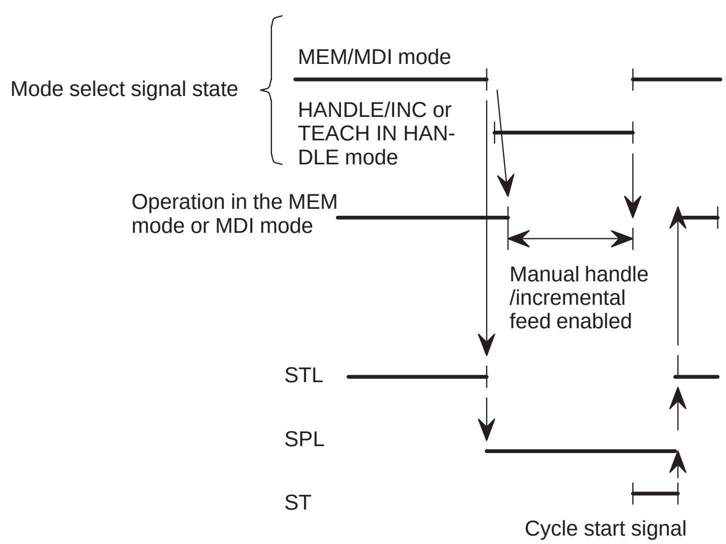


Fig.2.6 (c)

NOTE

7 When the JOG or TEACH IN JOG mode is selected during 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. Manual feed by feed axis direction select signal is possible under this state. Operation can be restarted by returning to the original state, as described for HANDLE/STEP or TEACH IN HANDLE mode . When the mode is switched to the JOG or TEACH IN JOG mode during manual handle feed or 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”. It is possible to perform handle feed in TEACH IN JOG mode by parameter TJHD no.0002#6. For details, refer to item (3) (Fig.2.6 (d), (e)).

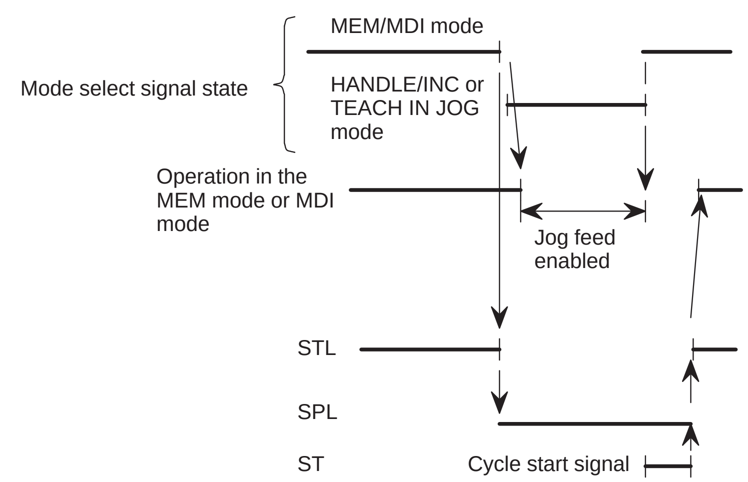


Fig.2.6 (d)

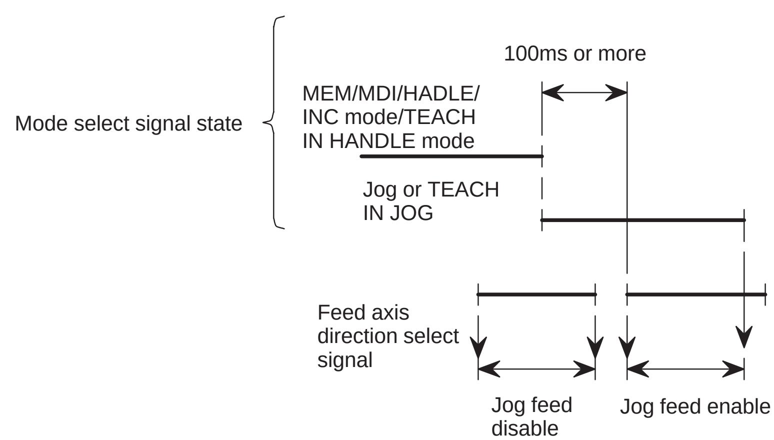


Fig.2.6 (e)

NOTE

8 The mode switching operation is summarized in the time chart below (Fig.2.6 (f)).

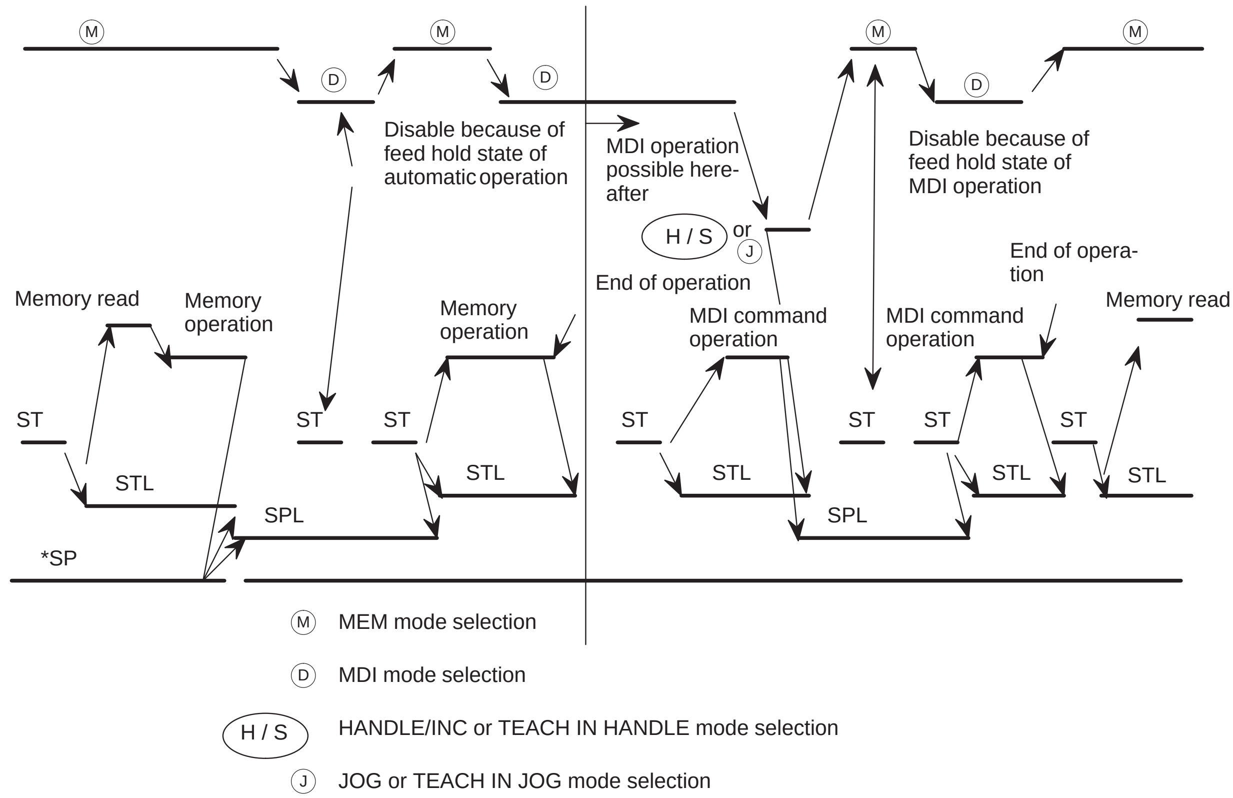


Fig.2.6 (f) Mode signal time chart

Reference item

CONNECTION MANUAL (This manual)	4.1	MANUAL REFERENCE POSITION RETURN

2.7 TOOL POST SELECTION (0–TTC)

General

Either tool post is selected for the display and setting of tool compensation data and other data, program input in MDI mode, the editing of a machining program in program memory, and other operations.

Signal

Tool post selection signal TRT2<X018#3> TRT2PC<G133#0>

[Classification] Input signal

[Function] The 0–TTC has a single CRT/MDI unit. This signal specifies whether the CRT/MDI unit is used for tool post 1 or 2.
Tool post select signal TRT2 is a DI signal that is directly read by the CNC at power–up only. Another tool post select signal, TRT2PC, is a PMC output signal that is constantly monitored after power–up. The TRT2 signal is used for clearing memory or performing other special maintenance–related operations at power–up.

[Operation] 1: The CRT/MDI panel is used to operate tool post 2.
0: The CRT/MDI panel is used to operate tool post 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X018					TRT2			
G133								TRT2PC

NOTE
1 The following data for tool posts 1 and 2 is output on a single screen:
– Position indicated by triple–size letters
– Alarm
– Operator message
2 The following operation or function is enabled for a selected tool post only:
– Input/output via the reader/punch interface
– External key input
– Tool compensation measurement value direct input B function
3 The following operation at power–up can be performed only for the tool post selected with the TRT2 signal.
– Program clear
4 Memory all clear can always be performed for both tool posts, irrespective of the setting of the TRT2 signal.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0046						IGANL		RSTSW

RSTSW The 0–TTC uses the reset key:

1: Only for the selected tool post.

2: For both tool posts.

IGNAL If an alarm is issued for a tool post of the 0–TTC, the other tool post:

1: Does not enter the feed hold status.

0: Enters the feed hold status.

	#7	#6	#5	#4	#3	#2	#1	#0
0047								TIST

TIST For the 0–TTC, tool post select signals TRT2 (X0018, #3) and TRT2PC (G0133, #0) are:

1: Disabled.

0: Enabled.

2.8 STATUS OUTPUT SIGNAL

General

The table below lists the status output signals for notifying 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	11.7
Moving signal	MVX – MV4	1.2.5
In–position signals	INPX – INP4	7.2.6.1
Cutting feed signal	CUT	2.8 (the section you are reading)
Thread cutting signal	THRD	6.4.1
Canned cycle start signal	FXST	2.8 (the section you are reading)

Signal

Cutting feed signal CUT<F188#6>

[Classification] Output signal

[Function] Notifies 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 cutting, thread cutting, skip cutting, or canned cycle)

CAUTION
This signal is not output in the feed hold state.

NOTE
This signal is output even when the feedrate override is 0%, or during interlock.

Canned cycle start signal FXST<F161#4>

[Classification] Output signal

[Function] This signal is kept to be 1 for 200 ms after the beginning of drilling axis movement, to indicate when a canned cycle such as G73, G74, G76, or G80 to G89 (including a rigid tapping cycle) begins.

[Output condition] This signal becomes 1 when:

- Drilling axis movement begins in a canned cycle such as G73, G74, G76, or G80 to G89 (including a rigid tapping cycle); when drilling axis movement begins at the R point, signal output is treated as one block, so be careful about single–block operation.

The signal becomes 0 when:

- 200 ms elapses after the signal becomes 1.
- A reset occurs, or an emergency stop signal is input.

> **NOTE**
> 1 If drilling axis movement ends in the 200 ms period during which the signal is 1, the signal will not be reset to 0.
> 2 If feed hold occurs in the 200 ms period during which the signal is 1, the signal will not be reset to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F161				FXST				
F188		CUT						

2.9 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<G123#0>(M) IGNVRY<G127#0>(T)

[Classification] Input signal

[Function] Disables the detection of servo alarm No.401 or 403, 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 or 403, VRDY OFF, even when the servo amplifier ready signal goes off.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G123								IGNVRY	(M series)
G127								IGNVRY	(T series)

Alarm and message

Number	Message	Description
401	SERVO ALARM: 1, 2TH AXIS VRDY OFF	1–axis, 2–axis servo amplifier READY signal (DRDY) went off.
403	SERVO ALARM: 3, 4TH AXIS VRDY OFF	3–axis, 4–axis 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 or 403 occurs, the control unit detects the alarm.

3 MANUAL OPERATION

3.1 JOG FEED/ INCREMENTAL FEED

General

- **Jog feed**

In the jog mode, turning a feed axis and direction selection signal to “1” on the machine operator’s panel continuously moves the tool along the selected axis in the selected direction.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter S3JOG (No.0049#4).

- **Incremental feed**

In the incremental feed mode, turning a feed axis and direction selection signal 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. Each step can be 10, 100, or 1000 times the least input increment.

The jog feedrate can be adjusted with the override signal.
With the rapid traverse selection signal the tool can be moved at the rapid traverse rate regardless of the override signal.

Signal

The following signals determine the way in which 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</td><td>MD1, MD2, MD4</td></tr>
<tr><td>Selection of the axis to move</td><td rowspan="3">+X, –X, +Y, –Y, +Z, –Z, . . .</td><td rowspan="2"></td></tr>
<tr><td>Selection of the direction to move the axis</td></tr>
<tr><td>Selection of the move amount</td><td>MP1, MP2</td></tr>
<tr><td>Selection of feedrate</td><td colspan="2">*OV1 – *OV8, 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”: +X, –X, +Y, –Y, +Z, –Z, 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 signal MP1 an d MP2.
For the signals selecting the mode, see Section 2.6, “Mode Selection Signals.” For the manual handle feed selection signals, MP1 and MP2 of selection of the move amount, see 3.2 “Manual handle feed.” For rapid traverse override signals ROV1 and ROV2, see Subsec. 7.1.7.1, “Feedrate Override Signals.”
Other signals are described below.

Feed Axis and Direction Selection Signal +X to +4 <G116#2 to G119#2> –X to –4 <G116#3 to G119#3>

[Classification] Input signal

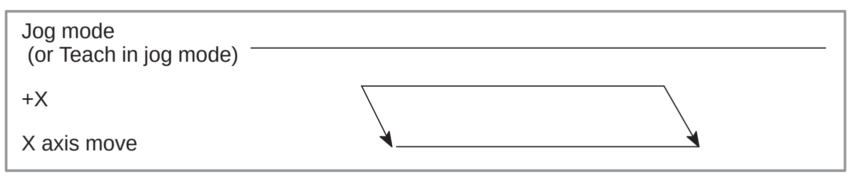
[Function] Selects a desired feed axis and direction in jog feed or incremental feed. The sign (+ or –) in the signal name indicates the feed direction. The number or alphabet indicates the number of the control axis.

+ x

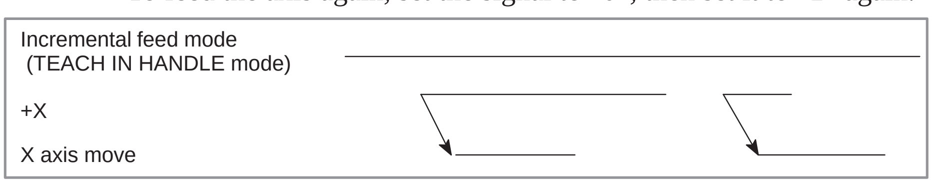
- x:
  - X X axis is fed
  - Y Y axis is fed
  - Z Z axis is fed
  - : :
  - 4 Fourth axis is fed
- +:
  - + Feed in positive direction
  - – Feed in negative direction

[Operation] When the signal is high, 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 signal is "1".



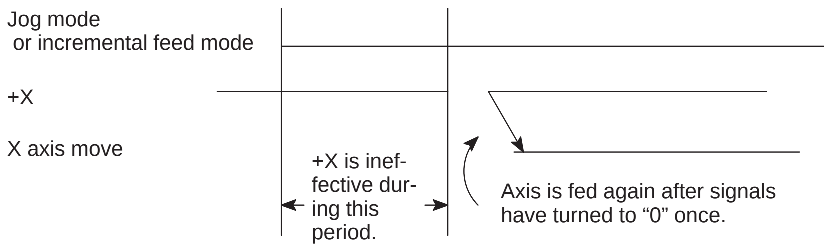
- In incremental feed, the control unit feeds the specified axis by the step distance which is specified by the manual handle feed move distance selection signal MP1, MP2. Then the control unit stops it. Even if the signal is set to "0" while the axis is being fed, the control unit does not stop feeding it.
  To feed the axis again, set the signal to "0", then set it to "1" again.



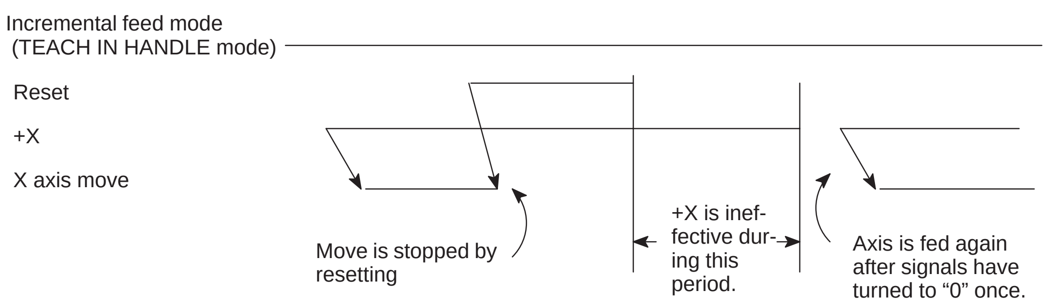
NOTE

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 cannot be validated by releasing the reset state. After the reset state is released, set these signals to “0”, then set them to “1” again.



Override signal *OV1 to *OV8 <G121#0 to #3>

[Classification] Input signal

[Function] Selects the feedrate for jog or incremental feed.

[Operation] During jog or incremental feed, the signal specifies the feedrate that is applied when the manual rapid traverse selection signal, RT, is 0.

A relationship between the state of the machine contact and the feedrate can be selected from the two in the following table according to parameter setting.

Machine contact state				When parameter OVRI is 0			When parameter OVRI is 1		
				Override value [%]	Jog feed		Override value [%]	Jog feed	
*OV1	*OV2	*OV4	*OV8		Metric system [mm/min]	Inch system [inch/min]		Metric system [mm/min]	Inch system [inch/min]
1	1	1	1	0	0	0	150	1260	50
0	1	1	1	10	20	0.08	140	790	30
1	0	1	1	20	3.2	0.12	130	500	20
0	0	1	1	30	5.0	0.2	120	320	12
1	1	0	1	40	7.9	0.3	110	200	8.0
0	1	0	1	50	12.6	0.5	100	126	5.0
1	0	0	1	60	20	0.8	90	79	3.0
0	0	0	1	70	32	1.2	80	50	2.0
1	1	1	0	80	50	2.0	70	32	1.2
0	1	1	0	90	79	3.0	60	20	0.8
1	0	1	0	100	126	5.0	50	12.6	0.5
0	0	1	0	110	200	8.0	40	7.9	0.3
1	1	0	0	120	320	12	30	5.0	0.2
0	1	0	0	130	500	20	20	3.2	0.12
1	0	0	0	140	790	30	10	2.0	0.08
0	0	0	0	150	1260	50	0	0	0

NOTE

1 Parameter OVRI is bit 4 of parameter No.003.
2 A newly selected feedrate becomes valid immediately when the override switch is operated during axis movement.
3 Generally, this signal is issued using the rotary switch.
4 Each feedrate value in the above table includes an error of ±3%.
5 In the M series units, the feedrate for jog feed can be changed by parameter setting.
6 This signal can be used also as an override signal for automatic dry run.

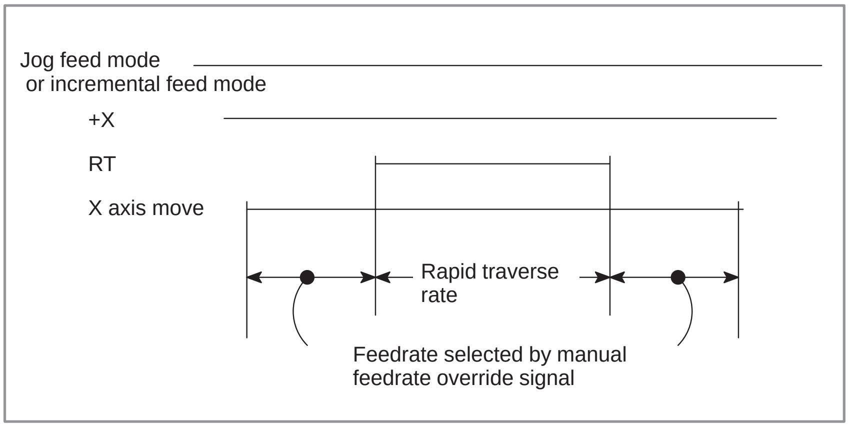
Manual rapid traverse selection signal RT<G121#6>

[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. A parameter ISOT (No.0010#0) can be specified so that the rapid traverse is validated before the reference position return is completed.

Jog feedrate override signal (M series) JOV1 to JOV8 <G104#0 to G104#3>

[Classification] Input signal

[Function] Specifies a value of jog feedrate override in the range of 0% to 150% in 10% increments. This signal is also effective for dry–run operation.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G104					JOV8	JOV4	JOV2	JOV1

Override (%)	JOV8	JOV4	JOV2	JOV1
0	0	0	0	0
10	0	0	0	1
20	0	0	1	0
30	0	0	1	1
40	0	1	0	0
50	0	1	0	1
60	0	1	1	0
70	0	1	1	1
80	1	0	0	0
90	1	0	0	1
100	1	0	1	0
110	1	0	1	1
120	1	1	0	0
130	1	1	0	1
140	1	1	1	0
150	1	1	1	1

	#7	#6	#5	#4	#3	#2	#1	#0	
G116					–X	+X			
G117					–Z	+Z			(T series)
					–Y	+Y			(M series)
G118					–3	+3			(T series)
					–Z	+Z			(M series)
G119					–4	+4			(M series)
G121		RT			*OV8	*OV4	*OV2	*OV1	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0008				MFPR					(T series)

[Data type] Bit

MFPR Jog feed (manual continuous feed)
0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

	#7	#6	#5	#4	#3	#2	#1	#0
0010								ISOT

[Data type] Bit

ISOT 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

	#7	#6	#5	#4	#3	#2	#1	#0
0049				S3JOG				

[Data type] Bit

S3JOG Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return
0 : 1 axis
1 : Up to 3 axes

0559 – 0562	Manual rapid traverse rate

[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	30 to 24000	30 to 12000
Inch machine	0.1 inch/min	30 to 9600	30 to 4800
Rotation axis	1 deg/min	30 to 24000	30 to 12000

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 parameters 0518 to 0521 (rapid traverse rate) is assumed.

0601 – 0604	Time constant of exponential acceleration/deceleration in jog feed

[Data type] Word

[Unit of data] 1 msec

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in jog feed.

0605 – 0608	FL rate of exponential acceleration/deceleration in jog 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
Rotation axis	1 deg/min	6 to 15000	6 to 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog 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.0013) enables both manual handle and incremental feed in the manual handle feed mode.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.3.2	JOG FEED
	III.3.3	INCREMENTAL FEED
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.3.2	JOG FEED
	III.3.3	INCREMENTAL FEED

3.2 MANUAL HANDLE FEED

General

In the manual handle feed mode, the tool can be minutely 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.0121 and 0699).
The handle magnifications can be selected by the manual handle feed move distance selection signal.

The number of manual pulse generators available depends on the type of an option used as listed below.

(M series)
– Control with one manual handle: Up to one generator
– Control with two or three manual handles: Up to three generators

(T series)
– Control with one manual handle: Up to one generator
– Control with two manual handles: Up to two generators

Two–path control

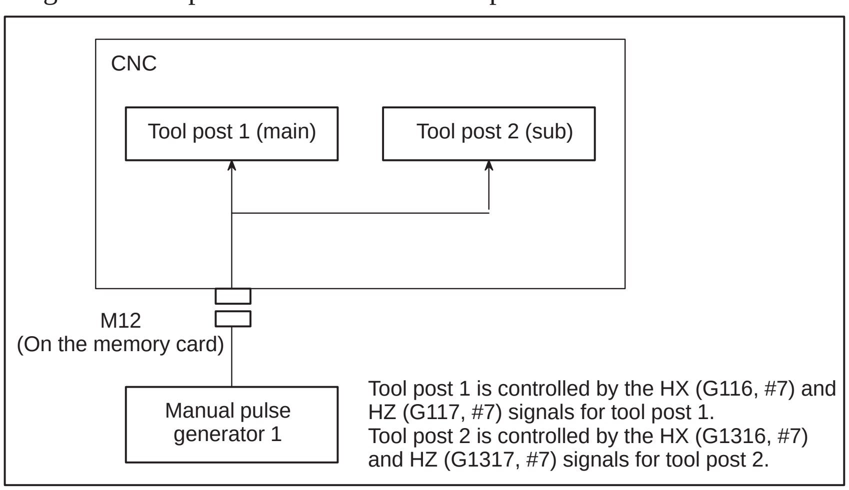
Which manual pulse generator moves which axis of which path depends on the setting of manual handle feed axis select signals for each path. For each tool post, eight bits are reserved as manual handle feed axis select signals.

Single/dual manual pulse generator control

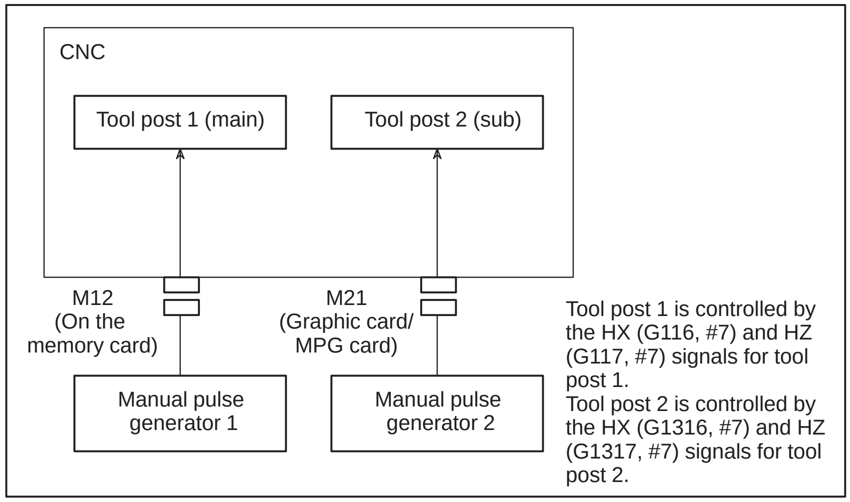
Single manual pulse generator control

With the 0–TTC, either of the following can be selected as single manual pulse generator control. The selection is made by specifying the SEPH bit (bit 2 of parameter 0047).

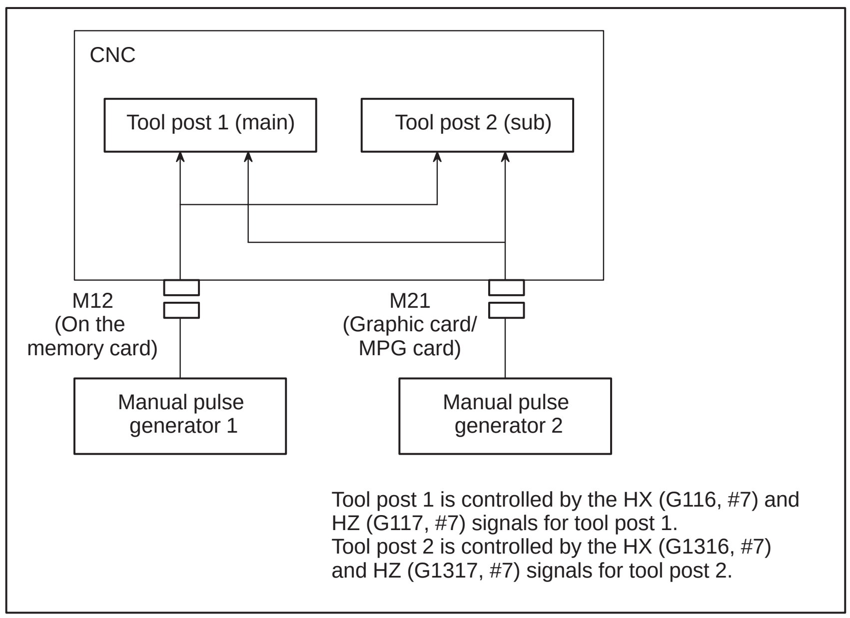
(1) Single manual pulse generator control when a single manual pulse generator is provided for the two tool posts. Set the SEPH bit to 0.



(2) Single manual pulse generator control when a single manual pulse generator is provided for each tool post. Set the SEPH bit to 1.



Dual manual pulse generator control



Availability of manual handle feed in Jog mode

Parameter JHD (bit 0 of No.0013) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(bit 0 of No.0013) is set 1,both manual handle feed and incremental feed are enabled.

Availability of manual handle feed in TEACH IN JOG mode

Parameter TJHD (bit 6 of No.0002) enables or disables the manual handle feed generator in the TEACH IN JOG mode.

A command to the MPG exceeding rapid traverse rate

Parameter HDLPM (No.0060#4 (M)/No.0077# (T)) 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.)

SET VALUE 1: 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 HPNEG (No.0386#0 to #3) (T) 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.

Signal

Manual handle feed axis selection signal HX to H4<G116#7 to G119#7>(M series)

[Classification] Input signal

[Function] (1) When only one manual pulse generator is used
The following table lists the relationships between the manual handle feed axis selection signal and the axis to which the signal is applied. Any combination not listed below does not cause axis movement at all.

Manual handle feed axis selection signal				Feed axis
H4	HZ	HY	HX	
0	0	0	1	X axis
0	0	1	0	Y axis
0	1	0	0	Z axis
1	0	0	0	Fourth axis

(2) With two manual pulse generators
Parameter 0118 (number of manual pulse generators) is set to 2.

(2.1) Multi–handle function of type A
(MHPGB bit (bit 0 of parameter 0019) set to 0)
The table below indicates the relationship between the manual handle feed axis select signals and feed axes. No other signal combination can cause axial movement.

Manual handle feed axis selection signal				Feed axis	
H4	HZ	HY	HX	1st handle	2nd handle
0	0	1	1	X axis	Y axis
0	1	0	1	X axis	Z axis
0	1	1	0	Y axis	Z axis
1	0	0	1	X axis	Fourth axis
1	0	1	0	Y axis	Fourth axis
1	1	0	0	Z axis	Fourth axis

(2.2) Multi–handle function of type B
(MHPGB bit (bit 0 of parameter 0019) set to 1)
The table below indicates the relationship between the manual handle feed axis select signals and feed axes.

Feed axis	Manual pulse generator to be used
X axis	First manual pulse generator
Y axis	Second manual pulse generator
Z axis	Determined by signals G133, #0 and #1
Fourth axis	Determined by parameter 0117

G133		Manual pulse generator corresponding to the Z–axis
#1 SLHZ1	#0 SLHZ0	
0	0	Determined by parameter 0117
0	1	First manual pulse generator
1	0	Second manual pulse generator

(3) With three manual pulse generators
Parameter 0118 (number of manual pulse generators) is set to 3.

(3.1) Multi–handle function of type A
(MHPGB bit (bit 0 of parameter 0019) set to 0)

(3.1.1) The HSLE bit (bit 7 of parameter 0003) is set to 0.
(The axis select signals are disabled.)
The handle rotation of a manual pulse generator causes the corresponding axial movement, irrespective of the axis select signals.
This setting cannot be made for a four–axis system. (If the setting is attempted on a four–axis system, axial movement cannot be performed.)

Feed axis		
1st handle	2nd handle	3rd handle
X axis	Y axis	Z axis

(3.1.2) The HSLE bit (bit 7 of parameter 0003) is set to 1.
(The axis select signals are enabled.)
The table below indicates the relationship between the manual handle feed axis select signals and feed axes. No other signal combination can cause axial movement.

Manual handle feed axis selection signal				Feed axis		
H4	**HZ**	**HY**	**HX**	**1st handle**	**2nd handle**	**3 handle**
0	1	1	1	X axis	Y axis	Z axis
1	0	1	1	X axis	Y axis	Fourth axis
1	1	0	1	X axis	Z axis	Fourth axis
1	1	1	0	Y axis	Z axis	Fourth axis

(3.2) Multi–handle function of type B
(MHPGB bit (bit 0 of parameter 0019) set to 1)

(3.2.1) The HSLE bit (bit 7 of parameter 0003) is set to 0.
(The axis select signals are disabled.)
Handle rotation of a manual pulse generator causes the corresponding axial movement, irrespective of the axis select signals.
This setting cannot be made for a four–axis system.
(If the setting is attempted on a four–axis system, axial movement cannot be performed.)

Feed axis		
1st handle	**2nd handle**	**3 handle**
X axis	Y axis	Z axis

(3.2.2) The HSLE bit (bit 7 of parameter 0003) is set to 1.
(The axis select signals are enabled.)
The table below indicates the relationship between the manual handle feed axis select signals and feed axes.

Feed axis	Manual pulse generator to be used
X axis	First manual pulse generator
Y axis	Second manual pulse generator
Z axis	Determined by signals G133, #0 and #1
Fourth axis	Determined by parameter 0117

G133		Manual pulse generator corresponding to the Z–axis
#1 SLHZ1	**#0 SLHZ0**	
0	0	Determined by parameter 0117
0	1	First manual pulse generator
1	0	Second manual pulse generator
1	1	Third manual pulse generator

Manual handle feed axis selection signal HX to H4 <G116#7 to G119#7> (T series)

[Classification] Input signal

[Function] (1) When only one manual pulse generator is used
The following table lists the relationships between the manual handle feed axis selection signal and the axis to which the signal is applied. Any combination not listed below does not cause axis movement at all.

Manual handle feed axis selection signal				Feed axis
H4	H3	HZ	HX	
0	0	0	1	X axis
0	0	1	0	Z axis
0	1	0	0	Third axis
1	0	0	0	Fourth axis

(2) With two manual pulse generators

(2.1) The HSLE bit (bit 5 of parameter 0002) is set to 0.
(The axis select signals are disabled.)
Handle rotation of a manual pulse generator causes the corresponding axial movement, irrespective of the axis select signals.

Feed axis	
1st handle	2nd handle
X axis	Z axis

(2.2) The HSLE bit (bit 5 of parameter 0002) is set to 1.
(The axis select signals are enabled.)
The table below indicates the relationship between the manual handle feed axis select signals and feed axes. No other signal combination can cause axial movement.

Manual handle feed axis selection signal				Feed axis	
H4	H3	HZ	HX	1st handle	2nd handle
0	0	1	1	X axis	Z axis
0	1	0	1	X axis	Third axis
0	1	1	0	Z axis	Third axis
1	0	0	1	X axis	Fourth axis
1	0	1	0	Z axis	Fourth axis
1	1	0	0	Third axis	Fourth axis

Manual Handle Feed Amount Selection Signal MP1, MP2 <G120#0, #1>(M) <0117#0, G118#0>(T) (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.0121 and 0699.

> **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.

> **NOTE**
> See Section 3.3, “Manual Handle Interrupt” for manual handle interrputs, Section 3.1, “Jog Feed/Incremental Feed” for incremental feed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G116	HX								
G117	HZ							MP1	(T series)
	HY								(M series)
G118	H3							MP2	(T series)
	HZ								(M series)
G119	H4								
G120							MP2	MP1	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0002		TJHD						

[Data type] Bit

TJHD Manual pulse generator in TEACH IN JOG mode
0 : Invalid
1 : Valid

	#7	#6	#5	#4	#3	#2	#1	#0	
0002			HSLE						(T series)
0003	HSLE								(M series)

[Data type] Bit

HSLE When two (T series) or three (M series) manual pulse generators are provided, the axis select signals are:
1 : Enabled. (Handle rotation of a manual pulse generator is ignored when the corresponding axis select signal is set to 0.)
0 : Disabled. (Handle rotation of a manual pulse generator causes the corresponding axial movement, irrespective of the axis select signals.)

	#7	#6	#5	#4	#3	#2	#1	#0
0013								JHD

[Data type] Bit

JHD Manual handle feed in JOG mode or incremental feed in the manual handle feed
0 : Invalid
1 : Valid

	#7	#6	#5	#4	#3	#2	#1	#0	
0018					NAMP2	NZMP2	NYMP2	NXMP2	(M series)

[Data type] Bit

N*MP2 Specify whether to enable the handle feed multiply value ($\times$ 100) for each axis, as follows:
1 : Disable
0 : Enable

	#7	#6	#5	#4	#3	#2	#1	#0	
0019								MHPGB	(M series)

[Data type] Bit

MHPG B The multi–handle function of:
1 : Type B is used.
0 : Type A is used.

	#7	#6	#5	#4	#3	#2	#1	#0	
0060				HDLPM					(M series)
0077				HDLPM					(T series)

[Data type] Bit

HDLPM 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
0075						INHND		

[Data type] Bit

INHND The units of movement by manual handle interrupt are:
1 : The input unit. Acceleration/deceleration is enabled.
0 : The output unit. Acceleration/deceleration is disabled.

0117	Manual pulse generator for the Z–axis or fourth axis.	(M series)

[Data type] Byte

[Valid data range] 12 to 32

[Description] When the type B multi–handle function is used, this parameter is used to set which manual pulse generator causes the movement on the Z–axis or fourth axis. Specify a two–digit decimal number. The units digit indicates the ordinal number of the manual pulse generator for the Z–axis, while the tens digit indicates that for the fourth axis.

Example) To set the second manual pulse generator for the Z–axis and the third manual pulse generator for the fourth axis, specify 32.

Set value = 3 2

2 — Ordinal number of the manual pulse generator for the Z–axis

3 — Ordinal number of the manual pulse generator for the fourth axis

0118	Number of manual pulse generators used	(M series)

[Data type] Byte

[Valid data range] 1, 2, or 3

This parameter sets the number of manual pulse generators.

0121	Manual handle feed magnification m

[Data type] Byte

[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.

	#7	#6	#5	#4	#3	#2	#1	#0
0386	HDPG4	HDPG3	HDPG2	HDPG1				

[Data type] Bit

HDPG* Specify whether to enable the handle feed multiply value ($\times$1000) for each axis, as follows:
1 : Disable
0 : Enable

	#7	#6	#5	#4	#3	#2	#1	#0	
0386					HPNEG4	HPNEG3	HPNEG2	HPNEG1	(T series)

[Data type] Bit

HPNEG* Axis movement direction for rotation direction of manual pulse generator
0 : Same in direction
1 : Reverse in direction

0699	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”.

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.3.4	MANUAL HANDLE FEED
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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. 0121 and 0699). 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 HIX to HI4 <G126#0 to #3>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied.

The selected feed axis is similar to the correspondence with the manual handle feed axis select signals. See Section 3.2, “Manual Handle Feed.”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G126					HI4	HI3	HIZ	HIX	(T series)
					HI4	HIZ	HIY	HIX	(M series)

Warning

WARNING
The travel distance 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 this 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 operation mode (for example, job feed mode, manual handle feed mode and TEACH IN HANDLE mode).

2 Handle interruption is disabled when the machine is locked or interlocked.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.4.7	MANUAL HANDLE INTERRUPTION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.4.8	MANUAL HANDLE INTERRUPTION

4 REFERENCE POSITION ESTABLISHMENT

4.1 MANUAL REFERENCE POSITION RETURN

General

The tool is moved in the direction specified in parameter ZM* (No.0003#0 – #3) for each axis by turning the feed axis and direction select signal to "1" in the manual reference position return mode, and is returned to the reference position.

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.

• Automatic setting of coordinate system

Bit 7 of parameter 0010 (APRS) can be set to automatically determine the coordinate system at manual reference position return. Parameters 0708 – 0711 can be set to determine the workpiece coordinate system by assigning, upon the completion of reference position return, the value set in a parameter to a reference point on the tool holder or the tip position of the reference tool.

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
Selection of axis to be moved	+X, –X, +Y, –Y, +Z, –Z, ...
Selection of direction to be moved	
Selection of speed to be moved	ROV1, ROV2
Deceleration signal for reference position return	*DECX, *DECY, *DECZ, ...
Completion signal for reference position return	ZPX, ZPY, ZPZ, ...
Reference position establishment signal	ZRFX, ZRFY, ZRFZ, ...

Basic Procedure for Reference Position Return

(1) Select the JOG mode or TEACH IN JOG mode, and set the manual reference position return selection signal ZRN to "1".

(2) Feed a target axis toward the reference position by making an appropriate feed axis and direction selection signal (+X, –X, +Y, –Y,…) "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 turned on, making the deceleration signal (*DECX, *DECY, *DECZ,…) for reference position deceleration “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.0534) setting).

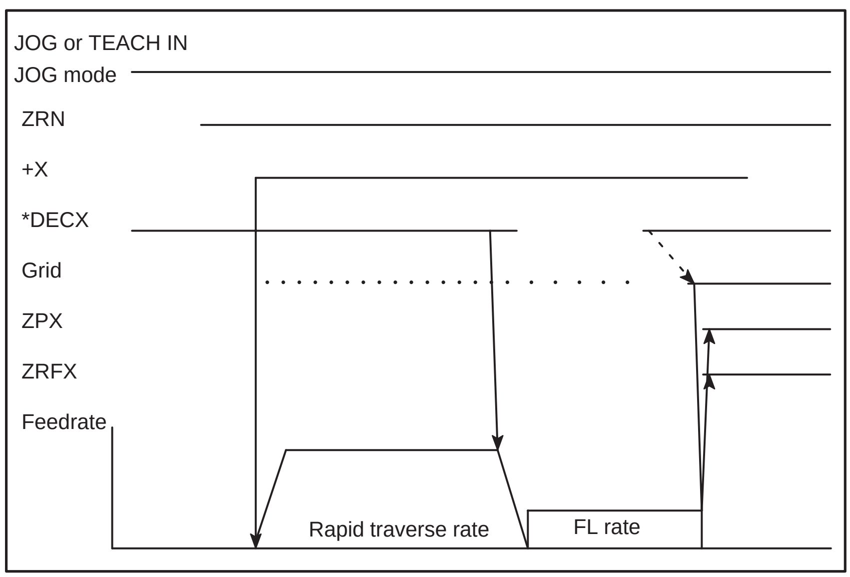
(5) When the deceleration signal turns to “1” again after the limit switch for deceleration is passed, the tool is fed with the feedrate unchanged, then the tool stops at the first grid point (electric grid point).

(6) Upon confirmation that the current position is in the in–position area, the reference position return end signal (ZPX, ZPY, ZPZ,…) and the reference position establishment signal (ZRFX, ZRFY, ZRFZ,…) turn to “1”.

Step (2) and subsequent steps are performed independently for each axis. The number of simultaneously controlled axes is usually one, but it becomes three by parameter S3JOG (No.0049#4).

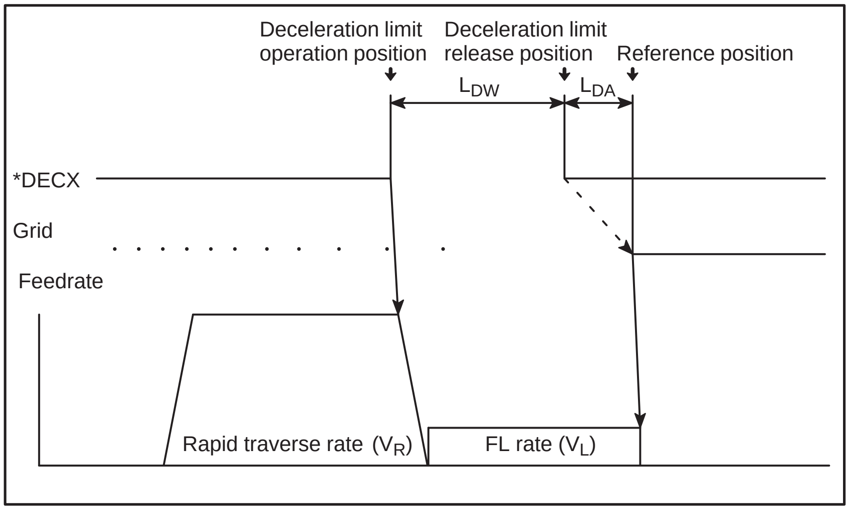
If the feed axis direction selection signal (+X, –X, +Y, –Y,…) turns to “0” between step (2) and (5) , the tool is stopped at once, and reference position return is assumed to be canceled. If the signal turn to “1” again, operation resumes from step (3) (rapid traverse).

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 manual reference position return when the reference position has not yet been established, 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 value exceeds 128. At this 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 set the servo position error to 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} and the detection unit is 1 μm, the tool must be fed in the reference position return direction at a speed of at least 230 mm/min before manual reference position return.

Grid shift

The grid can be shifted by the distance set in parameters 0508 to 0511, thus shifting the reference position. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No.0004#0 to #3, No.0007#0 to #3) (grid interval).

Signal

Manual reference position return selection signal ZRN<G120#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 becomes as described below.

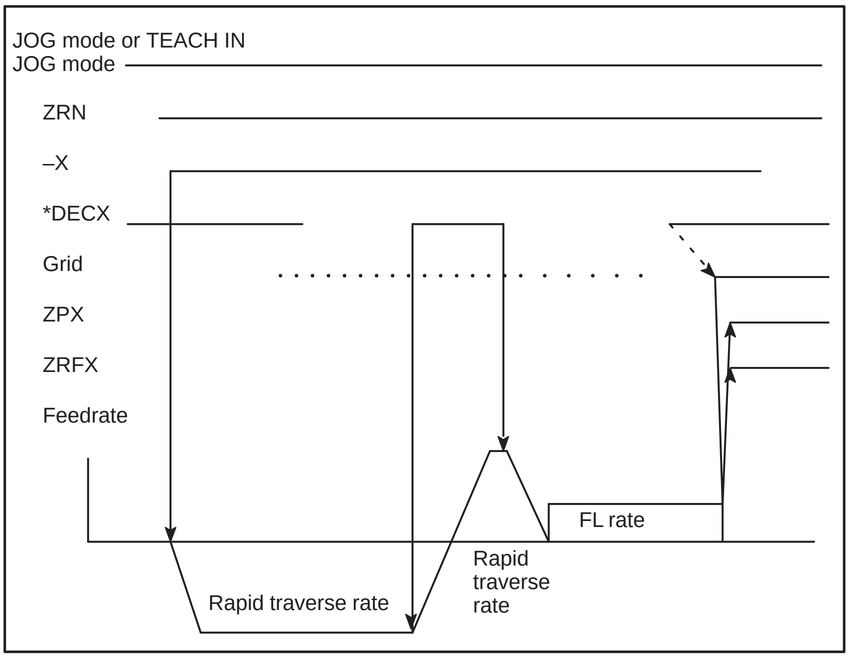
- If jog feed mode is not selected, the control unit ignores the manual reference position return selection signal.
- If jog mode is selected, manual reference position return is enabled.

NOTE
If the ZRN status changes from “0” to “1” or “1” to “0” during jog feed, the feedrate is decelerated to 0. Then, to make reference position return or jog feed, turn feed axis and direction selection signal to “0” then set it to “1”.

Feed Axis and Direction Selection Signal

For details about this signal, see 3.1.2, “Feed Axis and Direction Selection Signal”. Here, only notes on use of reference position return are given.

NOTE
1 The direction of reference position return is predetermined for each axis by parameter ZM* (No.0003#0 to #3). If the tool is fed in the opposite direction to the predetermined direction in manual reference position return, the deceleration signal for reference position return turns to “0”, and the tool is returned to the point at which the deceleration signal turns to “1” again (that is, the point where the deceleration limit switch would be encountered if the tool were fed in the predetermined direction). Then reference position return is performed automatically in the predetermined direction.



NOTE
2 When reference position return is selected, an axis whose reference position return end signal is already “1” or an axis whose reference position return end signal was set “1” upon completion of reference position return is locked, and movement along that axis is disabled while the reference position return selection signal (ZRN) is “1”. To perform movement along such an axis, ZRN must be set “0”, and the feed axis and direction selection signal must be set “0” then set “1” again.

Reference position return deceleration signals *DECX to *DEC4 <X016#5 to X019#5>

[Classification] Input signal

[Function] These signals decelerate the feedrate for manual reference position return so that the reference position is approached at a low feedrate.
The deceleration signals are provided for axes in a one–to–one correspondence. A number or alphabet appended to a deceleration signal represents a controlled axis number.

*DEC x

X : Reference position return deceleration signal for the X axis
Y : Reference position return deceleration signal for the Y axis
Z : Reference position return deceleration signal for the Z axis
:
:

[Operation] For the operation of the control unit in response to the deceleration signal, see the description of the basic procedure for manual reference position return.

Reference position return completion signals ZPX to ZP4 <F148#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 axes in a one–to–one corespondence.
A number or alphabet appended to a signal represents a controlled axis number.

ZP x

X : Reference position return end signal for the X axis
Y : Reference position return end signal for the Y axis
Z : Reference position return end signal for the Z axis
:
:

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

- Manual reference position returns is completed, and the current position is in the in–position area.
- Automatic reference position return (G28) is completed, and the current position is in the in–position area.
- Reference position return check (G27) is completed, and the current 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 ZRFX to ZRF4 <F168#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 name indicates the number or alphabet of the controlled axis.

ZRF x

X : X–axis reference position establishment signal
Y : Y–axis reference position establishment signal
Z : Z–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	
X016			*DECX						
X017			*DECZ						(T series)
			*DECY						(M series)
X018			*DEC3						(T series)
			*DECZ						(M series)
X019			*DEC4						(M series)

	#7	#6	#5	#4	#3	#2	#1	#0
G120	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0	
F148					ZP4	ZP3	ZPZ	ZPX	(T series)
					ZP4	ZPZ	ZPY	ZPX	(M series)
F168					ZRF4	ZRF3	ZRFZ	ZRFX	(T series)
					ZRF4	ZRFZ	ZRFY	ZRFX	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001			DECI					

[Data type] Bit

DECI Deceleration signal (*DECX 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.

	#7	#6	#5	#4	#3	#2	#1	#0	
0003					ZM4	ZM3	ZMZ	ZMX	(T series)
					ZM4	ZMZ	ZMY	ZMX	(M series)

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

[Data type] Bit

ZMx The direction of reference position return and the initial direction of backlash at power–on
0 : Positive direction
1 : Negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
0004 – 0007					Reference counter size			

[Data type] Bit

[Valid data range] 1000 to 16000

Set the size of the reference counter.

As 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.

	#7	#6	#5	#4	#3	#2	#1	#0
0010	APRS							

[Data type] Bit

APRS Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

	#7	#6	#5	#4	#3	#2	#1	#0	
0024		CLCL							(M series)
0075		CLCL							(T series)

[Data type] Bit

CLCL 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.

	#7	#6	#5	#4	#3	#2	#1	#0	
0041						JRNJF			(T series)

[Data type] Bit

JRNJF The manual reference position return at JOG feedrate
0 : Not performed
1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
0049				S3JOG				

[Data type] Bit

S3JOG Number of axes controlled simultaneously in JOG feed, manual rapid traverse and manual reference position return
0 : 1 axis
1 : Up to 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
0063							PRSTIN	

[Data type] Bit

PRSTIN Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter Nos.0708 to 0711 is used.
1 : For input in mm, the value set in parameter Nos.0708 to 0711 is used, or for input in inches, the value set in parameter Nos.0815 to 0818 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0065					PSOT			

[Data type] Bit

PSOT Checking of stored stroke limit during the time from power–on to the manual reference position return
0 : The stroke limit is checked.
1 : The stroke limit is not checked

	#7	#6	#5	#4	#3	#2	#1	#0	
0074					CFR4	CFR3	CFRZ	CFRX	(T series)
					CFR4	CFRZ	CFRY	CFRX	(M series)

[Data type] Bit

CFRx When a command specifying the movement except for G28 is issued in automatic operation (MEM or MDI) when a return to the reference position has not been performed since the power was turned on

0 : An alarm is not generated.

1 : An alarm is generated. (P/S alarm 224).

	#7	#6	#5	#4	#3	#2	#1	#0
0399	OUTZRN							

[Data type] Bit

OUTZRN 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 cycle such as a dwell cycle or canned cycle is being executed.

0508 – 0511	Grid shift value

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to $\pm$32767

A grid shift is set.

To shift the reference position, the grid can be shifted by the amount set in this parameter. Up to the half of 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.

0534	FL rate of the reference position return

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed.

0708 – 0711	Coordinate value of the reference position on each axis used for setting a coordinate system automatically

[Data type] Two–word

[Unit of data]

Increment system	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

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

0815 – 0818	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in inches)	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

NOTE

This parameter is valid when PRSTIN in parameter 0063#1 is set to 1.

Alarm and message

Number	Message	Description
090	REFERENCE RETURN IN-COMPLETE	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. Check the program contents.
091	REFERENCE RETURN IN-COMPLETE	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 No.0074#0 to #3 are 1. Do reference position return.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.3.1	MANUAL REFERENCE POSITION RETURN
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.3.1	MANUAL REFERENCE POSITION RETURN

4.2 SETTING THE REFERENCE POSITION WITHOUT DOGS

General

This function moves the tool near around the reference position set for each axis in the manual continuous feed mode. Then it sets the reference position in the reference position return mode without the deceleration signal for reference position return by turning 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 switch for deceleration for reference position return.

Also, 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

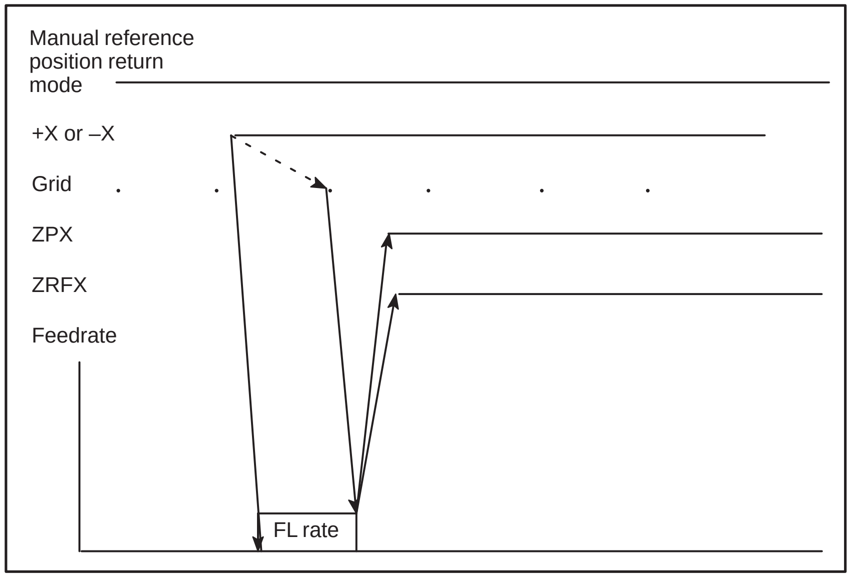
(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.

(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.

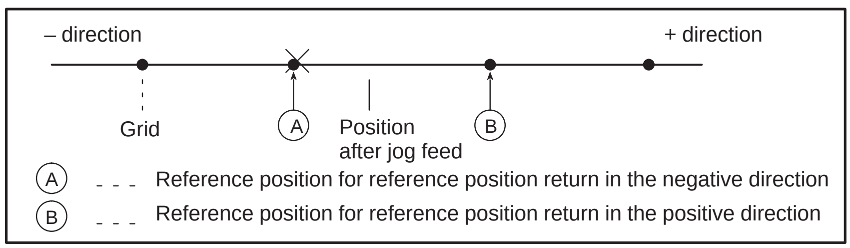
(3) The CNC positions the tool to the nearest grid line (based on one–rotation signals from the position detector) in the reference position return direction specified with bits 0 to 3 (ZMx) of parameter No.0003. The point to which the tool is thus positioned becomes the reference position.

(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 value exceeds 128. 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 Nos.0508 to 0511. The grid shift to be set in the parameter must not exceed the reference counter capacity (bits 0 to 3 of parameter Nos.0004 to 0007).

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 with 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	
0003					ZM4	ZM3	ZMZ	ZMX	(T series)
					ZM4	ZMZ	ZMY	ZMX	(M series)

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

[Data type] Bit

ZMx 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
0004 – 0007					Reference counter size			

[Data type] Bit

[Valid data range] 1000 to 16000

Set the size of the reference counter.

As 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.

	#7	#6	#5	#4	#3	#2	#1	#0
0010	APRS							

[Data type] Bit

APRS Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

	#7	#6	#5	#4	#3	#2	#1	#0	
0024		CLCL							(M series)
0075		CLCL							(T series)

[Data type] Bit

CLCL 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.

	#7	#6	#5	#4	#3	#2	#1	#0
0049				S3JOG				

[Data type] Bit

S3JOG Number of axes controlled simultaneously in manual continuous feed, manual rapid traverse and manual reference position return
0 : 1 axis
1 : Up to 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
0063							PRSTIN	

[Data type] Bit

PRSTIN Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter Nos.0708 to 0711 is used.
1 : For input in mm, the value set in parameter Nos.0708 to 0711 is used, or for input in inches, the value set in parameter Nos.0815 to 0818 is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0065					PSOT			

[Data type] Bit

PSOT Checking of stored stroke limit during the time from power–on to the manual position reference return
0 : The stroke limit is checked.
1 : The stroke limit is not checked

	#7	#6	#5	#4	#3	#2	#1	#0	
0074					CRF4	CRF3	CRFZ	CRFX	(T series)
					CRF4	CRFZ	CRFY	CRFX	(M series)

[Data type] Bit

CRFx When a command specifying the movement except for G28 is issued in automatic operation (AUTO or MDI) and when a return to the reference position has not been performed since the power was turned on
0 : An alarm is not generated.
1 : An alarm is generated (P/S alarm 224).

	#7	#6	#5	#4	#3	#2	#1	#0
0076							JZRN	

[Data type] Bit

JZRN Function setting the reference position without dog
0 : Disabled
1 : Enabled

> **NOTE**
> This function can be specified for each axis by JZRNx, bits 0 to 5 of parameter No.0391.

	#7	#6	#5	#4	#3	#2	#1	#0
0391			JZRN8	JZRN7	JZRN4	JZRN3	JZRNZ	JZRNX

[Data type] Bit

JZRNx Function for setting the reference position without dogs for each axis
0 : Enabled
1 : Disabled

> **NOTE**
> When parameter JZRN (No.0076#1) is 1, these parameters are enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
0399	OUTZRN							

[Data type] Bit

OUTZRN 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 a auxiliary function (miscellaneous function, spindle–speed function, tool function) is being executed.
- When a cycle such as a dwell cycle or canned cycle is being executed.

0508 – 0511	Grid shift value

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to ±32767

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 half of 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.

0534	FL rate of the reference position return

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed.

0708 – 0711	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word

[Unit of data]

Increment system	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

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

0815 – 0818	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (input in inches)	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically when input is performed in inches.

> **NOTE**
> This parameter is valid when PRSTIN in parameter 0063#1 is set to 1.

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	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. Check the program contents.
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 bits 0 to 3 of parameter No.0074 are 1). Do reference position return.

Note

> **NOTE**
> Alarm No. 090 is issued when G28 is specified if the reference position has not yet be established.

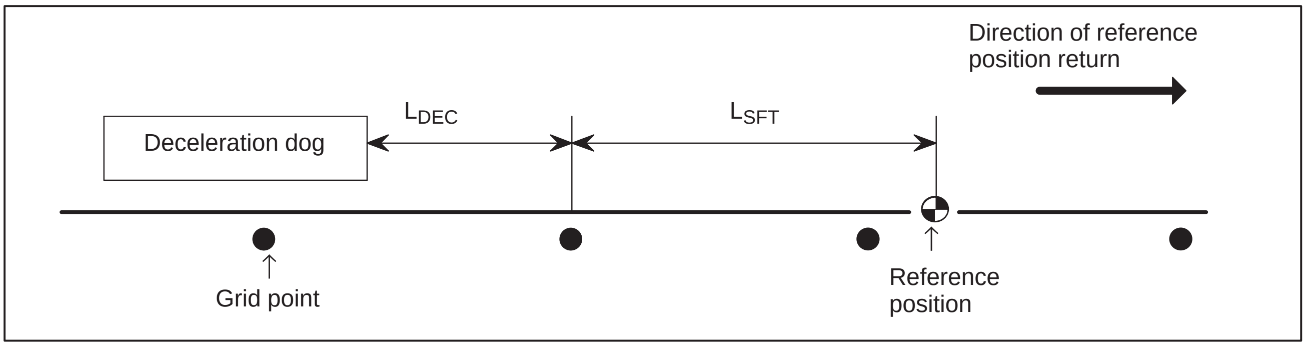
4.3 REFERENCE POSITION SHIFT (M SERIES)

General

When reference position return is performed using a 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 3 of parameter No. 0399 (SFDEC) to 1. When distance L_{SFT}, shown below, is set in parameter Nos. 0508 to 0511, the reference position can be shifted.

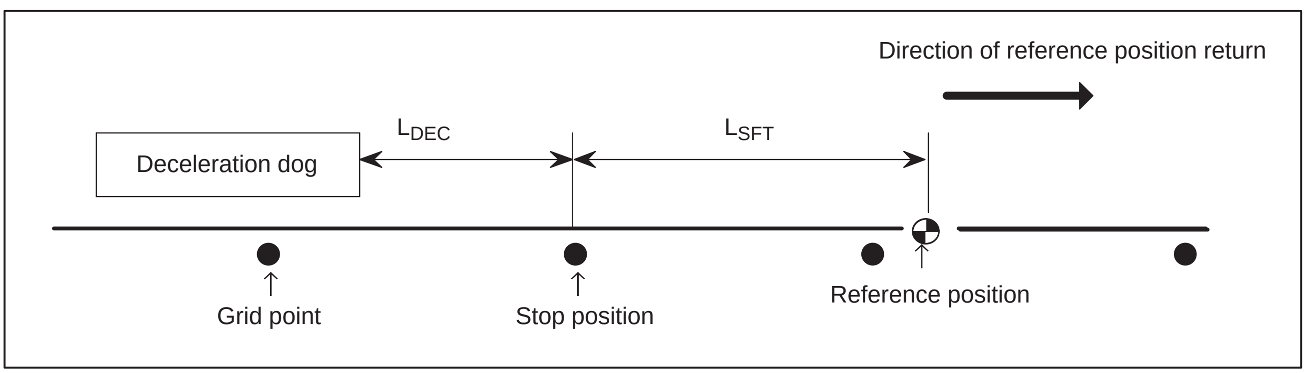
Distance L_{DEC}', shown below, for the axis along which reference position return was last made is indicated on the diagnostic screen (No. 0956).



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 SFDEC bit (bit 3 of parameter No. 0399) to 1, and set the reference position shift amount to 0. Then, perform reference position return.

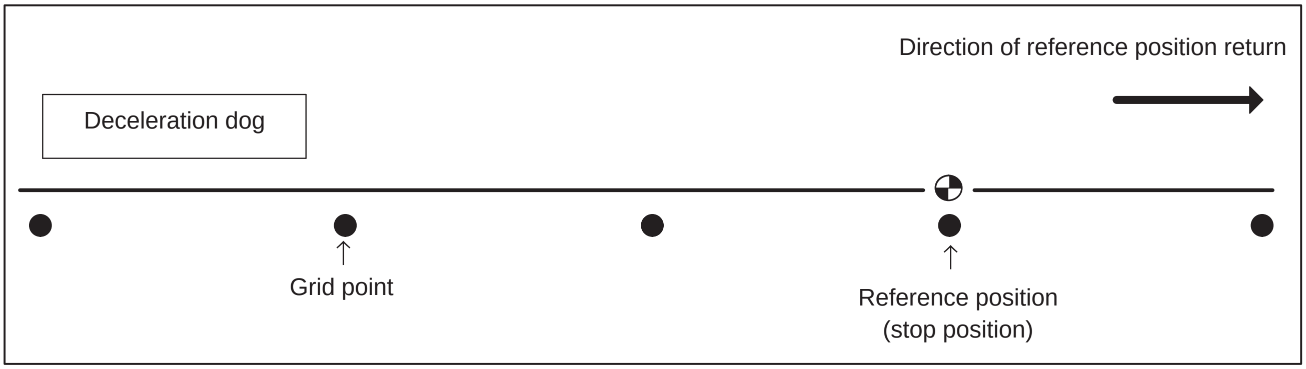


After the deceleration dog is turned off, the tool stops when the first grid point is reached. Distance L_{DEC} is indicated on the diagnostic screen (No. 0956).

(2) Determine the distance L_{SFT} (reference position shift amount) from the stop position to the reference position, and set it in parameter Nos. 0508 to 0511.

This completes the adjustment of the reference position.

(3) Perform reference position return again. Then, the tool stops when it reaches the reference position.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0399					SFDEC			

[Data type] Bit

SFDEC The function for shifting the reference position is
0 : Not used
1 : Used

0508 – 0511	Reference position shift

[Data type] Word

[Unit of data] Detection unit

[Valid data range] –32767 to 32767

A reference position shift is set for each axis.

CAUTION
When bit 3 of parameter No. 0399, SFDEC, 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.

● Diagnostic display

0956	Distance from the position where the deceleration dog is turned off to the first grid point

[Data type] Two–word

[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 SFDEC bit (bit 3 of parameter No. 0399) 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, along the specified axis, then sets the completion signal for reference position return (see Section 4.1) to 1.

The tool moves to the intermediate point or reference position at the rapid traverse rate.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0003					ZM4	ZM3	ZMZ	ZMX	(T series)
					ZM4	ZMZ	ZMY	ZMX	(M series)

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

[Data type] Bit

ZMx The direction of reference position return and the direction of initial backlash at power–on
0 : Positive direction
1 : Negative direction

Alarm and message

Number	Message	Description
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an CNC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.

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 bits 0 to 3 (ZMx) of parameter No.0003. The intermediate point must therefore 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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.6	REFERENCE POSITION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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, along the specified axis. Then, it sets the completion signal for 2nd, 3rd, or 4th reference position return to 1.

The 2nd, 3rd, or 4th reference position must be set in parameter Nos. 0735 to 0738 and 0780 to 0787 with coordinates in the machine coordinate system, before issuing the G30 command.

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 completion signals ZP2X to ZP24 <F161#0 to #3> Third reference position return completion signals ZP3X to ZP34 <F169#0 to #3> Fourth reference position return completion signals ZP4X to ZP44 <F169#4 to #7>

[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, respectively. These signals are provided for axes in a one–to –one correspondence. A numeric character or alphabet 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 x

- 1 : Return end signal for the X axis
- 2 : Return end signal for the Y axis
- 3 : Return end signal for the Z 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	
F161					ZP24	ZP23	ZP2Z	ZP2X	(T series)
					ZP24	ZP2Z	ZP2Y	ZP2X	(M series)
F169	ZP44	ZP43	ZP4Z	ZP4X	ZP34	ZP33	ZP3Z	ZP3X	(T series)
	ZP44	ZP4Z	ZP4Y	ZP4X	ZP34	ZP3Z	ZP3Y	ZP3X	(M series)

Parameter

0735 to 0738	Coordinate value of the second reference position on each axis in the machine coordinate system
0780 to 0783	Coordinate value of the third reference position on each axis in the machine coordinate system
0784 to 0787	Coordinate value of the fourth reference position on each axis in the machine coordinate system

[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
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
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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.6	REFERENCE POSITION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.6	REFERENCE POSITION

4.6 REFERENCE POSITION SETTING BY BUTTING AXIS AGAINST STOPPER

General

This function sets the machine zero point by butting an axis against its mechanical stopper while the servo motor is under torque control. Because this butting is performed automatically, stable reference position setting can be realized easily. This function is optional and is enabled only when an absolute–position detector is used.

Reference position setting is performed as follows:

1 The servo motor is subjected to torque control as specified in a parameter in order to suppress the torque when the axis is butted against the mechanical stopper.

2 The axis is butted against the stopper at the speed specified in a parameter. Whether the axis is butted against the stopper is determined from the torque limit reach signal, sent from the servo motor. Then, movement is performed in the opposite direction, by an amount specified in a parameter.

3 The axis is again butted against the mechanical stopper at the speed specified in another parameter. The position of the stopper is determined from the torque limit reach signal sent from the servo motor. Movement in the opposite direction is made, starting from that position, by the amount specified in a parameter. The resulting position is set as the machine zero point.

4 The servo motor is released from torque control.

Operation

Start reference position setting by butting the axis against the stopper, by applying the following procedure:

(1) Set the parameters required for reference position setting by butting the axis against the stopper.

(2) Select reference position return mode.

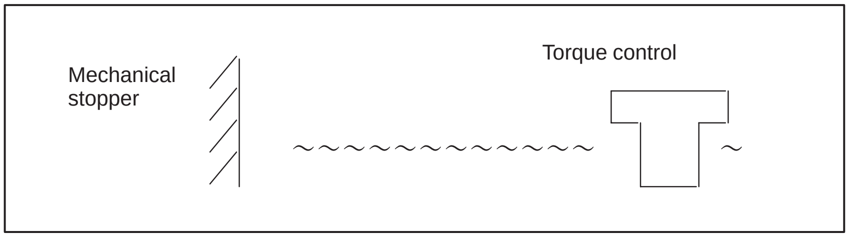
(3) Turn on the manual handle feed axis select signal for the target axis for reference position setting. (Because an absolute–position detector is provided, a reference position return request alarm, issued for the axis, is displayed.)

(4) Turn on the cycle start (ST) signal.

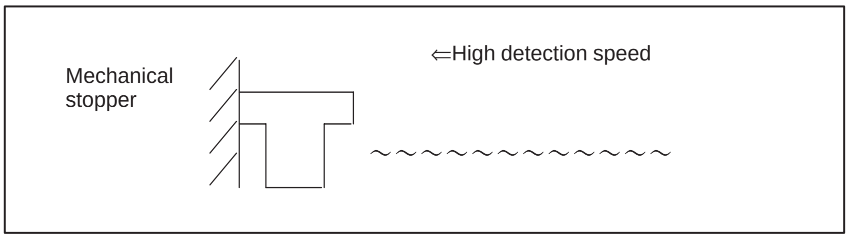
Now, reference position setting by pressing axis to stopper starts, causing the automatic operation start (OP) signal to be output. Upon the completion of reference position setting, the reference position return complete (ZP) signal is output, thus disabling the OP signal. In addition, alarm 000 is displayed. Repeat the above procedure for each necessary axis. Then, turn the power off. The reference position is set at the next power–up. If the above operation is performed again, positioning to the established reference position is done by rapid traverse. During positioning, however, the rapid traverse override function is disabled.

Operation of reference position setting

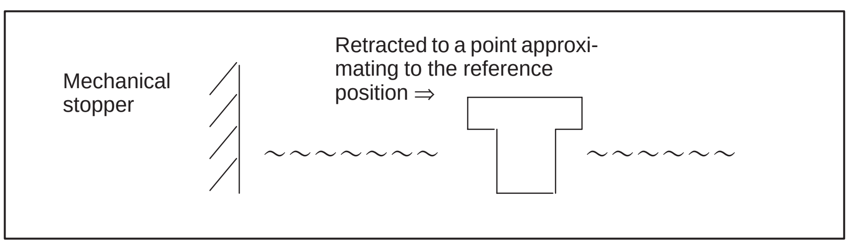
(1) The target axis of reference position setting is subjected to torque control as specified in parameters 360 to 363 (M series) or parameters 372 to 375 (T series).



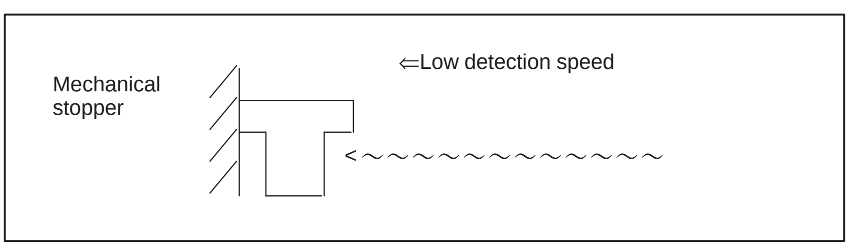
(2) The target axis of reference position setting is moved in the direction specified in bits 0 to 3 of parameter 003 at the speed (high detection speed) specified in parameter 942 (M series) or parameter 451 (T series).



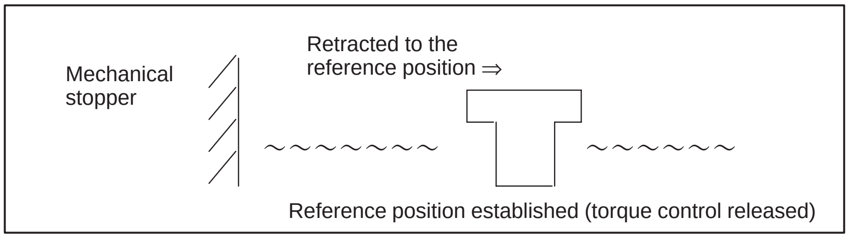
(3) When the torque limit reach signal sent from the servo motor goes on, the axis is assumed to be butted against the mechanical stopper. Then, movement is made in the opposite direction by the amount specified in parameters 944 to 947 (M series) or parameters 459 to 462 (T series) at the speed specified in parameters 944 to 947 (M series) or parameters 459 to 462 (T series).



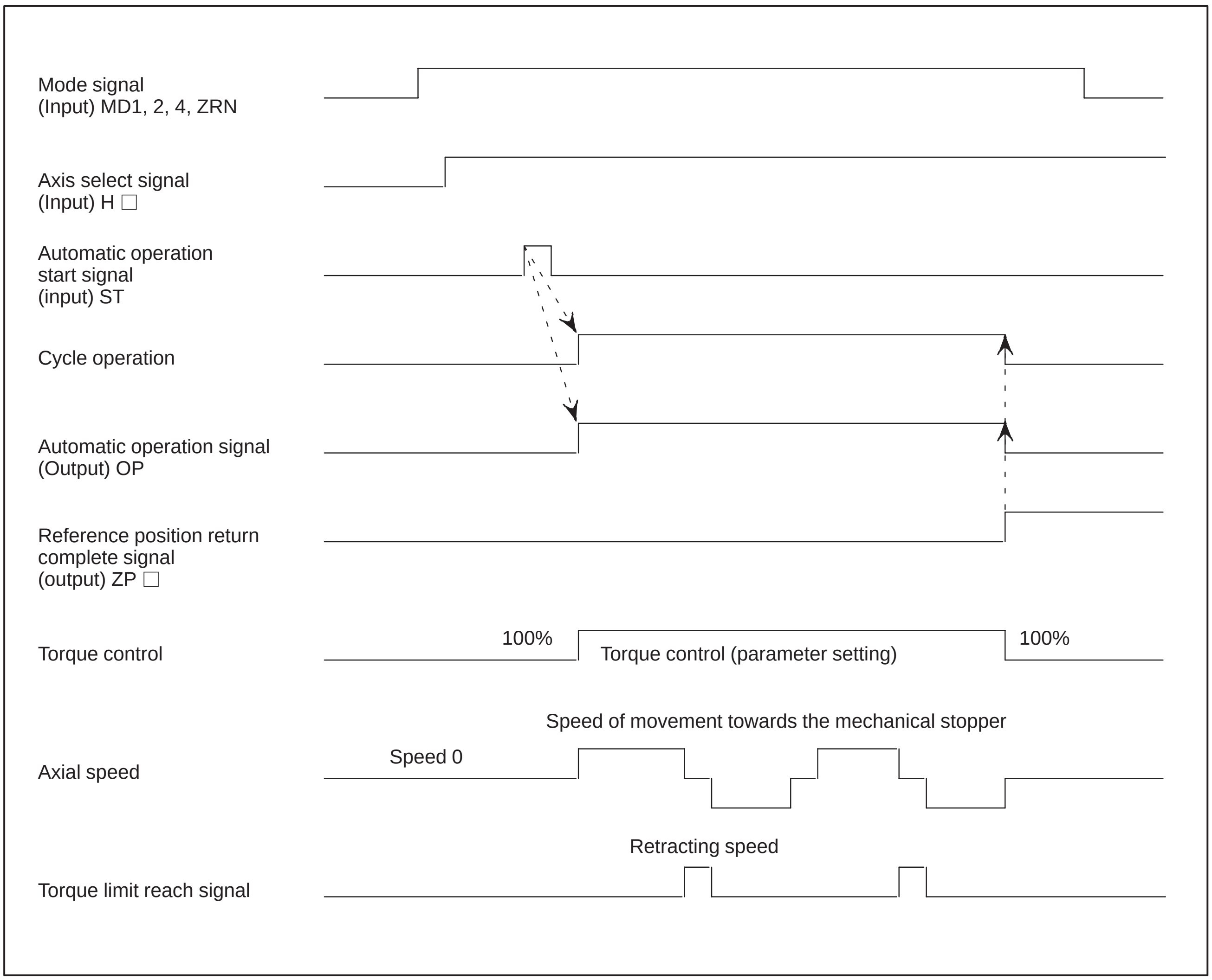
(4) The axis is moved in the direction specified in bits 0 to 3 of parameter 003 at the speed (low detection speed) specified in parameter 943 (M series) or parameter 458 (T series).



(5) When the torque limit reach signal sent from the servo motor goes on, the axis is assumed to be butted against the mechanical stopper. Movement is made in the opposite direction by the amount specified in parameters 944 to 947 (M series) or parameters 459 to 462 (T series) at the speed specified in parameters 944 to 947 (M series) or parameters 459 to 462 (T series). The resultant position is set as the reference position. The servo motor is released from torque control.



Time chart



Parameter

For reference position setting by butting the axis against the stopper, set the following six parameters:

(1) Direction of reference position

	#7	#6	#5	#4	#3	#2	#1	#0	
0003					ZM4	ZMZ	ZMY	ZMX	(M series)
0003					ZM4	ZM3	ZMZ	ZMX	(T series)

ZMx ZMx On the axis for which reference position setting is performed by butting the axis against the stopper, the reference position (mechanical stopper) is:

1 : In the negative direction.
0 : In the positive direction.

(2) Torque limit value

0360	TRQLTX	(M series)
0361	TRQLTY	(M series)
0362	TRQLTZ	(M series)
0363	TRQLT4	(M series)
0372	TRQLTX	(T series)
0373	TRQLTZ	(T series)
0374	TRQLT3	(T series)
0375	TRQLT4	(T series)

TRQLTx Set a torque limit value for each axis, to be applied for reference position setting by butting the axis against the stopper. The actual torque limit is calculated as follows:

$$\frac{\text{(Setting)}}{255} \times \text{(Maximum torque)}$$

When the setting is 0, torque control is not performed.
Valid data range: 0 to 255

(3) Butting speed 1

0942	STPSD1	(M series)
0451	STPSD1	(T series)

STPSD1 Set the speed at which the axis is butted against the mechanical stopper:

Setting	0 to 15000	Unit	1mm/min	(Millimeter output)
	0 to 6000	Unit	0.1inch/min	(Inch output)

(4) Butting speed 2

0943	STPSD2	(M series)
0452	STPSD2	(T series)

STPSD2 Set the speed at which the axis is butted against the mechanical stopper the second time:

Setting	0 to 15000	Unit	1mm/min	(Millimeter output)
	0 to 6000	Unit	0.1inch/min	(Inch output)

(5) Retracting speed (speed of travel for reference position setting)

0944	ZRNSPX	(M series)
0945	ZRNSPY	(M series)
0946	ZRNSPZ	(M series)
0947	ZRNSP4	(M series)
0459	ZRNSPX	(T series)
0460	ZRNSPZ	(T series)
0461	ZRNSP3	(T series)
0462	ZRNSP4	(T series)

ZRNSPx Set the speed at which the axis is retracted after being butted against the mechanical stopper.

Setting	0 to 15000	Unit	1mm/min	(Millimeter output)
	0 to 6000	Unit	0.1inch/min	(Inch output)

(6) Reference position

0948	SAZRNX	(M series)
0949	SAZRNY	(M series)
0950	SAZRNZ	(M series)
0951	SAZRN4	(M series)
0872	SAZRNX	(T series)
0873	SAZRNZ	(T series)
0874	SAZRN3	(T series)
0875	SAZRN4	(T series)

SAZRNx Set the position of the mechanical stopper, as viewed from the reference position set for each axis. The sign added to the data corresponds to the direction of the reference position (parameter 003). If the signs do not match, reference position setting by butting the axis against the mechanical stopper is not performed.

Setting	0 to ± 99999999	Unit	0.001mm/min	(Millimeter output)
			0.0001inch	(Inch output)

Caution

CAUTION

If a reset occurs during reference position setting, reference position setting is stopped. Torque control, however, continues to be applied to ensure safety. To release the current control, turn the power off.

Note

NOTE

1 This function is disabled for the fifth/sixth axis or for the seventh/eighth axis.
2 Two axes cannot be specified simultaneously.
3 Normal manual reference position return can be performed for some other axes.
4 The specified torque limit value must be higher than the traveling torque corresponding to the detection speed of the mechanical stopper.
  If the torque limit value is not greater than the traveling torque, the torque reach signal is enabled before the axis reaches the stopper. The position at which the torque reach signal goes on is assumed to be the position of the mechanical stopper.

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 in which memory (AUTO) mode 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 AUTO 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 starting
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 handle feed (HNDL), incremental feed (STEP), or jog feed (JOG).

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 manual data input mode (MDI) 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 “Signals” 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.

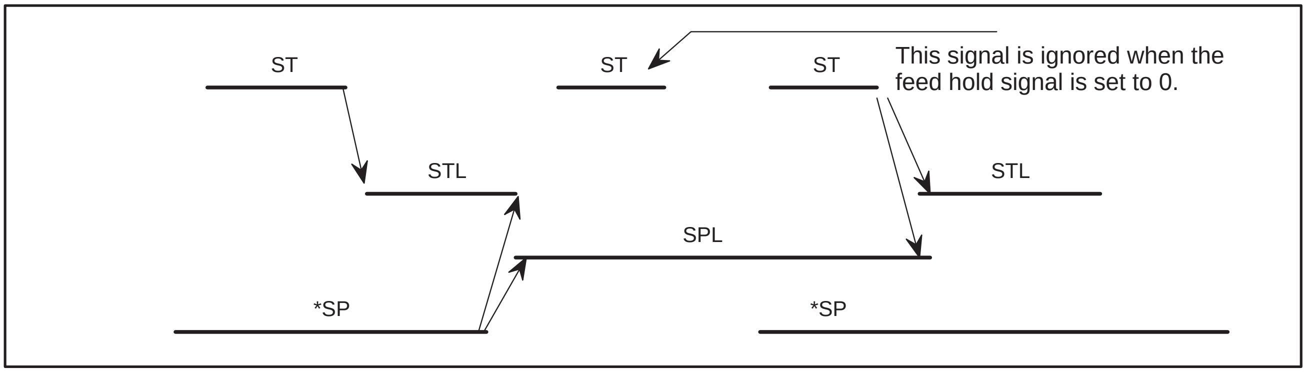


Fig. 5.1 Time Chart for Automatic Operation

When signal *SP is set to 0 during the execution of a block containing only the M, S or T 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 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 cannec 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.

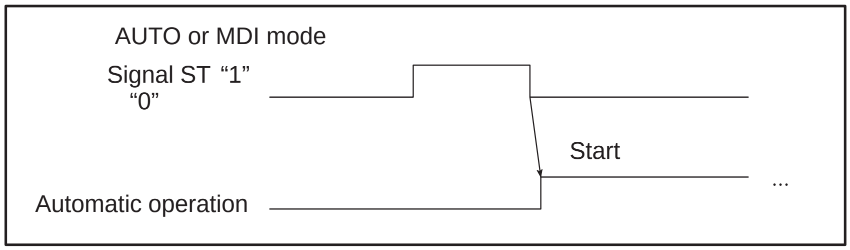
Signal

Cycle start signal ST<G120#2>

[Classification] Input signal

[Function] Starts automatic operation.

[Operation] When signal ST is set to 1 then 0 in memory (AUTO) mode or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.

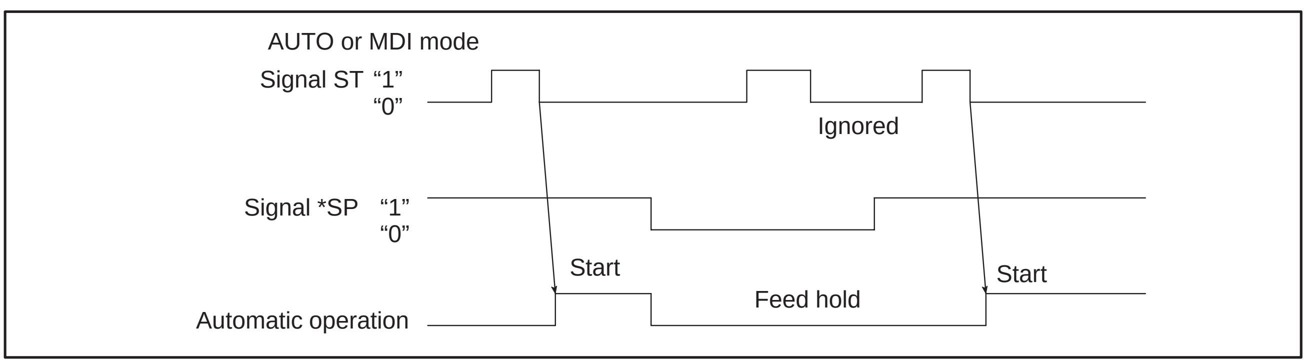


Feed hold signal *SP<G121#5>

[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<F148#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<F148#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<F148#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.

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

· When the reference position has not yet been established
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 (AUTO), the signal op turns to “1”.

Manual data input start signal DST<F150#5>

[Classification] Output signal

[Function] Informs when the [START] button on the MDI panel is pressed.

[Output condition] The signal becomes 1 when:

- The [START] button on the MDI panel is pressed.

The signal is 0 when:

- The [START] button on the MDI panel is not pressed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G120						ST		
G121			*SP					

	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP		STL	SPL				
F150			DST					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001						DCS		

[Data type] Bit

DCS Specifies how the [START] button on the MDI panel is used, as follows:

1 : The signal from the button does not passes the machine, so that manual data input is started directly by the NC.

0 : The signal from the button is sent to the machine as the manual data input start signal (DST, bit 5 of F150). If the machine returns a start signal, manual data input begins.

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 0700 and 0701).

Detailed information on the automatic operation stop or feed hold state can also be displayed (diagnosis number 0712).

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.4.1	MEMORY OPERATION
	III.4.2	MDI OPERATION
	III.7.2	CHECKING BY SELF–DIAGNOSTIC SCREEN
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.4.1	MEMORY OPERATION
	III.4.2	MDI OPERATION
	III.7.2	CHECKING BY SELF–DIAGNOSTIC SCREEN

5.2 RESET AND REWIND

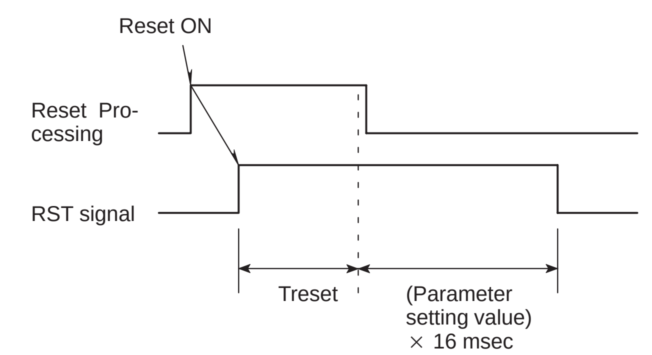
General

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.0252, has elapsed after the above conditions have been released.

RST signal output time = Treset (Reset processing time) + (parameter setting value) $\times$ 16 msec.



CAUTION
Treset 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, or T function, signal MF, SF, or TF is set to 0 within 100 ms.

Tool movement along the controlled axis is also decelerated and stopped(*1) in jog feed (JOG), manual handle feed (HNDL), or incremental feed (STEP).

CAUTION
(*1) When the emergency stop signal (*ESP) is set to 0, the tool is stopped by an emergency stop.

Bit 6 (CLER) of parameter No.0045 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 (MBCLR) of parameter No.0057
  Whether programs created in MDI mode are erased or stored
- Bit 6 (COMC) of parameter No.0040
  Whether custom macro variables #100 to #149 are cleared or stored
- Bit 7 (LOCC) of parameter No.0040
  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. In automatic operation mode, if the DNC operation selection signal (DNCI) is 1, and the selected input/output unit is connected with a portable tape reader, the tape on the 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 RWDOUT, bit 2 of parameter no. 0045, determines whether the rewinding–in–progress signal is output.

   When RWDOUT 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<G121#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<G104#6>

[Classification] Input signal

[Function] CNC is reset and a program under an automatic operation is rewound.

[Operation] As described in the item, “RESET AND REWIND”.

Resetting signal RST<F149#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 RESET key is pressed

This signal is set to 0 in the following case:
When the resetting signal output time, set with parameter No.0252, has elapsed after the above conditions have been released and the CNC is reset

Rewinding signal RWD<F164#6>

[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
G104		RRW						
G121	ERS							

	#7	#6	#5	#4	#3	#2	#1	#0
F149							RST	
F164		RWD						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0040	LOCC	COMC						

[Data type] Bit

COMC Custom macro’s common variables Nos. 100 to 149
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

LOCC Custom macro’s local variables Nos. 1 to 33
0 : Cleared to “vacant” by reset
1 : Not cleared by reset

		#7	#6	#5	#4	#3	#2	#1	#0
0045			CLER				RWDOUT		

[Data type] Bit

RWDOUT 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

CLER Reset key on the CRT/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
0057		MBCLR							

[Data type] Bit

MBCLR Whether a program prepared in the MDI mode is cleared by reset

0 : Not deleted

1 : deleted

0252	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

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 actually 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 is 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 <G117#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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G117							MLK	

Note

NOTE

1 **Automatic operation in the machine lock state (M, S, and T 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, and T 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 (ZPX 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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 (*OV1 to *OV8), 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	Maximum value on the manual feed feedrate dial
0	Feedrate determined using the override dial, or rapid traverse rate (*1)	Feedrate determined using the override dial

Rapid traverse rate . . . . Setting by parameter Nos.0518 to 0521

*1:If parameter RDRN (bit 6 of parameter No.0001) is 1, the feedrate determined using the override dial is used. If the parameter is 0, the rapid traverse rate is used.

Signal

Dry run signal DRN<G118#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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G118	DRN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001		RDRN						

[Data type] Bit

RDRN Dry run for rapid traverse command
0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0	
0065			TAPDRN						(M series)

[Data type] Bit

TAPDRN Dry run during tapping (tapping cycle G74 or G84; rigid tapping)
0 : Enabled
1 : Disabled

0518 – 0521	Rapid traverse 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	30 to 24000	6 to 12000
Inch machine	0.1 inch/min	30 to 9600	6 to 4800
Rotation axis	1 deg/min	30 to 24000	30 to 12000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

0527	Maximum cutting feedrate for all axes

[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

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.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.5.4	DRY RUN
OPERATOR’S MANUAL (For Lathe) (B–61394E)	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 (SBKM) of parameter No.0011, as follows:

SBKM = 0: Operation does not stop in the custom macro statements but stops once the next NC command has been executed.

SBKM = 1: Operation stops after each block in the custom macro statements.

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 (HND), incremental feed (STEP), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK<G116#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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G116							SBK	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0011			SBKM					

[Data type] Bit

SBKM Custom macro statement
0 : Not stop the single block
1 : Stops the single block

Caution

CAUTION
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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.5.5	SINGLE BLOCK
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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.

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.

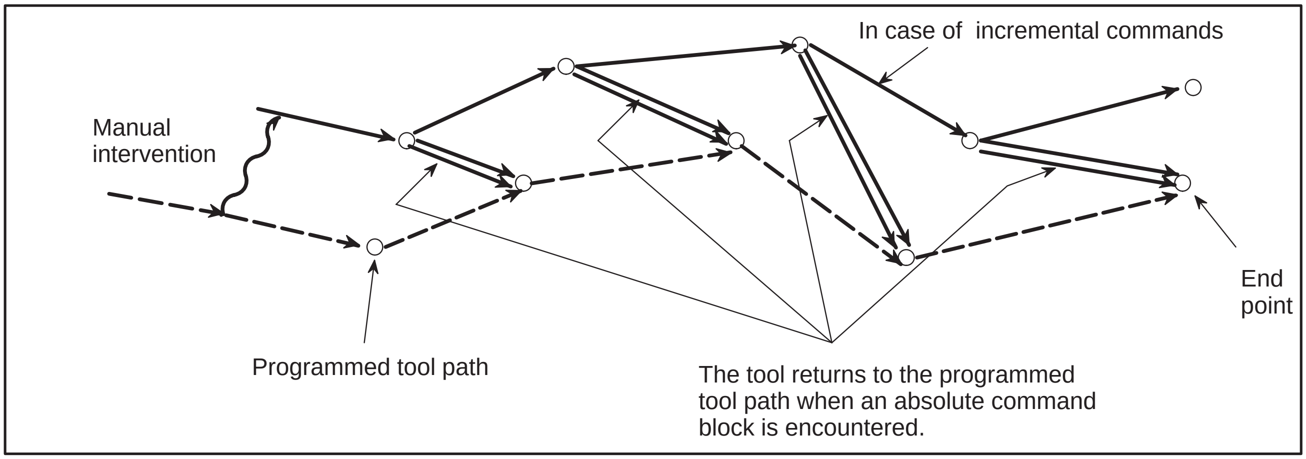


Fig. 5.4 (a) Manual absolute ON

> **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 current 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, when operation in the automatic operation mode, or MDI mode 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.

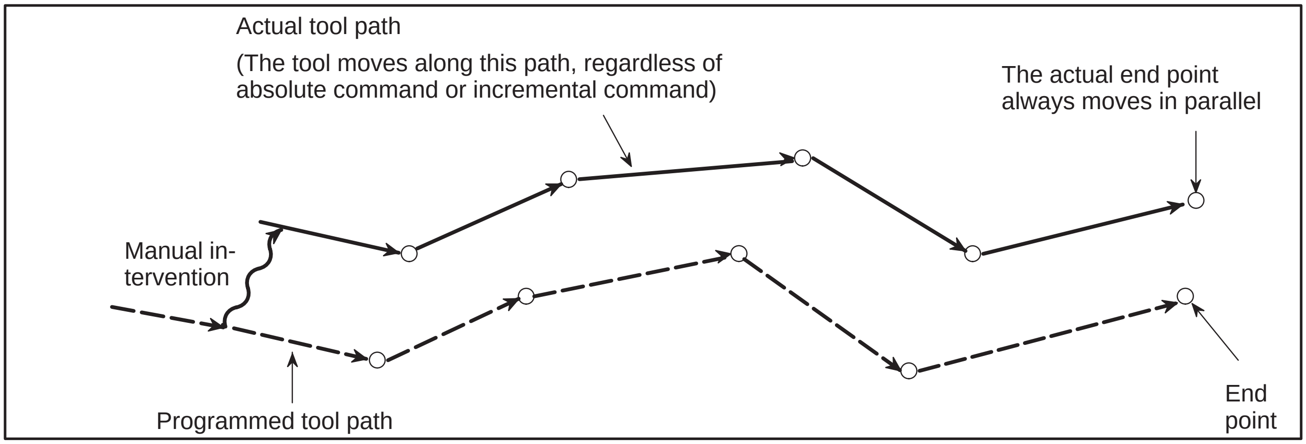


Fig. 5.4 (b) Manual absolute OFF

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<G127#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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G127						*ABSM		

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.3.5	MANUAL ABSOLUTE ON AND OFF
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.3.5	MANUAL ABSOLUTE ON AND 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).

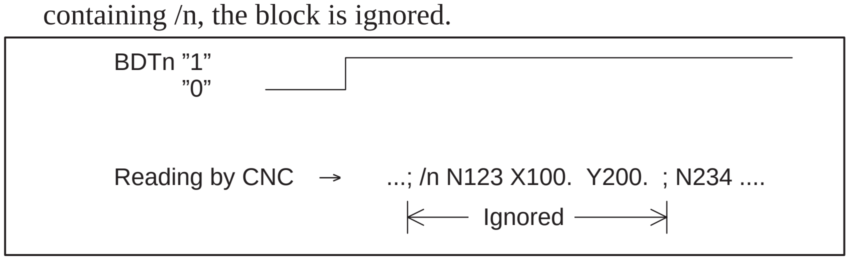
(Example) /2 N123 X100. Y200. ;

Input signal	Code specified at teh head of a block
BDT1	/ or /1 (Note)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

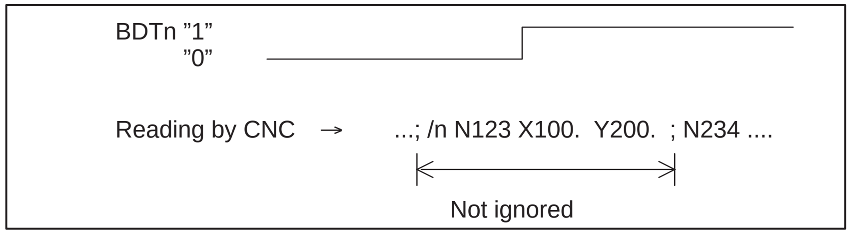
NOTE
Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.
(Example)
//3 N123 X100. Y200. ; — Invalid
/1 /3 N123 X100. Y200. ; — Valid

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:

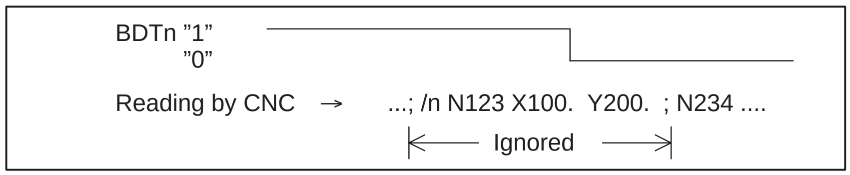
1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.



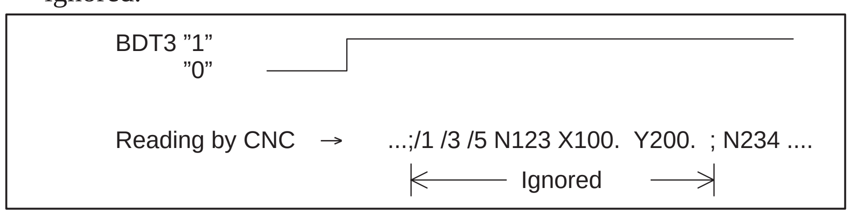
2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.



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 <G116#0> BDT2 to BDT9 <G141>

[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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G116								BDT1
G141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.12.2	PROGRAM SECTION CONFIGU-RATION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.12.2	PROGRAM SECTION CONFIGU-RATION

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.

Setting data

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.11.5.4	Sequence Number Comparison and Stop
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.11.5.2	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<G103#0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to logical 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 logical 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 Nos. 124 to 127. When the tool is set to the restart point, machining restarts.

Program restart under way signal SRNMV<F188#4>

[Classification] Output signal

[Function] Notifies that return operation at program restart is in progress.

[Output condition] The signal is output during return operation at program restart.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G003								SRN
F188				SRNMV				

Parameter

0124–0127	Movement sequence to program restart position

[Data type] Byte

[Valid data range] 1 to 4

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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.4.4	PROGRAM RESTART
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.4.5	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 till G63 (tapping mode), G62 (automatic corner override), or G64 (cutting mode), is commanded.

• Tapping Mode (G63)

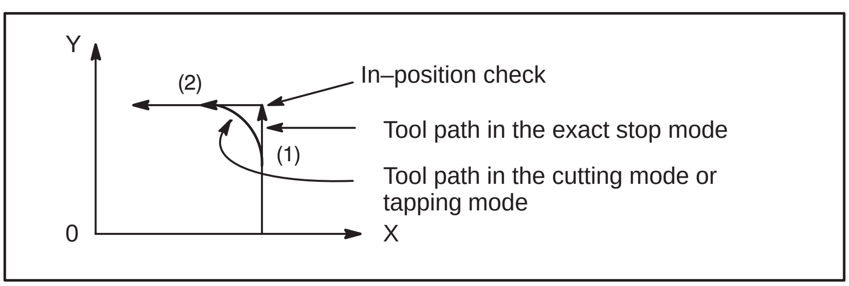
When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. This G63 is valid till G61 (exact stop mode), G62 (automatic cornter override), or G64 (cutting mode) 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 till G61 (exact stop mode) or G62 (automatic corner override) 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 Subsec. 7.2.6.1 and 7.2.6.2 for details.

(Example) Tool paths from block (1) to block (2)



Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)

5.9 BALANCE CUT (0–TTC)

General

When a thin workpiece is to be machined as shown in fig. 5.9, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously;this function can prevent the workpiece from distortion that results when only one side is machined at a time. When both sides are machined at the same time, the movement of one tool must synchronize with that of the other tool. Otherwise, the workpiece may vibrate, resulting in poor machining. With this function, the movement of one tool post can easily synchronize with that of the other tool post.

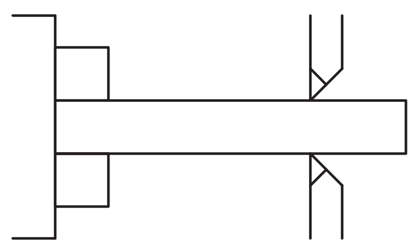


Fig. 5.9 Balance cut

Alarm and message

No.	Message	Contents
163	COMMAND G68/G69 INDEPENDENTLY (0–TTC)	G68 and G69 are not independently commanded in balance cut. Correct program.

Caution

CAUTION

1. If feed hold operation is performed during balance cutting using both tool posts, balance cut processing is not performed at restart time, it is performed when the next move command is specified for both tool posts.
2. Balance cutting is not performed in dry run or machine lock state.
3. When rapid traverse operation is specified, balance cut processing is not performed.
4. A workpiece for which thread cutting has been performed in the balance cut mode cannot be subjected to thread cutting in the cancel mode. Thread cutting starts at a different position.
5. Balance cut only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the data for both tool posts, such as the travel distance and feedrate, must be the same.

Note

NOTE

1. Time delay before the pulse distribution of both tool posts is started is 2 ms or shorter.
2. In the balance cut mode, synchronization is established at the start of a move block, so movement may momentarily stop.
3. The cancel mode (G69) is set by a reset.
4. When the option ”mirror image for double turrets” is selected, the balance cut function cannot be used.

Reference item

OPERATOR’S MANUAL (For Lathe) (B–61394E)	II. 24.4	BALANCE CUT (G68, G69)

5.10 DNC OPERATION

General

By activating automatic operation during the DNC operation mode, it is possible to perform machining (DNC operation) while a program is being read in via the reader/puncher interface, or remote buffer.
If the floppy cassette directory display option is available, 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/punch interface, and remote buffer in advance.

Signal

DNC operation select signal DNCI<G127#5>

[Classification] Input signal

[Function] Selects the DNC operation mode.
To select the DNC operation mode, it is necessary to select the automatic operation mode (AUTO) and set the DNC operation select signal to logical 1 simultaneously.

[Operation] When the DNC operation select signal becomes logical 1, the control unit operates as follows:

– If the automatic operation mode (AUTO) has not been selected, the signal is ignored, and nothing happens.
– If the automatic operation mode (AUTO) has been selected, the DNC operation mode is selected, and DNC operation becomes possible.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G127			DNCI					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0390	NODC3							

[Data type] Bit

NODC3 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 NODC3=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.

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/M199	M198 and M99 are executed in the schedule operation. Or M198 is executed in the DNC operation.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.4.3	DNC OPERATION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.4.3	DNC OPERATION
CONNECTION MANUAL (This manual)	13.1	READER/PUNCHER INTERFACE
	13.2	REMOTE BUFFER

5.11 REMOTE BUFFER DI/DO SIGNALS

General

Eight input signals and eight output signals are provided to interface the PMC with the host CPU. The host manages the signals by means of a predetermined method.

[Transmission]

SDI (SEND DI) PMC → Host CPU
CS SDI DA E

When the host CPU issues a DI read request, or RDI, the contents of the DI signals (RMTDI0 to RMTDI7) are sent to the host CPU.

CS: Checksum data
DA: Two hexadecimal bytes representing the eight DI bits
E: End code

[Reception]

SDO (SEND DO) PMC ← Host CPU
CS SDO DA E

Upon receiving GDT, SAT, or SDI, the host CPU can issue this command. The command causes the contents of the DO signals (RMTDO0 to RMTDO7) to be output to the PMC.

CS: Checksum data
DA: Two hexadecimal bytes representing the eight DO bits
E: End code

NOTE
This function is added to protocol A of the remote buffer. For details of the controlling method, refer to Descriptions (B–61392EN–1) for remote buffer.

• Protocol

The following two protocols are supported to enable communication between the host computer and remote buffer. Either protocol is selected according to the setting of the corresponding parameter.

(1) Protocol A
Handshaking. Transmission and reception are repeated between the host computer and remote buffer.

(2) Protocol B
The communication between the host computer and remote buffer is controlled according to the control code output from the remote buffer.

• Protocol A

Protocol A is provided for handshaking, in which transmission and reception are repeated between the remote buffer and host computer.

• Signals

For an explanation of each signal, refer to Descriptions (B–61392EN–1) for remote buffer.

• Communication system

For details of the communication system, refer to Descriptions (B–61392EN–1) for remote buffer.

- **Message format**

For details of the message format, refer to ”FANUC REMOTE BUFFER Descriptions (B–90699).”

Signal

(Output) RMTDO0 to RMTDO7 <F289> (T series), <F290> (M series)

	#7	#6	#5	#4	#3	#2	#1	#0	
F289	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0	(T series)
F290	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0	(M series)

(Input) RMTDI0 to RMTDI7 <G239>

	#7	#6	#5	#4	#3	#2	#1	#0
G239	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0

5.12 IN–FEED CONTROL

Signal

In–feed control cut–in start signal (0–GSC) INFD<G237#6>

[Classification] Input signal

[Function] When this signal is turned on, the tool moves through a specified cut–in amount along a programmed figure.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G237		INFD						

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 rapid traverse rate in the G00 command is set to the parameter (Nos.0518 to 0521 for 1st axis to 4th axis, Nos. 7518 and 7519 for 5th axis and 6th axis, Nos. 0643 and 0644 for 7th axis and 8th axis) 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

0518	Rapid traverse rate for 1st axis
0519	Rapid traverse rate for 2nd axis
0520	Rapid traverse rate for 3rd axis
0521	Rapid traverse rate for 4th axis
0643	Rapid traverse rate for 7th axis
0644	Rapid traverse rate for 8th axis
7518	Rapid traverse rate for 9th axis
7519	Rapid traverse rate for 10th 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 to 24000	30 to 12000
Inch machine	0.1 inch/min	30 to 9600	30 to 4800
Rotaion axis	1 deg/min	30 to 24000	30 to 12000

[Description] Set a rapid traverse rate for each axis.

NOTE
By setting the FML 10 bit (bit 7 of parameter 0049) to 1, the units of data can be multiplied by ten. Doing so causes the maximum value to change as follows:
IS–A, IS–B: 100 [m/min] (4000 [inch/min])
IS–C: 12 [m/min] (480 [inch/min])

Note

NOTE
The rapid traverse rate cannot be specified in the address F.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.4.1	POSITIONING (G00)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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.

G01α_α_β_β_γ_γ_ζ_ζ_ $F\underline{f}$;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of β axis direction : $F_\beta = \frac{\beta}{L} \times f$

Feed rate of γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

Feed rate of ζ axis direction : $F_\zeta = \frac{\zeta}{L} \times f$

$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$$

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 axis β (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the α and β 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 X20.0B40.0 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

$$\frac{40 \text{ deg}}{0.14907 \text{ min}} \doteq 268.3 \text{ deg/min}$$

In simultaneous 3 axes control, the feed rate is calculated the same way as in 2 axes control.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0011		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)

0527	Maximium cutting feedrate for all axes

[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

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.

0549	Cutting feedrate when the power is turned on	(M series)

[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

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.

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

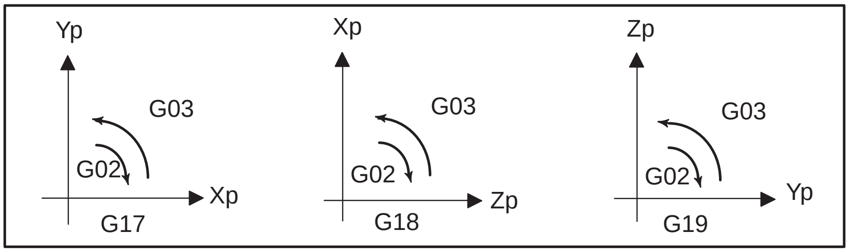
OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.4.3	LINEAR INTERPOLATION (G01)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.4.2	LINEAR INTERPOLATION (G01)

6.3 CIRCULAR INTERPOLATION

General

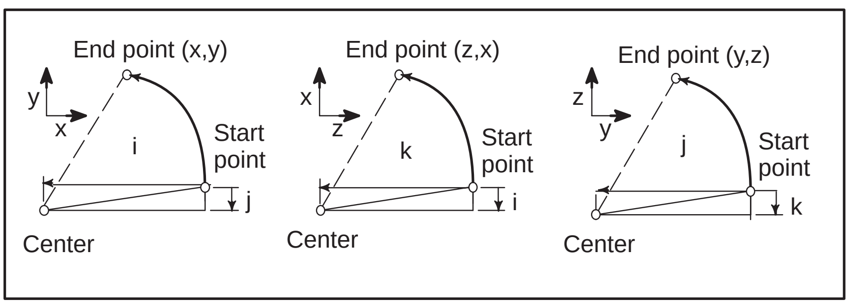
The command below can move a tool along a circular arc in the defined plane.

“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.



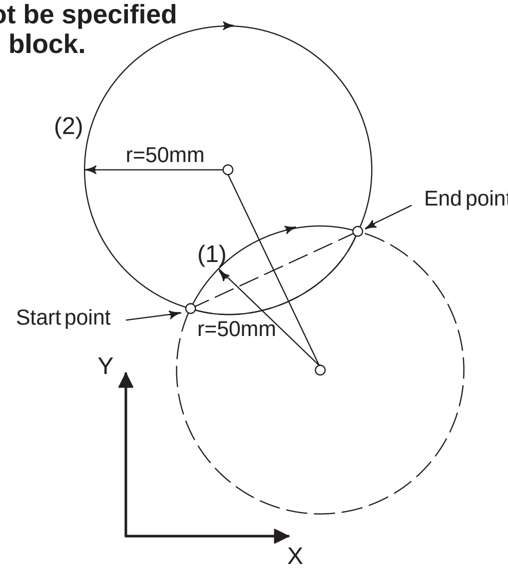
I0,J0, and K0 can be omitted. When $X_{p,}$ Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02I; Command for a circle
If, when the RADCHK bit (bit 6 of parameter 0393) is set to 1, the difference in the radius between the start and end points exceeds the value set in parameter 0876, alarm No. 020 is issued.

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.

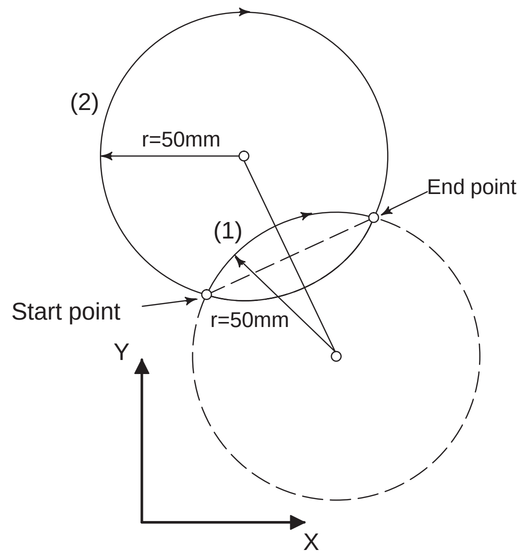
For T series, an arc with a sector angle of 180° or wider cannot be specified (Alarm 023).
For M series, specify an arc more than 180° with a negative radius value commanded.

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 <u>R50.0</u> 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 <u>R50.0</u> F300.0 ;
For arc (2)**(greater than 180°)**
G91 G02 X_P60.0 Y_P20.0 <u>R–50.0</u> F300.0 ;



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

0212	Plane selected at power–up	(M series)

[Data type] None

[Valid data range] 0 to 2

[Description] Set one of the following as the plane that will be selected at power–up:

Setting value	Selected plane
0	X – Y plane (G17)
1	Z – X plane (G18)
2	Y – Z plane (G19)

0279	Setting of 3rd axis in the basic coordinate system	(T series)
	Setting of 4th axis in the basic coordinate system	(M series)
0280	Setting of 4rd axis in the basic coordinate system	(T series)

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

[Data type] Byte

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.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Set value	Meaning
2	Y axis of the basic three axes (Only for T series)
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
0393		RADCHK						

RADCHK When a circular interpolation command is specified, the difference in the radius between the start and end points is:
1 : Checked.
0 : Not checked.

0876	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.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COM-MANDED	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.
023	ILLEGAL RADIUS COM-MAND (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 SE-LECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.

Note

NOTE

1. The U, V and W axes (parallel with the basic axis) can be used with G–code system B and C. (T series)
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 alarm (No.028)is displayed.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.4.4	CIRCULAR INTERPOLATION (G02,G03)
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)

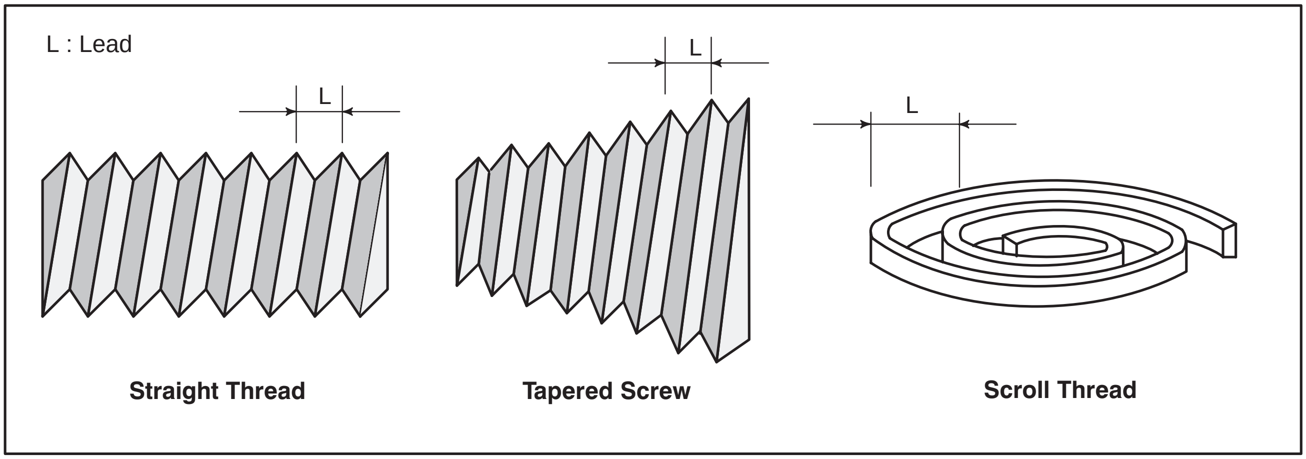
6.4 THREAD CUTTING

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<F188#3>(T series)

[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 are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F188					THRD			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0024						SCTO		

[Data type] Bit

SCTO The spindle speed arrival signal (G120#4 SAR) is:
0 : Not checked
1 : Checked

	#7	#6	#5	#4	#3	#2	#1	#0	
0065			G92ZAX						(T series)

G92ZAX Parameters for the Z–axis time constant and acceleration/deceleration lower limit (FL) in threading (G92):
0 : Parameters common to all axes
1 : Parameters 0627 and 0628

0526	Time constant of threading (G92) for the X–axis	(T series)

[Data type] Word

[Unit of data] msec

[Valid data range] 1 to 4000

[Description] Set a time constant of threading (G92) for the X–axis.

> **NOTE**
> Set an optimum value, considering parameter 0528.

● Setting the time constant for the threading cycle

0627	Time constant of exponetial acceleration/deceleration in the thread cutting cycle for Z axis	(T series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G92) for Z axis.

- **Setting the FL feedrate for the thread cutting cycle**

0528	FL rate of acceleration/deceleration in the thread cutting cycle for X axis	(T series)

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the FL rate of acceleration/deceleration in the thread cutting cycle (G92) for X axis.

NOTE
Set an optimum value, considering parameter 0526.

0628	FL rate of exponential acceleration/deceleration in the thread cutting cycle for Z axis	(T series)

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the lower limit (FL rate) of acceleration/deceleration in the thread cutting cycle (G92) for each axis.

- **Setting the chamfering distance for the thread cutting cycle**

0109	Chamfering distance in the thread cutting cycles G76 and G92	(T series)

[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**

0725	Minimium depth of cut in the multiple repetitive canned cycle G76	(T series)

[Data type] Two–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 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**

0726	Finishing allowance in the multiple repetitive canned cycle G76	(T series)

[Data type] Two–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 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**

0723	Repetition count of final finishing in the multiple repetitive canned cycle G76	(T series)

[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**

0724	Tool nose angle in the multiple repetitive canned cycle G76	(T series)

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0, 29, 30, 55, 60, 80

This parameter sets the tool nose angle in the multiple repetitive canned cycle G76.

Warning

- **Warnings applicable to both the M and T series**

WARNING

1. Feedrate override is ignored during thread cutting, 100% being assumed.
2. During threading, spindle override is ignored, 100% being assumed.
3. It is very dangerous to stop feeding the thread cutter without stopping the spindle. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If the feed hold button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is executed as if the SINGLE BLOCK button were pushed. However, the feed hold lamp (SPL lamp) lights when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is turned off (Single Block stop status).
4. 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.
5. When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.
6. 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.

- **Warning applicable to the T series only**

WARNING

The thread cutting retract function is supported only for the threading cycle.

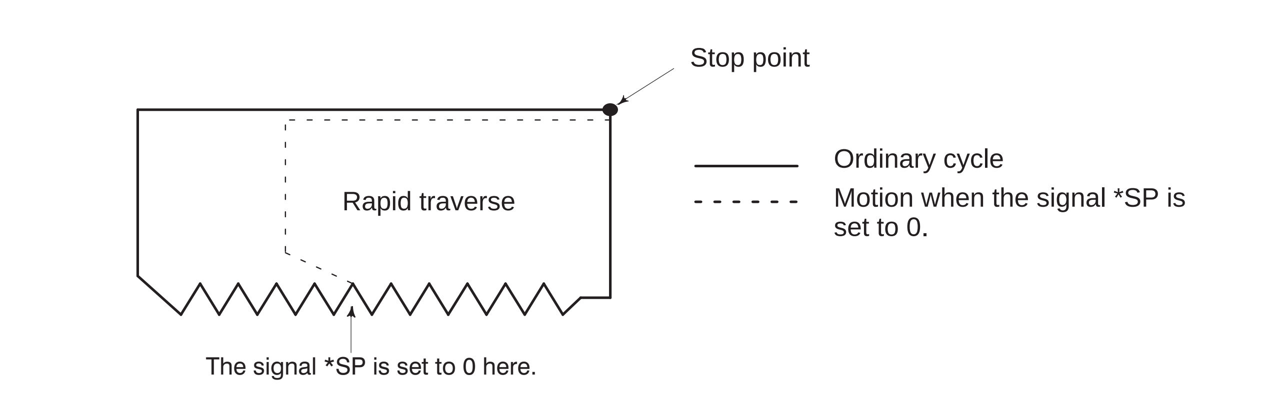
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.4.7	CONSTANT LEAD THREADING
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.4.6 II.4.8 II.13.1.2 II.13.2.7	CONSTANT LEAD THREADING CONTINUOUS THREAD CUTTING Thread Cutting Cycle Multiple Thread Cutting Cycle

6.4.2 Thread Cutting Cycle Retract (T series)

General

When the automatic operation stop signal *SP <G121#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
0394							TFHOVR	

[Data type] Bit

TFHOVR 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**

0109	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.

Warning

WARNING
While the tool is retracting, automatic operation stop signal *SP <G121#5> is ignored.

Note

NOTE
The chamfering distance for retraction is determined by the setting of parameter No.0109.

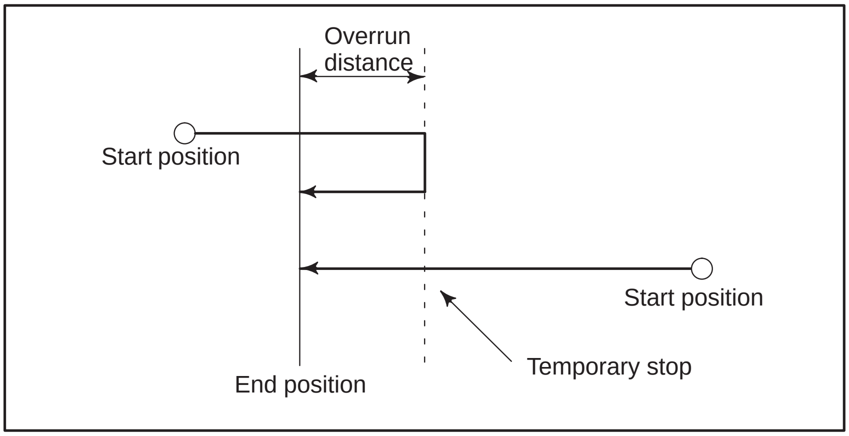
Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.13.1.2	Thread Cutting Cycle
	II.13.2.7	Multiple Thread Cutting Cycle

6.5 SINGLE DIRECTION POSITIONING (M series)

General

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



An overrun and a positioning direction are set by the parameter (Nos.0204 – 0207). Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0029					G604	G60Z	G60Y	G60X

G60x The approach direction for each axis in unidirectional positioning (G60) is:
0 : Positive.
1 : Negative.

0204	Approach distance along the X–axis in unidirectional positioning
0205	Approach distance along the Y–axis in unidirectional positioning
0206	Approach distance along the Z–axis in unidirectional positioning
0207	Approach distance along the fourth–axis in unidirectional positioning

[Unit of data]

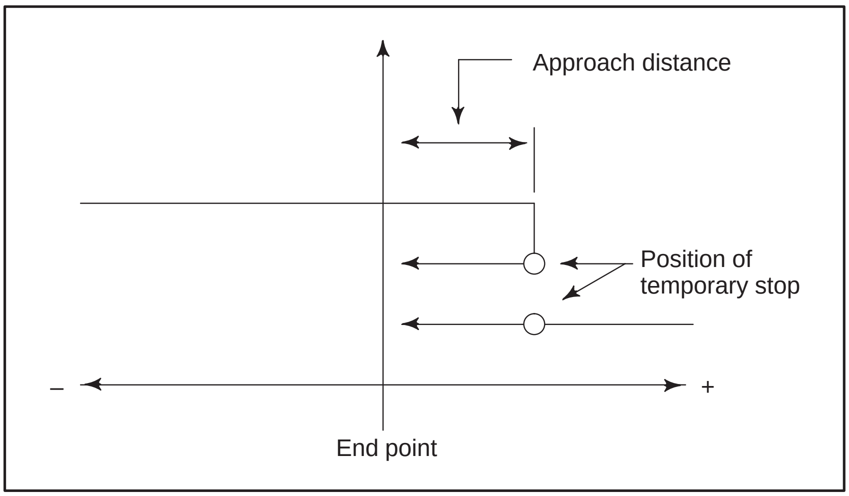
Unit of data	IS–A	IS–B	IS–C
Millimeter machine [mm]	0.1	0.01	0.01
Inch machine [inch]	0.01	0.001	0.001
Rotation axis [deg]	0.1	0.01	0.01

[Valid data range] 0 to 255 (For IS–C, the maximum value is 163.)

[Description] Set an approach distance for unidirectional positioning (G60) for each axis.

NOTE
Set the approach direction for each axis in bits G60X to G604 (bits 0 to 3 of parameter 0029).

Example Unidirectional positioning when the approach direction is negative



Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.4.2	SINGLE DIRECTION POSITION-ING

6.6 HELICAL INTERPOLATION (M SERIES)

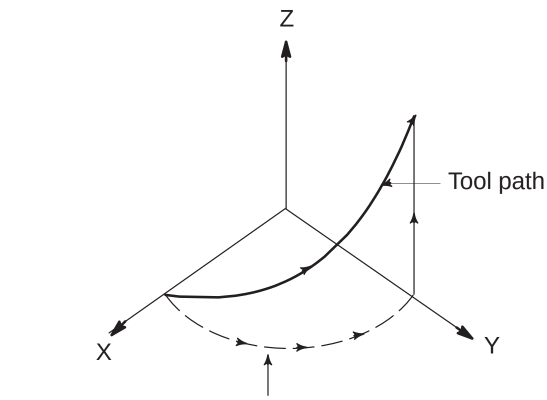
General

Helical interpolation which moved helically 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 is 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
0393								HFC

HFC The feedrate for helical interpolation is:

0: The tangential velocity of circular interpolation is clamped to the value specified in parameter 0527. The speed along a linear axis is calculated as follows:

$$\text{Actual circular interpolation speed} \times \frac{\text{Segment length}}{\text{Arc length}}$$

1: The tangential velocity of circular interpolation is clamped to the value specified in parameter 0527.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.4.5	HELICAL INTERPOLATION

6.7 POLAR COORDINATE INTERPOLATION (T SERIES)

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.

G112 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.7). Polar coordinate interpolation is performed on this plane.

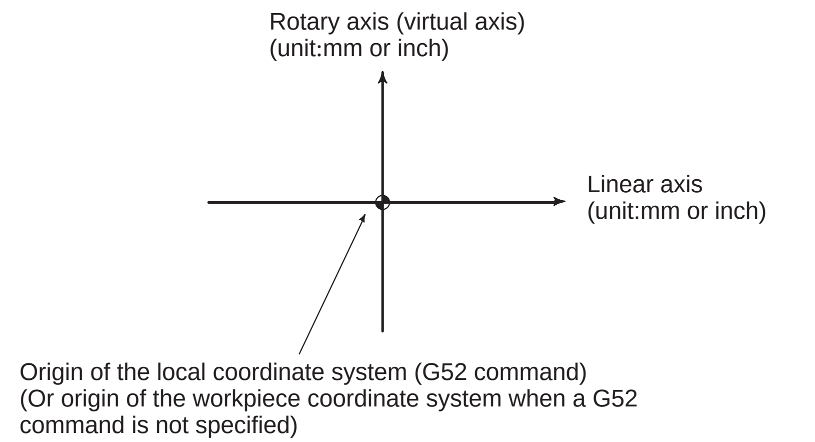


Fig. 6.7 Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G113).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 0291 and 0292) beforehand.

Parameter

0291	Axis (linear axis) specification for polar coordinate interpolation
0292	Axis (rotary axis) specification for polar coordinate interpolarion

[Data type] Byte

[Valid data range] 1 to 4

These parameters set control axis numbers of linear and rotary axes to execute polar interpolation.

0527	Maximum cutting feedrate for all axes

[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

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.

0663	Maximum cutting feedrate during polar coordinate interpolation

[Data type] Word

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0, 6 to 15000	0, 6 to 12000
Inch machine	0.1 inch/min	0, 6 to 6000	0, 6 to 4800
Rotation axis	1 deg/min	0, 6 to 15000	0, 6 to 12000

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 0527.

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, G112/G113 was specified. 2) An error is found in the plane selection. Parameters No. 0291 and No. 0292 are incorrectly specified. Modify the value of program or parameter.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.4.4	POLAR COORDINATE INTERPOLATION (G112, G113)

6.8 CYLINDRICAL INTERPOLATION

General

The amount of travel of a rotary axis specified by an angle is once 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.

Only one rotation axis can be set for cylindrical interpolation.

Parameter

0279	Attribute of 4th axis	(M series)

[Valid data range] 5 to 7

[Description] Set the axis in the basic coordinate system for the 4th axis.

Setting value	Attribute
5	Axis parallel to X axis
6	Axis parallel to Y axis
7	Axis parallel to Z axis

0279	Attribute of 3rd axis	(T series)
0280	Attribute of 4th axis	(T series)

[Valid data range] 2, 5 to 7

[Description] Set the axis in the basic coordinate system for the 3rd axis or 4th axis.

Setting value	Attribute
2	Y axis of basic 3 axis
5	Axis parallel to X axis
6	Axis parallel to Y axis
7	Axis parallel to 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 “G107 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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.4.6	CYLINDRICAL INTERPOLATION (G07.1)
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.4.5	CYLINDRICAL INTERPOLATION (G07.1)

6.9 POLYGONAL TURNING (T SERIES)

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

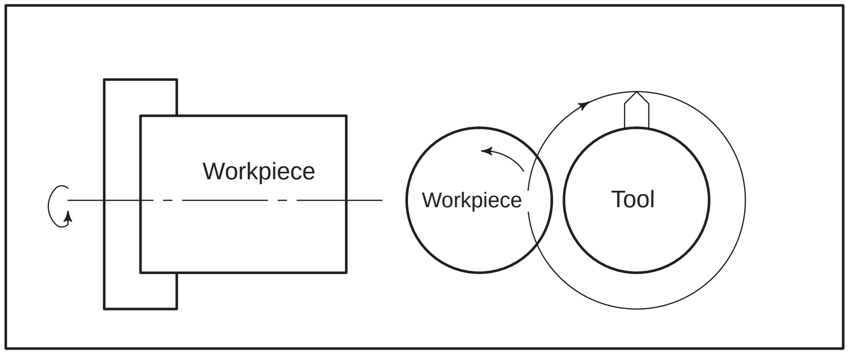


Fig. 6.9 (a) Polygonal turning

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.

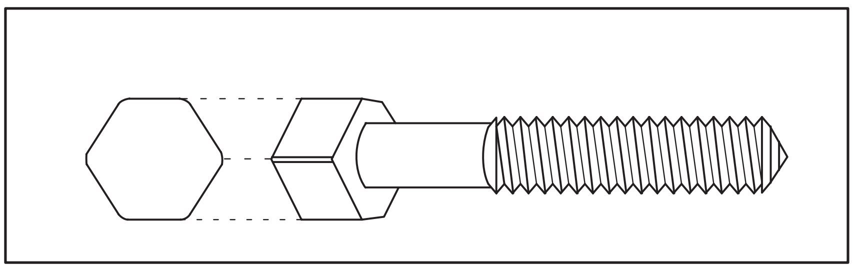


Fig. 6.9 (b) Hexagon bolt

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 Chapter 20, Part II of the “Operator’s Manual (For Lathe).”)

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).

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**

Bits #0 to #3 of parameter No. 0069 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.

A move command such as Y_ cannot be specified for the servo axis specified as a tool rotation axis, unlike the other controlled axes.

Only the command for reference position return, G28 V0; can be specified. See Chapter II.21 in the ”FANUC Series 0/00/0–Mate for Lathe Operator’s Manual” and the examples given below:

- **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 PLGN4 (bit 1 of No.0069) =1

- 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. 0103 = 2 (CMR)
  Parameter No. 0573 = 3600 (reference counter capacity)
  Parameter No. 8484 = 36 (DMR numerator)
  Parameter No. 8485 = 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 minimum command increment specified here is provided for the polygon axis only. This is determined irrespective of the ISA/ISC setting. However, IS–B should be specified.

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)

Angle to rotate through per tool axis rotation $= \frac{360}{0.01} = 36000$ (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 rpm, and the servo motor is directly connected to the servo motor:

Upper limit to the tool rotation axis speed = 2000 × 1 = 2000 (rpm)

This means the parameters must be set as follows:

No. 0778 = 36000 (angle to rotate through per tool axis rotation)
No. 0667 = 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 rpm, for example, specify as follows:

No. 0521 = 72000 (= $2000 \times \frac{360}{10}$)

Also specify other feedrates in 10 degrees/min units.

Signal

Polygon synchronization under way signal PSYN <F160#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 (G251) and stays at "1" during the polygonal turning mode.

The signal is reset to logical "0" by the polygon turning mode reset command (G250) 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 Chapter II–21 in operator's manual (B–61394E) for lathe.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
F160	PSYN								(T series)

Parameter

0067	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)	(T series)

[Data type] Word

[Unit of data] rpm

[Valid data range] For polygonal turning using servo motors:

$$\frac{0 \text{ to } 1.2 \times 10^8}{\text{set value of the parameter No. 0778}}$$

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 set upper–limit rotation speed during polygon turning. The spindle and tool rotation axis go out of synchronization when the rotation speed is clamped.

	#7	#6	#5	#4	#3	#2	#1	#0	
0069	PLHZ				PLG8	PLG7	PLGN4	PLGN3	(T series)

[Data type] Bit

PLGN3, PLGN4, PLG7, PLG8 As a synchronous axis used for polygonal turning, the third, fourth, seventh, or eighth axis is:
0: Not used.
1: Used.

> **NOTE**
> Set any one axis.

PLHZ 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.

0778	Movement of tool rotation axis per revolution	(T series)

[Data type] Two–word

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.

Alarm and message

Number	Message	Description
217	DUPLICATE G251 (COMMANDS)	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 G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G251 and 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 G251, rotate the spindle. If it is not rotating when the G251 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

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.21	POLYGONAL TURNING

6.10 NORMAL DIRECTION CONTROL (M SERIES)

General

When a tool with a rotation axis (4th axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the 4th axis is always perpendicular to the tool path (Fig. 6.10).

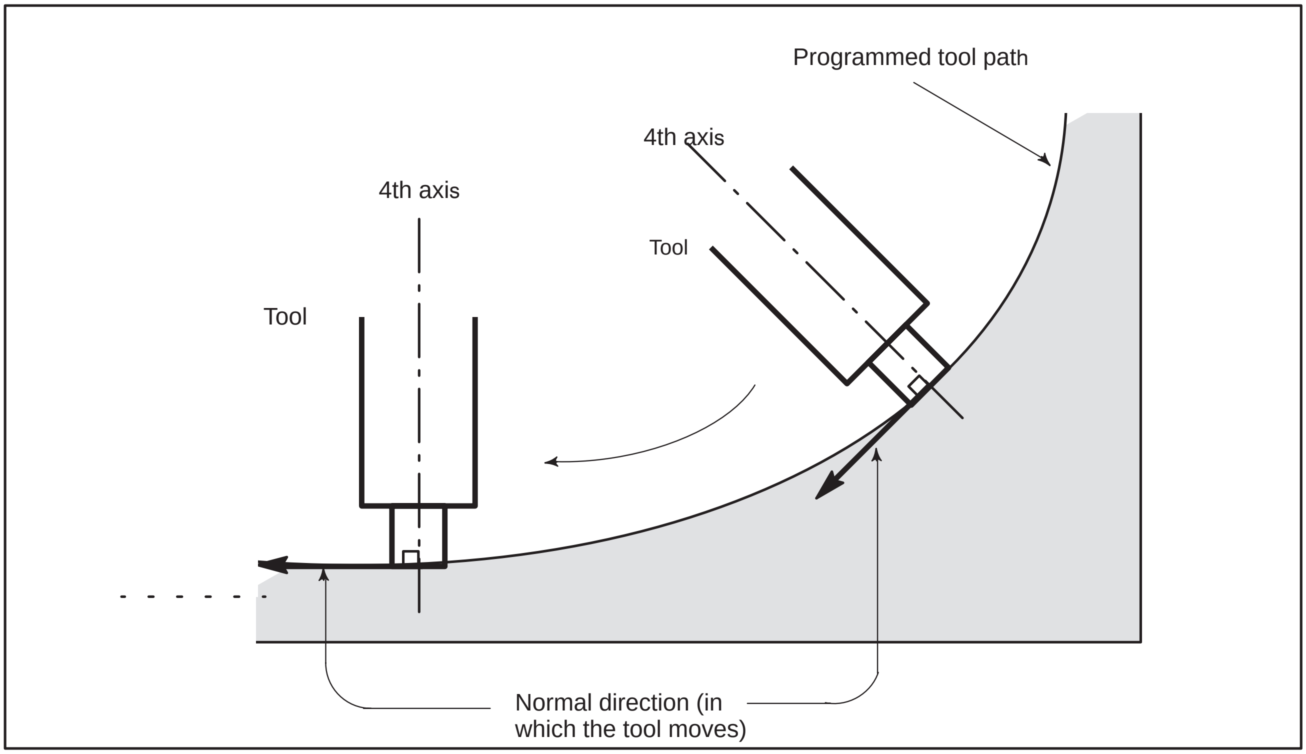


Fig. 6.10 Sample Movement of the tool

Movement of the 4th axis 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 4th axis exceeds the maximum cutting feedrate of the 4th axis specified to parameter No.0527, the federate of each of the other axes is clamped to keep the federate of the 4th axis below the maximum cutting feedrate of the 4th axis.

Parameter

0527	Maximum cutting feedrate for all axes

[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

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.

0683	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.

0832	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.

0833	Limit value of movement that is executed at the normal direction angle of a preceding block

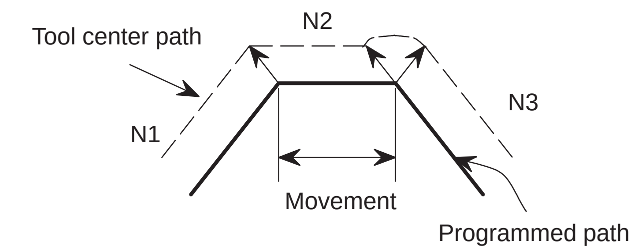
[Data type] Two–word

[Unit of data]

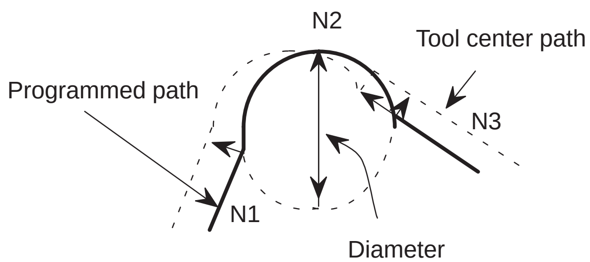
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[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.

Note

> **NOTE**
> The helical interpolation option is required to use this function. Helical interpolation cannot be specified in the normal direction control mode.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.14.11	NORMAL DIRECTION CONTROL

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 Nos.0518 – 0521).
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 switch on the machine operator's panel.

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 MTB (in–position check).
A rapid traverse rate is set for each axis by parameter Nos.0518 – 0521, so no rapid traverse rate need be programmed.
The following overrides can be applied to a rapid traverse rate using the switch on the machine operator's panel :F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for all axes by parameter No.0533.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0001		RDRN						

[Data type] Bit

RDRN Dry run for rapid traverse command
0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
0393			STOV0					

[Data type] Bit

STOV0 When cutting feedrate override is 0% during rapid traverse,
0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

0518	Rapid traverse for X axis
0519	Rapid traverse for Y axis
0520	Rapid traverse for Z axis
0521	Rapid traverse for 4th axis
0643	Rapid traverse for 7th axis
0644	Rapid traverse for 8th axis
7518	Rapid traverse for 5th axis
7519	Rapid traverse for 6th 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 to 24000	30 to 12000
Inch machine	0.1 inch/min	30 to 9600	30 to 4800
Rotaion axis	1 deg/min	30 to 24000	30 to 12000

Set the rapid traverse rate for each axis.

NOTE
By setting the FML 10 bit (bit 7 of parameter 0049) to 1, the units of data can be multiplied by ten. Doing so causes the maximum value to change as follows:
IS–A, IS–B: 100 [m/min] (4000 [inch/min])
IS–C: 12 [m/min] (480 [inch/min])

0559 – 0562	Manual rapid traverse rate

[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	30 to 24000	30 to 12000
Inch machine	0.1 inch/min	30 to 9600	30 to 4800
Rotaion axis	1 deg/min	30 to 24000	30 to 12000

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 No.0518 – 0521 is assumed.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.2	RAPID TRAVERSE
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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.0527. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Parameter

0527	Maximum cutting feedrate for all axes

[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

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

NOTE

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.3	CUTTING FEED
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 to be directly 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.
Refer to manuals of machine tool builder for details.

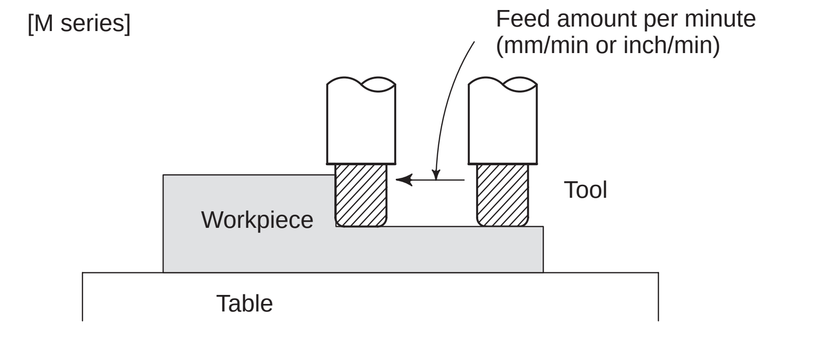


Fig. 7.1.3 Feed per minute

> **WARNING**
> No override can be used for any commands such as for threading.

Format (M series)

For M series	
G94;	G code for feed per minute (Group 05)
F_;	Feed rate (mm/min or inch/min)

Format (T series)

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
0077			HICFR					

[Data type] Bit

HICFR 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.

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.3	CUTTING FEED
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

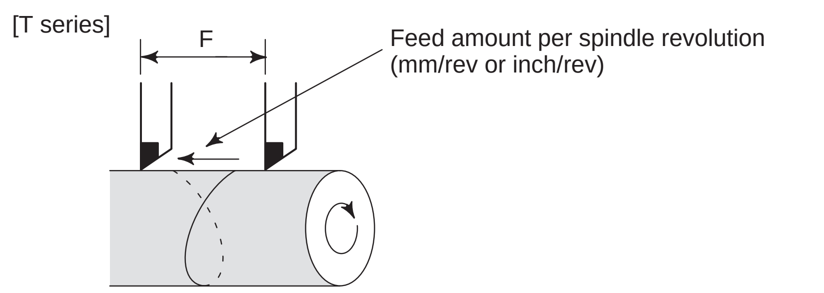


Fig. 7.1.4 Feed per revolution

- **Manual feed per revolution**

Jog feedrate can be specified by feed per revolution.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0008				MFPR					(T series)

[Data type] Bit

MFPR Jog feed
0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.3	CUTTING FEED
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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. 0788 to 0796) 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 0583, or feedrate upper limit for F5–F9 set by parameter 0584

X : any value of 1–127 set by parameter 0216

The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the CRT screen.

Signal

F1-digit feed select signal F1D <G140#7>

[Classification] Input signal

[Function] Increases or decreases F1-digit speed set by the parameters No. 0788 to 0796 using the manual pulse generator.

Since the manual pulse generator may also be used for axis feeding, signal F1D (G140#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	
G140	F1D								(M series)

Parameter

0216	Change of feedrate for one graduation on the manual pulse generator during F1 digit feed

[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, set n. That is, the number of revolutions of the manual pulse generator, required to reach feedrate Fmaxi is obtained. Fmaxi refers to the upper limit of the feedrate for an F1-digit feed command, and set it in parameter 0583 or 0584.

Fmax1: Upper limit of the feedrate for F1 to F4 (parameter 0583)

Fmax2: Upper limit of the feedrate for F5 to F9 (parameter 0584)

0583	Upper limit of feedrate for the F1–digit feed command (F1 to F4)
0584	Upper limit of feedrate for the F1-digit feed command (F5 to F9)

[Data type] Two–word

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

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 0583. If an F1-digit command F5 to F9 is executed, the upper limit is that set in parameter 0584.

0788	Feedrate for F1 digit command F1
0789	Feedrate for F1 digit command F2
0790	Feedrate for F1 digit command F3
0791	Feedrate for F1 digit command F4
0792	Feedrate for F1 digit command F5
0793	Feedrate for F1 digit command F6
0794	Feedrate for F1 digit command F7
0795	Feedrate for F1 digit command F8
0796	Feedrate for F1 digit command F9

[Data type] Two–word

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

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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.3	CUTTING FEED

7.1.6
Override

7.1.6.1
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.0533).

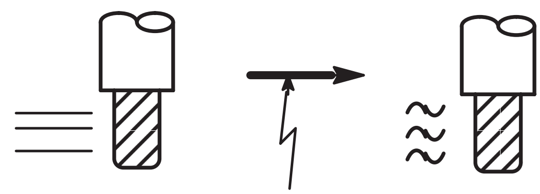


Fig.7.1.6.1 Rapid traverse override

● Feedrate

Actual feedrate is obtained by multiplying the rapid traverse rate preset by parameter Nos.0518 to 0521 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.0533 within a range of 0 to rapid traverse rate.

Signal

Rapid traverse override signal ROV1, ROV2 <G116#7, G117#7>

[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	Fo %

Fo: Set in parameter No.0533

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G116	ROV1							
G117	ROV2							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0078	EAXOVE								(M series)

[Data type] Bit

EAXOVE Dry run and override signals during axis control by the PMC

0 : Use the same signals as CNC
(1) Cutting feed override signal *OV1 to *OV128
(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) Cutting feed override signal *OV1E to *OV8E
(2) Override cancel signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

0533	F0 rate of rapid traverse override

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the F0 rate of the rapid traverse override.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.5.3	RAPID TRAVERSE OVERRIDE
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.5.3	RAPID TRAVERSE OVERRIDE

7.1.6.2 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.

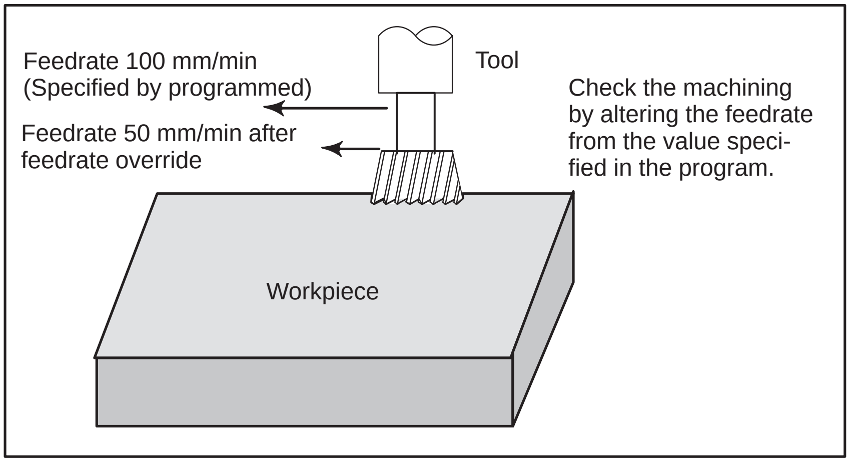


Fig.7.1.6.2 Feedrate override

Signal

Feedrate Override signal *OV1 to *OV8 <G121#0 to G121#3> (T/M series) *AOV16 to *AOV128 <G116#4 to G116#6, G117#6>(M series) *AOVR16 to *AOVR128 <G140#4 to G140#7> (T series)

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

$$\text{Override value} = \sum_{i=0}^{7} \left| 2^i \times Vi \right| \%$$

Vi=0 when *OVi is “1” and
Vi=1 when *OVi is “0”

These signals have the following weight.

*OV1=1%, *OV2=2%, *OV4=4%, *OV8=8%,
*AOV16=16%, *AOV32=32%, *AOV64=64%, *AOV128=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 speed specified in cutting feed in automatic operation mode 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	
G116		*AOV64	*AOV32	*AOV16					(M series)
G117		*AOV128							(M series)
G121					*OV8	*OV4	*OV2	*OV1	
G140	*AOVR128	*AOVR64	*AOVR32	*AOVR16					(T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0393			STOV0					

[Data type] Bit

STOV0 When cutting feedrate override is 0% during rapid traverse,
0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

	#7	#6	#5	#4	#3	#2	#1	#0	
0397						OVR255			(T series)
							OVR255		(M series)

OVR255 The signals of speed override in units of 1% (*AOVR16 to *AOVR128) are:
0 : Disabled.
1 : Enabled.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.5.3	CUTTING FEED
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.5.3	CUTTING FEED

7.1.6.3 Rapid traverse override B (T series)

General

The rapid traverse rate can be overridden by selecting a rapid traverse override signal (ROV1D to ROV3D).

Signal

[Operation]

Machine contact status			Override value	
ROV3D	ROV2D	ROV1D	When parameter OVRI is 0	When parameter OVRI is 1
0	0	0	100%	20%
0	0	1	50%	15%
0	1	0	25%	10%
0	1	1	F0	5%
1	0	0	5%	F0
1	0	1	10%	25%
1	1	0	15%	50%
1	1	1	20%	100%

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G116		ROV3D	ROV2D	ROV1D				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0003				OVRI				

OVRI 1 : When the override signal (*OV1 to *OV8, ROV1, ROV2, ROV1D to ROV3D) is set to 1, the speed increases.
0 : When the override signal (*OV1 to *OV8, ROV1, ROV2, ROV1D to ROV3D) is set to 0, the speed increases.

	#7	#6	#5	#4	#3	#2	#1	#0
0041					ROVB			

ROVB 1 : ROV1D to ROV3D are used as the rapid traverse override signals. (Enabled for PMC is used.)
0 : ROV1 and ROV2 are used as the rapid traverse override signals.

7.1.6.4 Override cancel

General

The override cancel signal fixes the feedrate override to 100%.

Signal

Override cancel signal OVC<G126#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
G126				OVC				

7.1.7 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. 7.1.7).
$2, \leqq \theta \leqq \theta p \leqq 178$, in Fig. 7.1.7
θp is a value set with parameter No. 0215. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

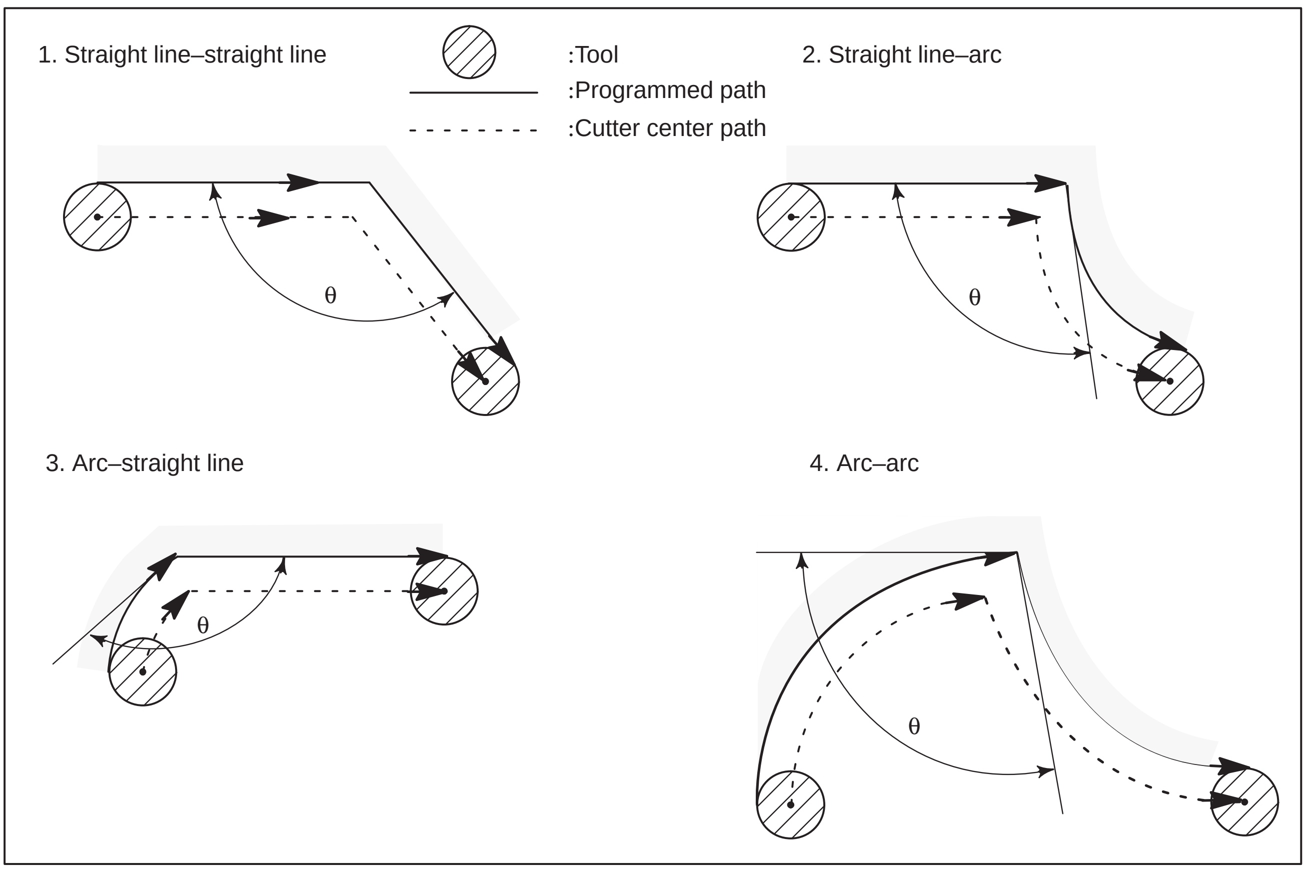


Fig. 7.1.7 (a) Inner corner

> **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. 0214. 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.

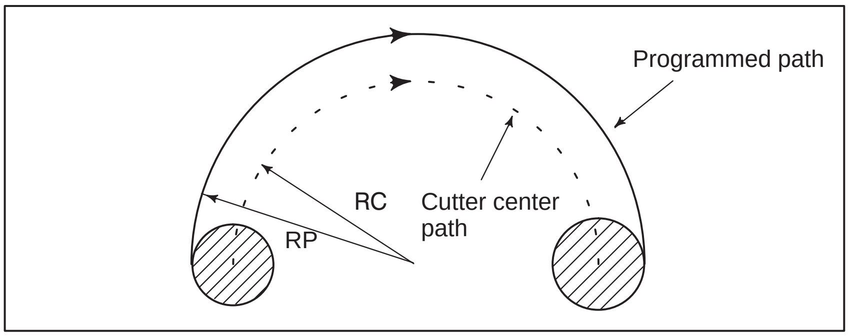


Fig. 7.1.7 (b) Internal circular cutting feedrate change

If Rc is much smaller than Rp, $Rc/Rp \doteq 0$; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 0213. When $Rc/Rp \leqq MDR$, 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

0213	Minimum deceleration ratio (MDR) of the inner circular cutting rate in automatic corner override

[Data type] Byte

[Unit of data] %

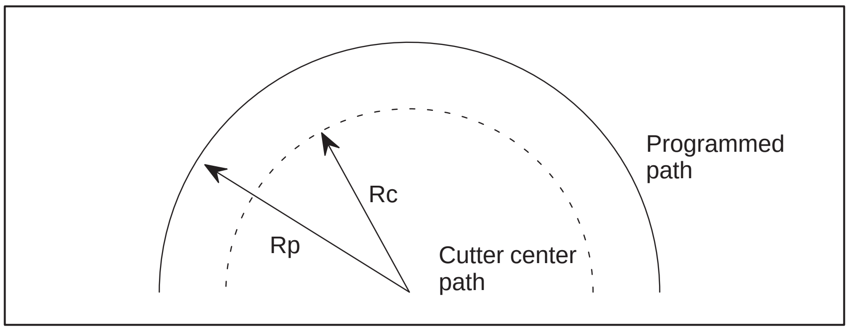
[Valid data range] 1 to 100

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:

$$F \times \frac{Rc}{Rp} \times \quad \left(\begin{array}{l} \text{Rc: Radius of the path of the cutter's center} \\ \text{Rp: Programmed radius} \end{array} \right)$$

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:

$F \times (MDR)$

0214	Amount of automatic override for an inner corner

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100 (standard value = 50)

Set inner corner automatic override value when automatic corner override is performed.

0215	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

0580	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.

0581	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 0215.)

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 0580 and 0581.

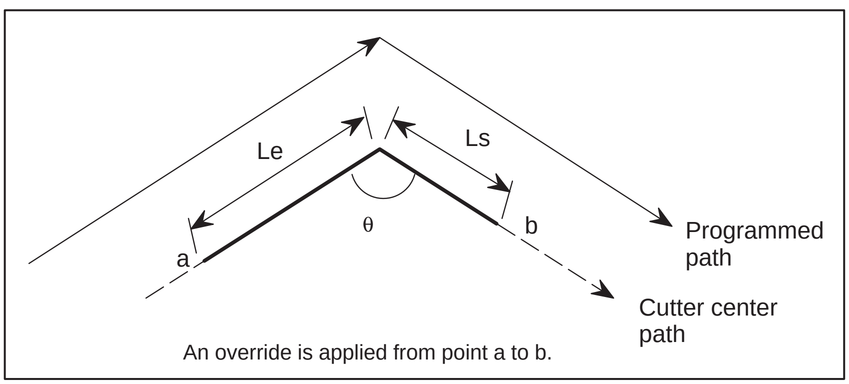


Fig. 7.1.7 (c) Distance Le and Ls in the automatic corner override at an inner corner

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.4.2	Automatic Override for Inner Corners
	II.5.4.3	Internal Circular Cutting Feedrate Change

7.1.8 External Deceleration

General

These signals decelerate the feedrate of the control axes down to the speed which has been set by parameter No. 0636.

Signal

External deceleration signal *+EDCX to *–EDCZ<G138>

[Classification] Input signal

[Function] These signals are used to apply deceleration; provided for each direction of each control axis; +/– indicates the direction, while the signal number corresponds to the number of the controlled axis.

* + EDC ×

X … The X axis is decelerated.

Y … The Y axis is decelerated.

Z … The Z axis is decelerated.

\+ … 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 decelerate to stop in the specified direction. If the speed with the signal set to 1 is lower, the speed does not change. (Acceleration will not occur.) The override function is disabled for external deceleration.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G138				*–EDCZ	*–EDCX		*+EDCZ	*+EDCX	(T series)
G138			*–EDCZ	*–EDCY	*–EDCX	*+EDCZ	*+EDCY	*+EDCX	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0059				EDMZ	EDMX		EDPZ	EDPX	(T series)
0059			EDMZ	EDMY	EDMX	EDPZ	EDPY	EDPX	(M series)

[Data type] Bit

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

0636	External deceleration rate

[Data type] Word

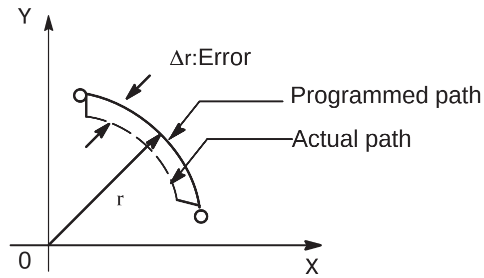
Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the external deceleration rate.

7.1.9 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:



$$\Delta r = \frac{1}{2}(T_1{}^2+T_2{}^2)\frac{V^2}{r}$$

Δr:Maximum radial error (mm)
V:Feedrate (mm/s)
r:Arc radius (mm)
T_1:Time constant (s) for exponential acceleration/deceleration of cutting feed
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.

Parameter

0495	Maximum feedrate for arc radius R

[Data type] Word

.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 maximum feedrate for the arc radius set in parameter No. 0863.
Set this parameter when the function for clamping the feedrate by an arc radius is supported.

0496	Minimum value (RV min) for arc radius–based feedrate clamp

[Data type] Word

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

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.

0863	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. 0495.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.9	FEEDRATE CLAMP BY CIRCLE RADIUS

7.1.10 Automatic Corner Deceleration (M series)

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 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 CHEAFD bit (bit 3 of parameter No. 0395).

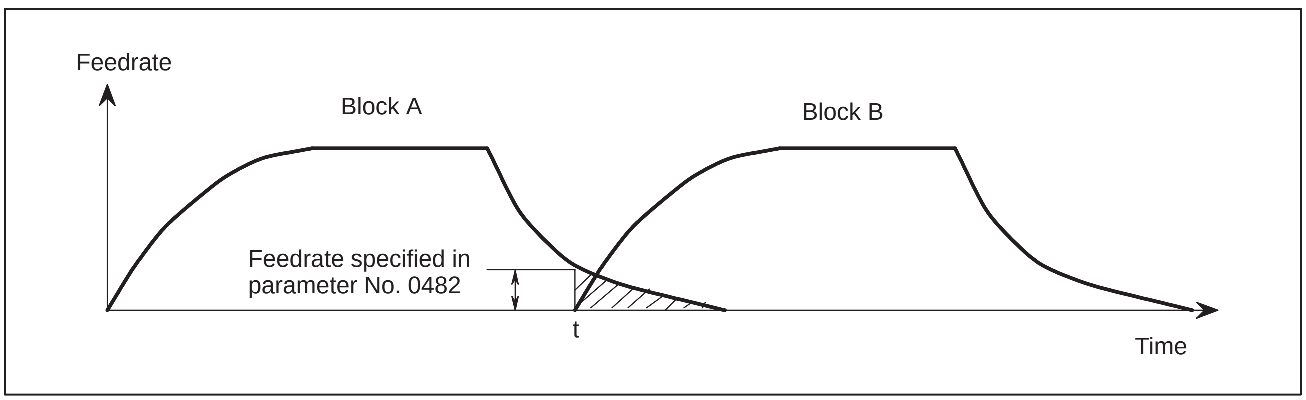
Feedrate control according to corner angle

- **Overview**

If the angle made by blocks A and B is smaller than that specified in parameter No. 0865 (for the selected plane), and if the feedrate is lower than that specified in parameter No. 0482, the system executes block B, assuming that no pulses are accumulated.

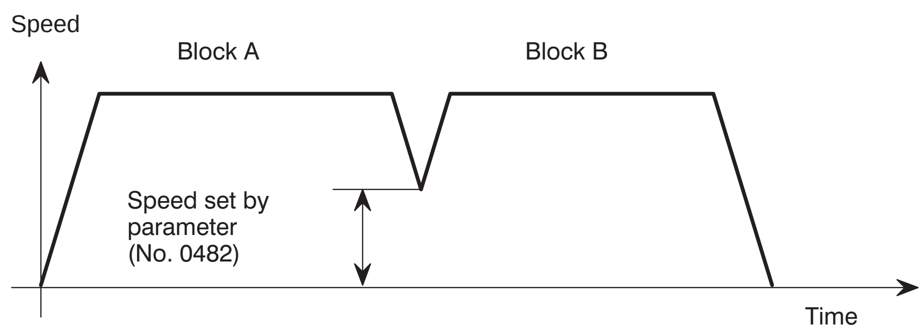
The figure shows the relationship between feedrate and time when a corner angle is smaller than the angle specified in the parameter.

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. 0865 (for the selected plane), and if the feedrates programmed for blocks A and B are higher than the value set in parameter No. 0482, 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.



● Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0393								FERDT

[Data type] Bit

FERDT Function for automatically reducing the feedrate at corners (automatic corner override function)
0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0395					CHEAFD			

[Data type] Bit

CHEAFD 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.

0482	Feedrate for assuming the termination of automatic corner deceleration

[Data type] Word

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

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

0865	Critical angle subtended by two blocks for automatic corner deceleration

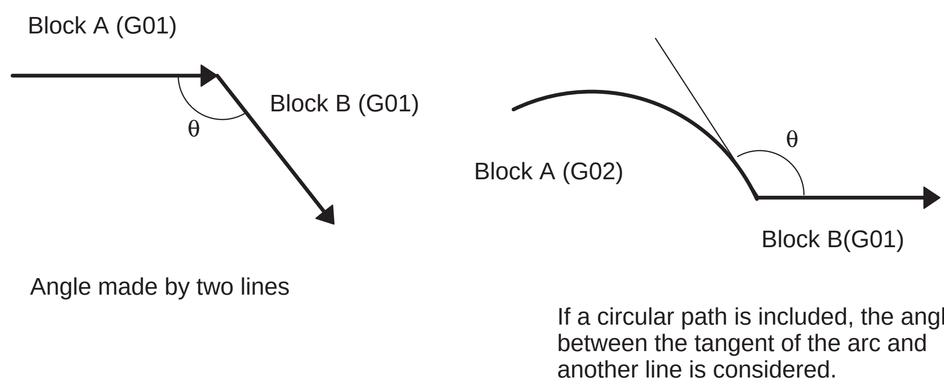
[Data type] Two–word

[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.



Caution

CAUTION

1 The angle of the machining tool path is compared with that specified in parameter No. 0865 only for the X–Y plane. The actual feedrate and that specified in parameter No. 0482 are compared only for the X and Y axes of the X–Y plane. Even if simultaneous movement is performed along three or more axes, the feedrates of only the X and Y axes are compared for the X–Y plane.
2 The roundness of a corner is determined by the angle and feedrate specified in parameter Nos. 0865 and 0482, 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, irrespective of the angle and feedrate specified in parameter Nos.0865 and 0482.
4 This function is disabled in single block and dry run mode.

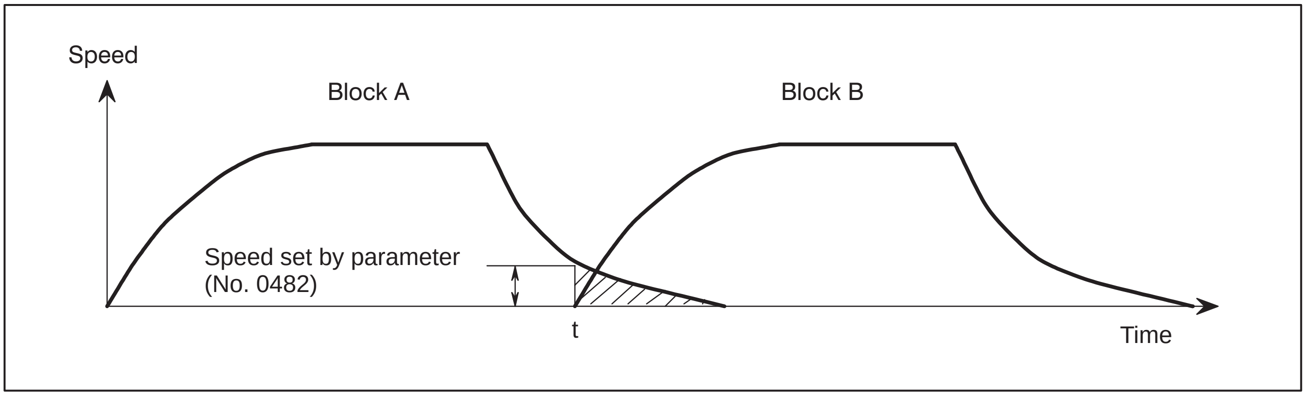
Feedrate control according to the feedrate difference for each axis

- **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. 0483, and if the feedrates for all axes are lower than that specified in parameter No. 0482, 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. 0483.

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. 0482.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

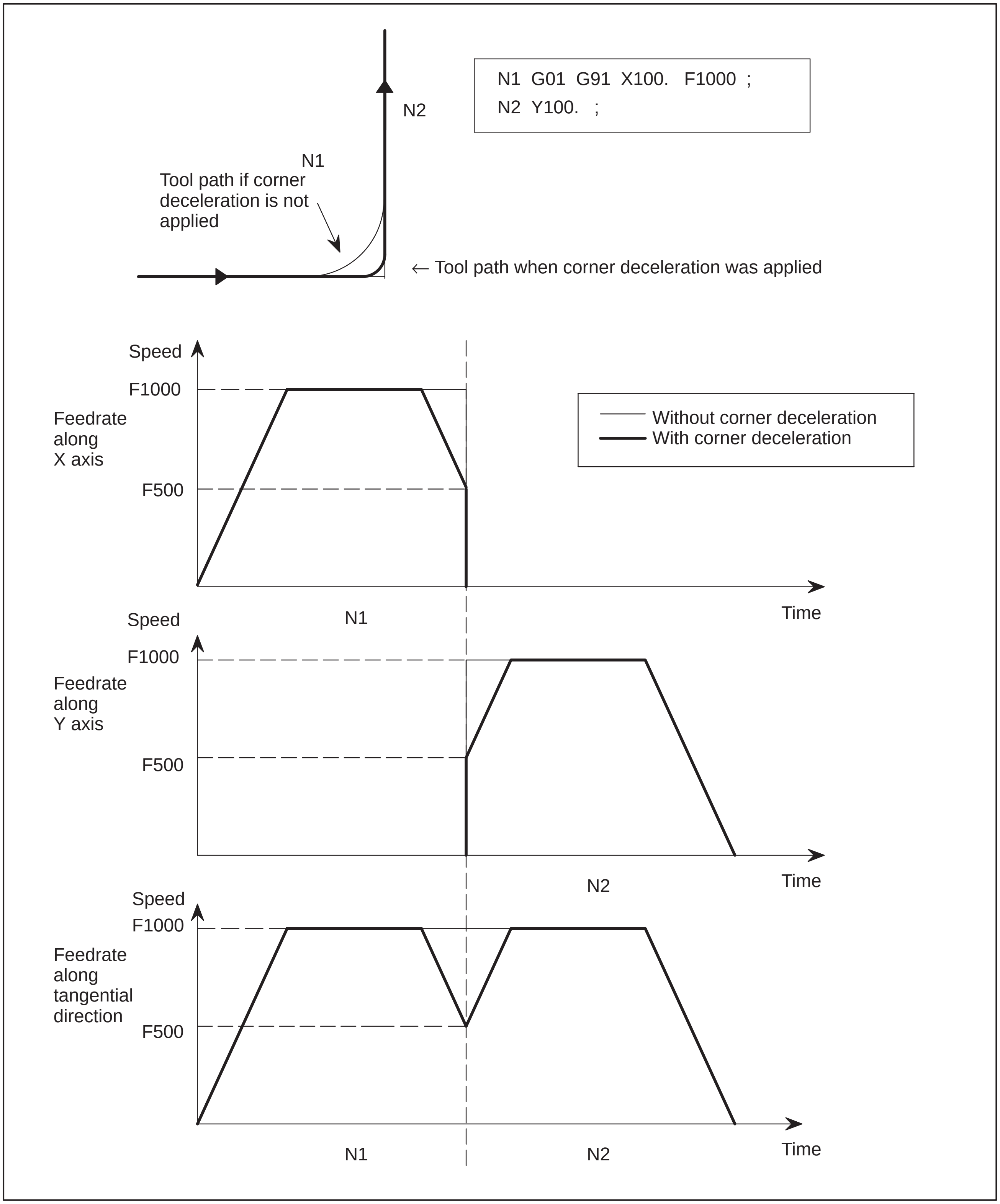
If the difference between the feedrates of blocks A and B for each axis exceeds the value specified in parameter No. 0483, the feedrate at the corner is calculated from the difference for each axis, as shown below. The feedrate is reduced to the calculated value in block A.

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. 0483. 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. 0483 is 500 mm/min, the deceleration shown below is performed:


N1 G01 G91 X100. F1000 ;
N2 Y100. ;
N2
N1
Tool path if corner deceleration is not applied
← Tool path when corner deceleration was applied
Speed
F1000
Feedrate along X axis
F500
Without corner deceleration
With corner deceleration
N1
Time
Speed
F1000
Feedrate along Y axis
F500
N2
Time
Speed
F1000
Feedrate along tangential direction
F500
N1
N2
Time

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0393								FERDT

[Data type] Bit

FERDT Function for automatically reducing the feedrate at corners (automatic corner override function)
0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0395					CHEAFD			

[Data type] Bit

CHEAFD 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.

0482	Feedrate for assuming the termination of automatic corner deceleration

[Data type] Word

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

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

0483	Allowable speed difference for the speed difference–based automatic corner deceleration function

[Data type] Word

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.

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, irrespective of the values specified for the parameters.
3 This function is invalid for the feed per rotation command, F1–digit feed command, 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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.5.8	AUTOMATIC CORNER DECELERATION

7.1.11 Advanced Preview Control (M Series)

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 as the feedrate becomes higher can be suppressed.
The tool can then follow specified values accurately and errors in the machining profile can be reduced.
This function becomes effective when advanced preview control mode is entered 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
(3) Block overlap function
(4) Look–ahead feed forword

For details on the above functions, see the descriptions of the functions.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0395		FWBTYP			CHEAFD	LSUP2		

[Data type] Bit

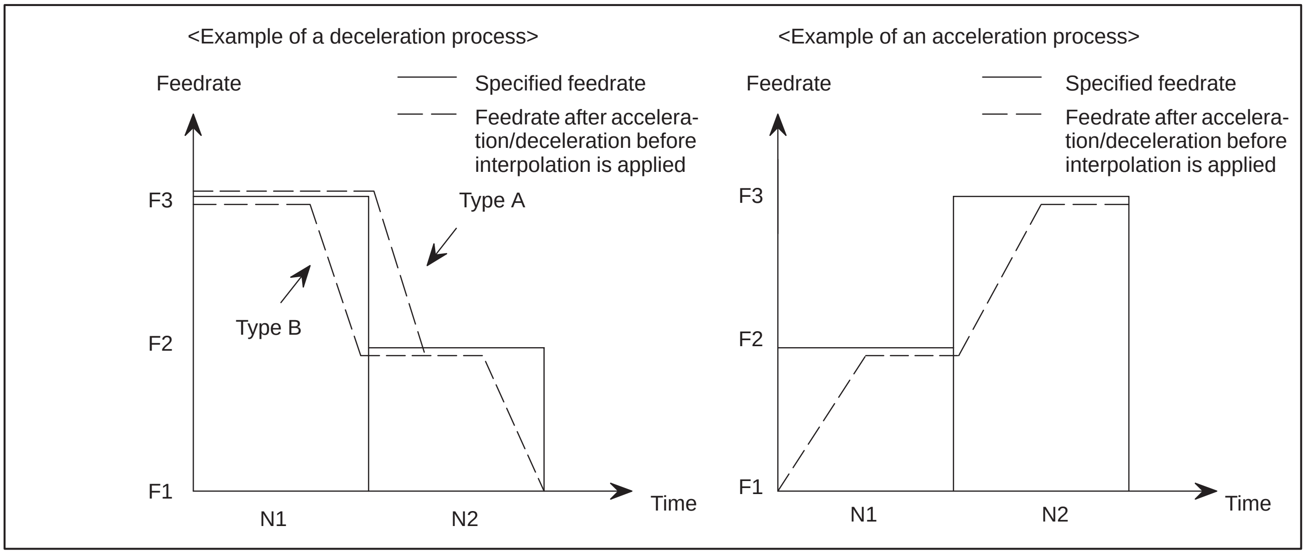
LSUP2 Acceleration/deceleration after interpolation for cutting feed in the advanced preview control mode
0 : Exponential acceleration/deceleration
1 : Linear acceleration/deceleration. (The function for linear acceleration/deceleration after interpolation for cutting feed is required.)

CHEAFD 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.

FWBTYP 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.



0483	Allowable speed difference for the speed difference based corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

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

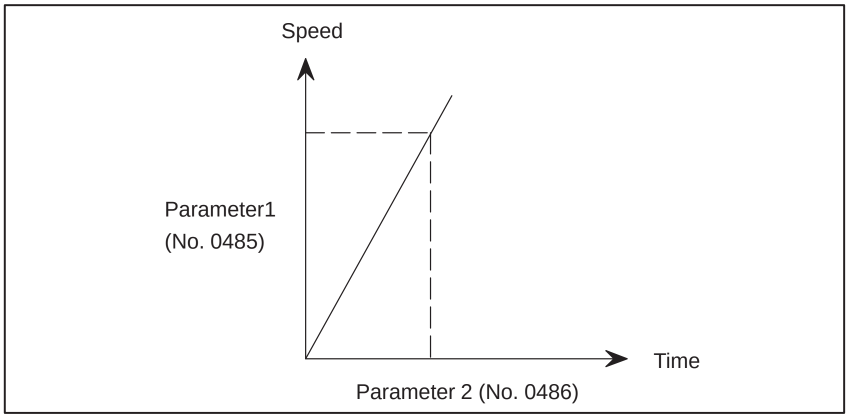
Set the speed difference for the speed difference based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

0485	Parameter 1 for setting an acceleration for linear acceleration/deceleration before interpolation in the advanced preview control mode (maximum machining speed during linear acceleration/deceleration before interpolation)

[Data type] Word

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

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation in the advanced preview 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. 0486.



CAUTION
When 0 is set in parameter No. 0485 or parameter No. 0486, linear acceleration/deceleration before interpolation is disabled.

0486	Parameter 2 for setting an acceleration for linear acceleration/deceleration before interpolation in the advanced preview control mode (time used to reach the maximum machining speed during linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation in the advanced preview control mode. In this parameter, set the time (time constant) used to reach the speed set in parameter No. 0485.

CAUTION
1 When 0 is set in parameter No. 0485 or parameter No. 0486, linear acceleration/deceleration before interpolation is disabled.
2 In parameter Nos. 0485 and 0486, set values that satisfy the following: Parameter No. 0485/Parameter No. 0486 ≥ 5

0487	Minimum speed for the automatic corner deceleration function (for the advanced preview 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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

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.

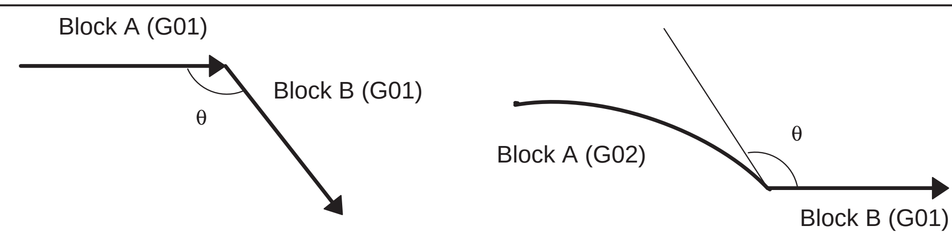
0864	Critical angle subtended by two blocks for automatic corner deceleration (for the advanced preview control)

[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 subtended by two straight lines

Angle subtended an arc tangent and straight line

0877	Exponential acceleration/deceleration time constant for cutting feed in the advanced preview control mode

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

Set an exponential acceleration/deceleration time constant for cutting feed in the advanced preview control mode.

0878	Minimum speed in exponential acceleration/deceleration for cutting feed in the advanced preview control mode

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set a minimum speed (FL) in exponential acceleration/deceleration for cutting feed in the advanced preview control mode.

0879	Time constant for linear acceleration/deceleration during cutting feed in the advanced preview control mode.

[Data type] Word

[Unit of data] ms

[Valid data range] 8 to 1024

This parameter sets a time constant for linear acceleration/deceleration for cutting feed in the advanced preview control mote.

NOTE
The function for linear acceleration/deceleration after interpolation for cutting feed is required.

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/deceleration type (A type/B type)	FWBTYP (0395#6)	←
Acceleration (Parameter 1)	0480	0485
Acceleration (Parameter 2)	0481	0486

· **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	CHEAFD (0395#3)	←
Minimum speed (according to the corner angle)	0482	0487
Critical angle (according to the corner angle)	0865	0864
Allowable speed difference for all axes (according to speed difference)	0483	←

Alarm and message

Number	Message	Description
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.

Note

NOTE

In the advanced preview control mode, the functions listed below cannot be specified. To specify these functions, cancel the advanced preview control mode, specify the desired function, then set advanced preview control mode again.

·Rigid tapping function
·Cs contour axis control function
·Feed per rotation
·F1–digit feed
·C–axis normal direction control function
·Polar coordinate interpolation function
·Cylindrical interpolation function
·Normal direction control
·Polar coordinate command
·Index table indexing
·Threading
·Synchronous feed
·High–speed cycle machining
·Handle interrupt
·Program restart
·Simplified synchronization control
·High–speed skip function
·Constant surface speed control
·Interrupt type custom macro
·Small–diameter peck drilling cycle
·High–speed remote buffer A/B
·Automatic tool length measurement
·Skip cutting
·G28 (low–speed reference position return)
·Axis control by PMC

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.24	ADVANCED PREVIEW CONTROL

7.2 ACCELERATION/ DECELERATION CONTROL

7.2.1 Automatic Acceleration/ Deceleration

- Rapid traverse linear acceleration/deceleration
- Cutting feed exponential 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 (a)).

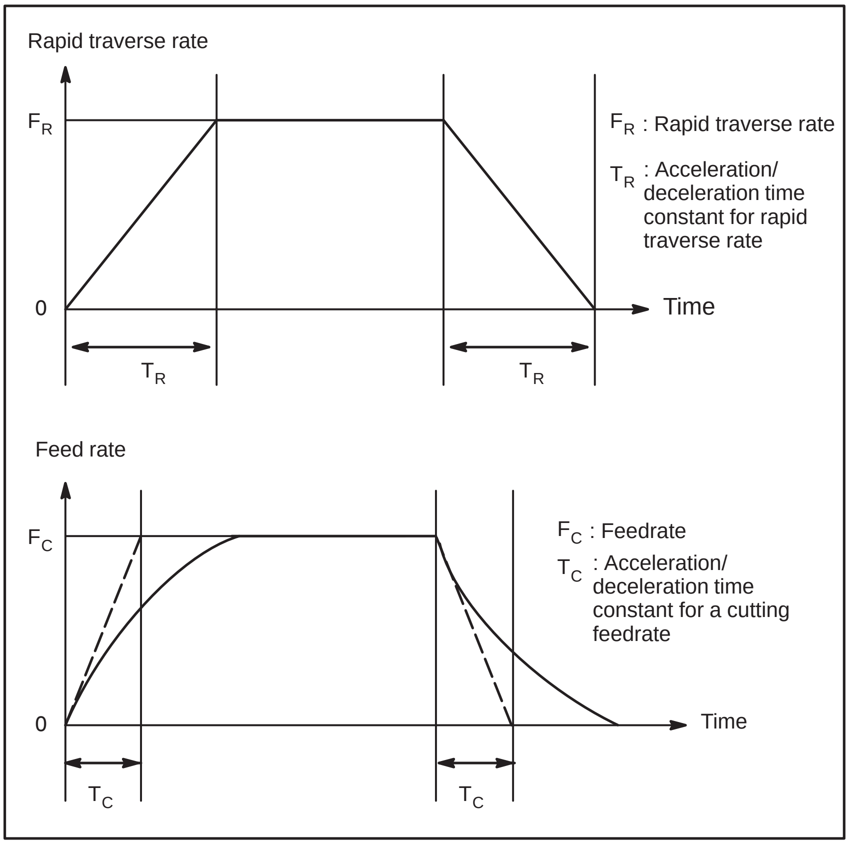


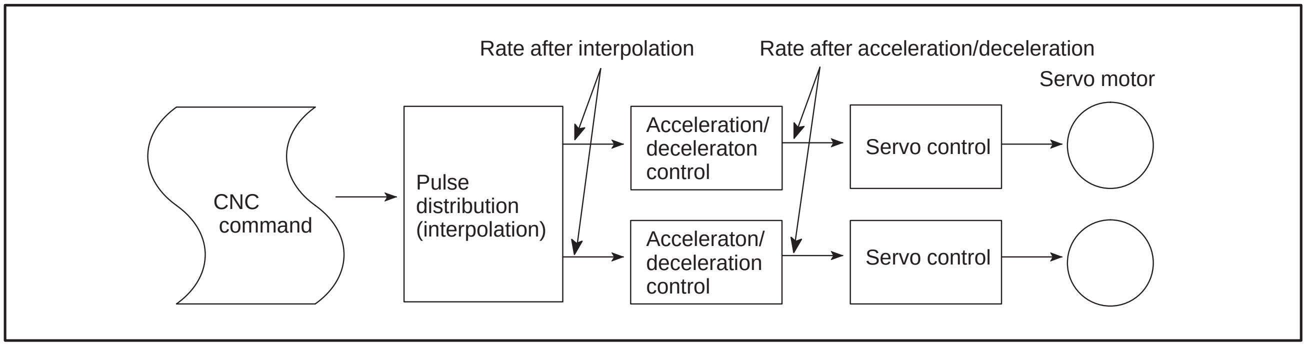
Fig.7.2.1 (a) Automatic acceleration/deceleration (example)

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 change in speed is also smoothly done.
It is not necessary to take acceleration/deceleration into consideration when programming.

Rapid traverse: Linear acceleration/deceleration (time constant per axis is set by parameters 0522 to 0525)

Cutting feed: Exponential acceleration/deceleration (time constant per axis is set by parameter 0529)

Jog feed : Exponential acceleration/deceleration (time constant per axis is set by parameters 0601 – 0604)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0045					LSUP			

[Data type] Bit axis

LSUP 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**
> If the optional function of linear acceleration/deceleration after interpolation in cutting feed is not provided, exponential acceleration/deceleration is used irrespective of this setting.

	#7	#6	#5	#4	#3	#2	#1	#0	
0065			G92ZAX						(T series)

G92ZAX Parameters for the Z–axis time constant and acceleration/deceleration lower limit (FL) in threading (G92):
0 : Parameters common to all axes
1 : Parameters 0627 and 0628

0522–0525	Time constant used for linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for linear acceleration/deceleration in rapid traverse. When the optional function of bell–shaped acceleration/deceleration in rapid traverse is provided, bell–shaped acceleration/deceleration is applied in rapid traverse. If the function is not provided, linear acceleration/deceleration is applied.

(1) When the function is provided, set this parameter to time constant T_1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter Nos. 0952 to 0955 to time constant T_2.

(2) When the function is not provided, specify a time constant used in linear acceleration/deceleration.

NOTE

When parameter Nos. 0952 to 0955 (time constant T_2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used in linear acceleration/deceleration in rapid traverse.

0526	Time constant of exponential acceleration/deceleration at X axis in thread cutting cycle	(T series)
0627	Time constant of exponential acceleration/deceleration at Z axis in thread cutting cycle	(T series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in thread cutting cycle (G76, G78 (G92 in G code system A)).

0528	FL rate of exponential acceleration/deceleration at X axis in thread cutting cycle	(T series)
0628	FL rate of exponential acceleration/deceleration at Z axis in thread cutting cycle	(T series)

[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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in thread cutting cycle (G76, G78 (G92 in G code system A)).

0529	Time constant of exponential acceleration/deceleration in cutting feed

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in cutting feed.

0530	FL rate of exponential acceleration/deceleration in 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
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed.

0555	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks	(T series)
0379	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks	(M series)

[Data type] Byte

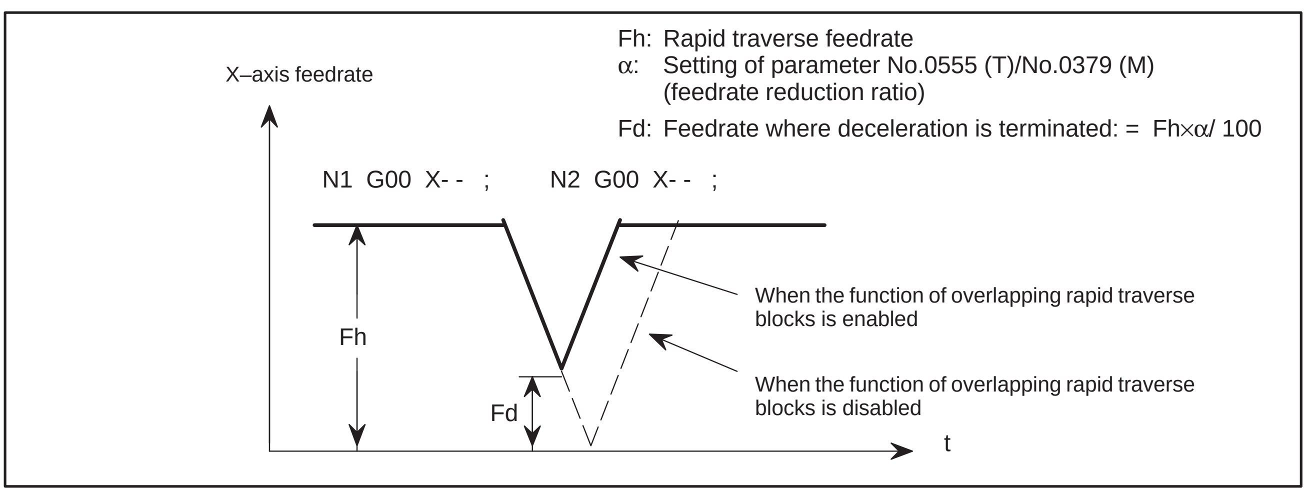
[Unit of data] %

[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**
> This parameter is effective when parameter SMZCT (No.0050#4 (T)/No.0048#4 (M)) is set to 1.

(Example)



0601 – 0604	Time constant of exponential acceleration/deceleration in jog feed

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in jog feed.

0605 – 0608	FL rate of exponential acceleration/deceleration in jog feed

[Data type] Word

[Unit of data]

[Valid data range]

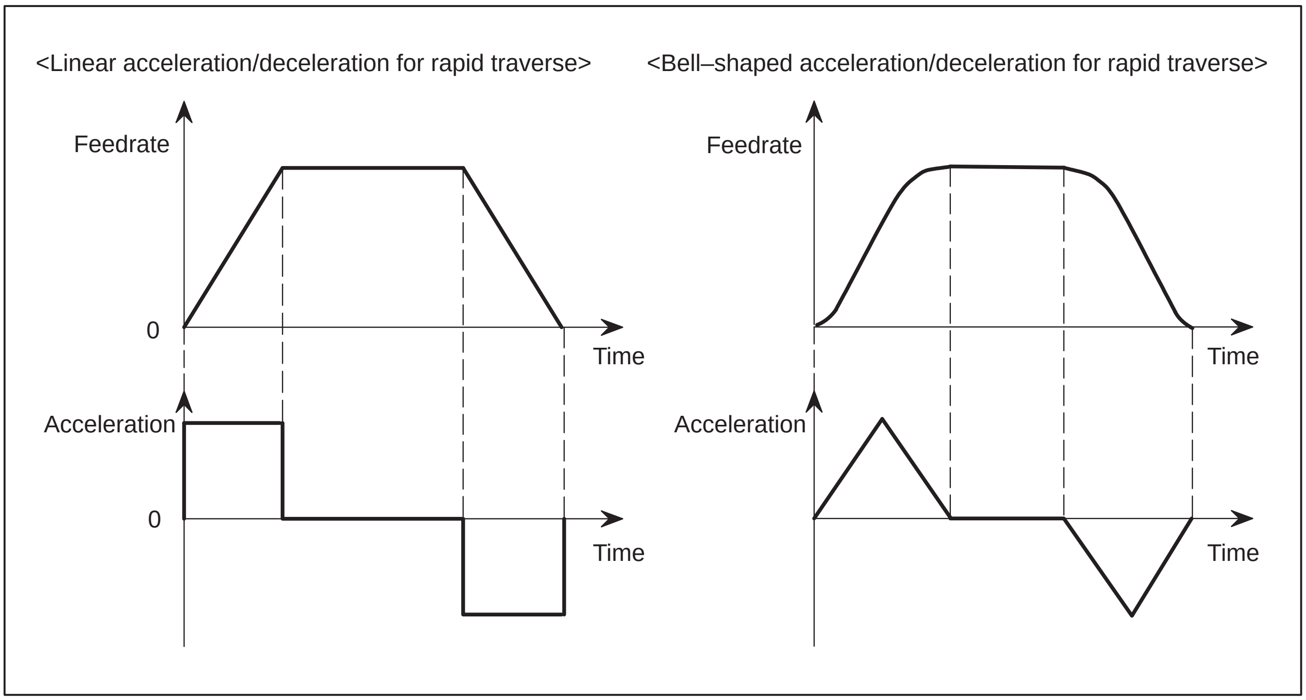
Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

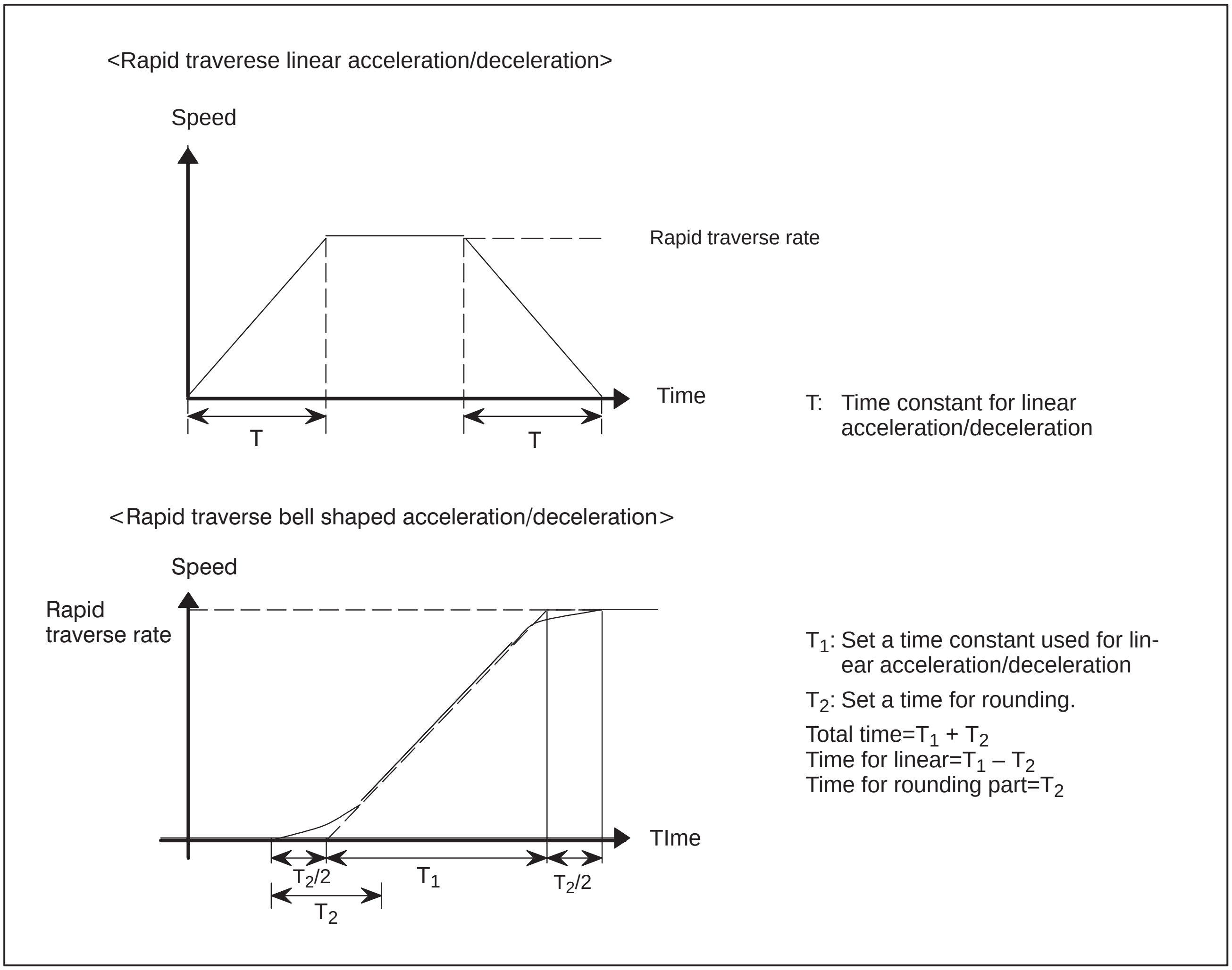
7.2.2 Bell–shaped Acceleration/ Deceleration for Rapid Traverse (M Series)

General

Bell–shaped acceleration/deceleration for rapid traverse 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 bell–shaped acceleration/deceleration for rapid traverse T_1 and T_2 are specified in parameter Nos. 0522 to 0525 and 0952 to 0955, respectively.



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

0522–0525	Time constant T_1 used in linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse. When the optional function of bell–shaped acceleration/deceleration in rapid traverse is provided, bell–shaped acceleration/deceleration is applied in rapid traverse. If the function is not provided, linear acceleration/deceleration is applied.

- When the function is provided, set this parameter to time constant T_1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter Nos. 0952 to 0955 to time constant T_2.

- When the function is not provided, specify a time constant used for linear acceleration/deceleration.

NOTE

When parameter Nos.0952 to 0955 (time constant T_2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used for linear acceleration/deceleration in rapid traverse.

0952–0955	Time constant T_2 used for bell–shaped acceleration/deceleration in rapid traverse	(M series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 512

Specify time constant T_2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis.

NOTE

1 This parameter is effective when the function of bell–shaped acceleration/deceleration in rapid traverse is provided. Set parameter Nos. 0522 to 0525 to time constant T_1 used for bell–shaped acceleration/deceleration in rapid traverse, and set this parameter to time constant T_2.
For details of time constants T_1 and T_2, see the description of General of this section.

2 When this parameter is set to 0, linear acceleration/deceleration is applied in rapid traverse. The setting in parameter Nos. 0522 to 0525 is used as a time constant in linear acceleration/deceleration.

Reference item

CONNECTION MANUAL (This manual)	7.2.1	Automatic Acceleration/Deceleration

7.2.3 Linear Acceleration/ Deceleration after Interpolation for Cutting Feed

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 3 of parameter No. 0045, LSUP), acceleration/deceleration is performed as follows:

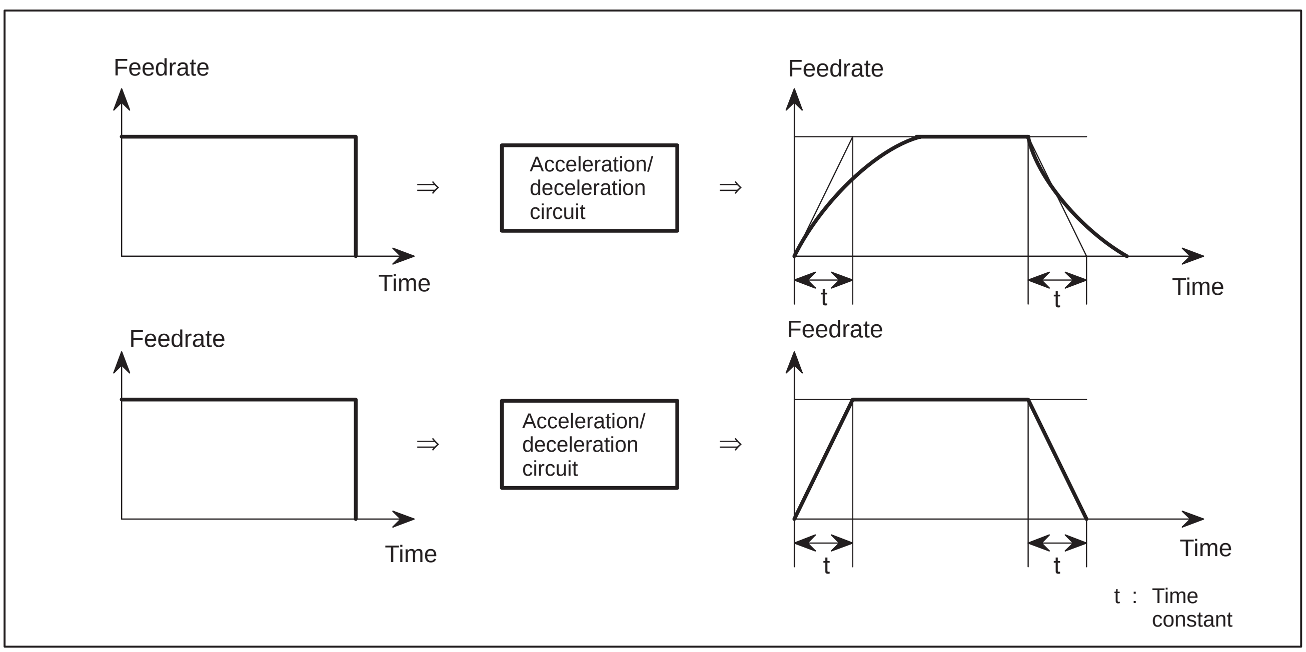
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 0635.

Jog feed: Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter Nos. 0601 to 0604

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 interpolation for cutting feed is an optional function. This function is enabled when the LSUP bit (bit 3 of parameter No. 0045) is specified. The time constants for cutting feed and jog feed are specified in parameter Nos. 0635 and 0601 to 0604 respectively. The values specified for the FL feedrate for cutting feed (parameter No. 0530) and the FL feedrate for jog feed (parameter No. 0548) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0045					LSUP			

[Data type] Bit

LSUP Acceleration/deceleration in cutting feed including feed in dry run
0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

0601–0604	Time constant of exponential acceleration/deceleration in jog feed

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in jog feed.

0635	Time constant of exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for linear acceleration/deceleration after interpolation in cutting feed.

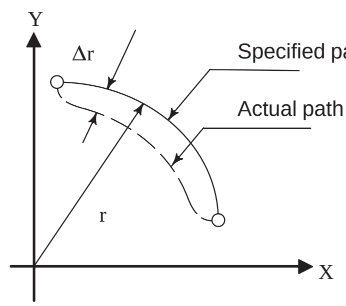
Note

If the optional function for linear acceleration/deceleration after interpolation for cutting feed is not provided, exponential acceleration/deceleration is always selected, irrespective of the setting.

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.

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 or bell shaped acceleration /deceleration after interpolation

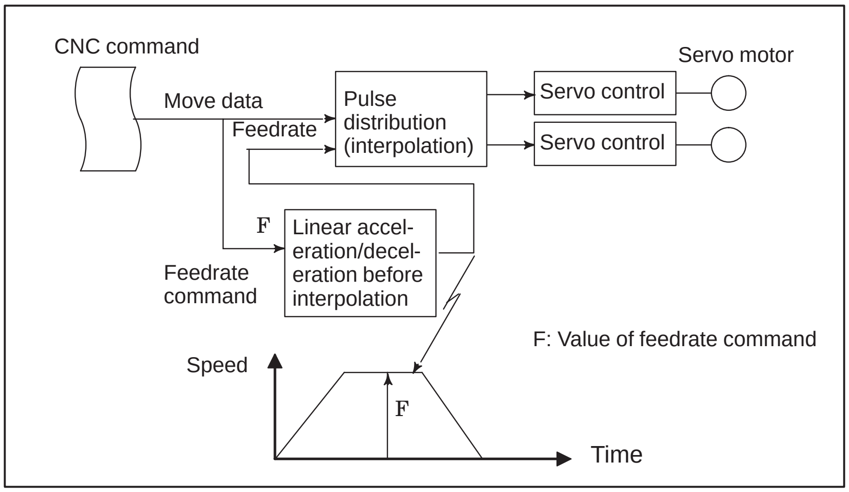
If the error caused by the time constant of the servo loop is excluded, the error cause by linear acceleration/deceleration or bell shaped acceleration/deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration.

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 Linear Acceleration/ Deceleration of Cutting Feed before Interpolation (M Series)

General

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.



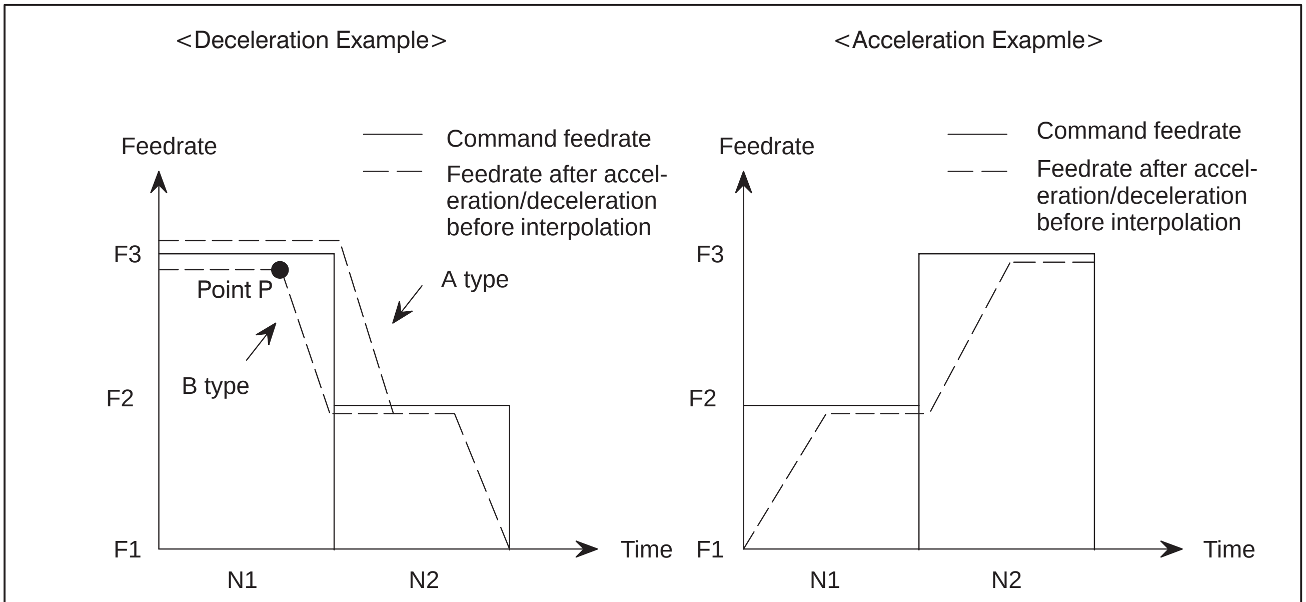
The function of linear acceleration/deceleration before interpolation increases or decreases the feedrate specified in the tangential direction.

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 TYP bit (bit 6 of parameter No. 0395) 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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0395		FWBTYP						

[Data type] Bit

FWBTYP Linear acceleration/deceleration of cutting feed before interpolation
0 : Type A of acceleration/deceleration before interpolation is used.
1 : Type B of acceleration/deceleration before interpolation is used.

0480	Parameter 1 for setting an acceleration for linear acceleration/deceleration before interpolation (maximum machining feedrate during linear acceleration/deceleration before interpolation)

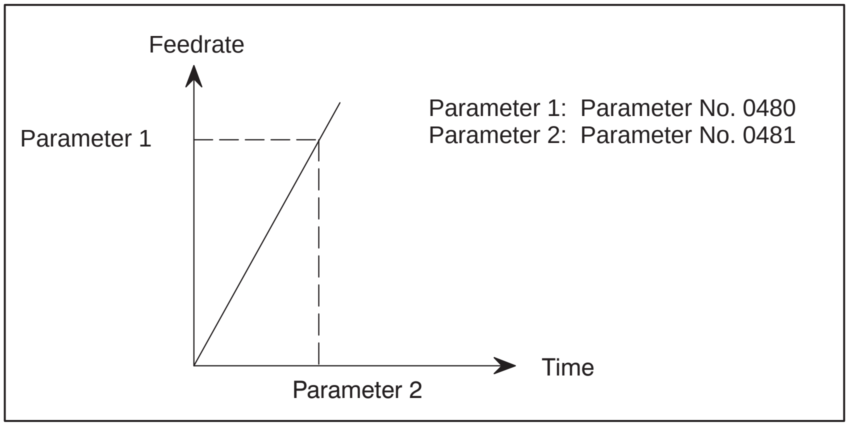
[Data type] Word

[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 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 48000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation. In this parameter, set a maximum machining feedrate during linear acceleration/deceleration before interpolation. In parameter No. 0481, set a time used to reach the maximum machining feedrate.



> **NOTE**
> 1 When 0 is set in parameter No. 0480 or parameter No. 0481, linear acceleration/deceleration before interpolation is disabled.
> 2 In the advanced preview control mode, parameter No. 0485 and parameter No. 0486 are valid.

0481	Parameter 2 for setting an acceleration for linear acceleration/deceleration before interpolation (time used to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation.)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter is used to set an acceleration for linear acceleration/deceleration before interpolation. In this parameter, set the time (time constant) used to reach the feedrate set in parameter No. 0480.

> **NOTE**
> 1 When 0 is set in parameter No. 0480 or parameter No. 0481, linear acceleration/deceleration before interpolation is disabled.
> 2 In parameter Nos. 0480 and 0481, set values that satisfy the following:
>
> $$\frac{\text{Parameter No. 0480}}{\text{Parameter No. 0481}} \geqq 5$$
>
> 3 In the look–ahead control mode, parameter No. 0485 and parameter No. 0486 are valid.

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 Even during acceleration/deceleration before interpolation, the block of G31 (skip function) is not subjected to acceleration/deceleration.
5 During acceleration/deceleration before interpolation, automatic corner override is enabled only when the internal circular cutting feedrate is changed.
6 Even 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.
7 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.
8 If inch input system in used, avoid specifying a command for simultaneous movement on two axes, including a rotation axis during acceleration/deceleration before interpolation.

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.

> **NOTE**
> The purpose of in–position check is to check that the servo motor has reached within a specified range (specified with parameter Nos.500 – 503).

Signal

In–position signals INP1 to INP4 <F184#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.

4 . . . . The 4rd axis is set to the in-position condition.

[Output condition] These signals turn to “1” in the following case :

· The acceleration/deceleration delay of the corresponding control axis is zero and the servo error is within the specified allowance.

These signals turn to “0” in the following cases :

· The acceleration/deceleration delay of the corresponding control axis is not zero.

· The servo error of the corresponding control axis exceeds the specified allowance

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F184					INP4	INP3	INP2	INP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0020			NCIPS					

[Data type] Bit

NCIPS Inposition check at deceleration
0 : Performed
1 : Not performed

0500 – 0503	In–position width

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

The in–position width is set.

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 (M series)

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
0020			NCIPS					

[Data type] Bit

NCIPS Inposition check at deceleration
0 : Performed
1 : Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
0045				CCINP				

[Data type] Bit

CCINP The in–position area for cutting feed is:
0 : Set in parameter Nos.0500 – 0503 (same as for rapid traverse).
1 : Set in bit 4 (CINPS) of parameter No.0399.

	#7	#6	#5	#4	#3	#2	#1	#0
0399				CINPS				

[Data type] Bit

CINPS When bit 4 (CCINP) of parameter No.0045 = 1, the in–position area for cutting feed is:
0 : Use value in parameter Nos.0609 to 0612 if the next block is also for cutting feed, or use value in parameter Nos.500 to 503 if the next block is for rapid traverse.
1 : Use value in parameter Nos.0609 to 0612, regardless of the next block. (The setting of parameter Nos.500 to 503 is used for rapid traverse, and the setting of parameter Nos.0609 to 0612 is used for cutting feed.)

		Parameter CINPS (No.0399#4)			
		0		1	
Parameter CCINP (No.0045#4)	0	Rapid→Rapid	Nos.0500 to 0503	Rapid→Rapid	Nos.0500 to 0503
		Rapid→Feed	Nos.0500 to 0503	Rapid→Feed	Nos.0500 to 0503
		Feed→Feed	Nos.0500 to 0503	Feed→Feed	Nos.0500 to 0503
		Feed→Rapid	Nos.0500 to 0503	Feed→Rapid	Nos.0500 to 0503
	1	Rapid→Rapid	Nos.0500 to 0503	Rapid→Rapid	Nos.0500 to 0503
		Rapid→Feed	Nos.0500 to 0503	Rapid→Feed	Nos.0500 to 0503
		Feed→Feed	Nos.0609 to 0612	Feed→Feed	Nos.0609 to 0612
		Feed→Rapid	Nos.0500 to 0503	Feed→Rapid	Nos.0609 to 0612

0500 – 0503	In–position width

[Data type] Word

[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.

0609 – 0612	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 (CCINP) of parameter No.0045=1.

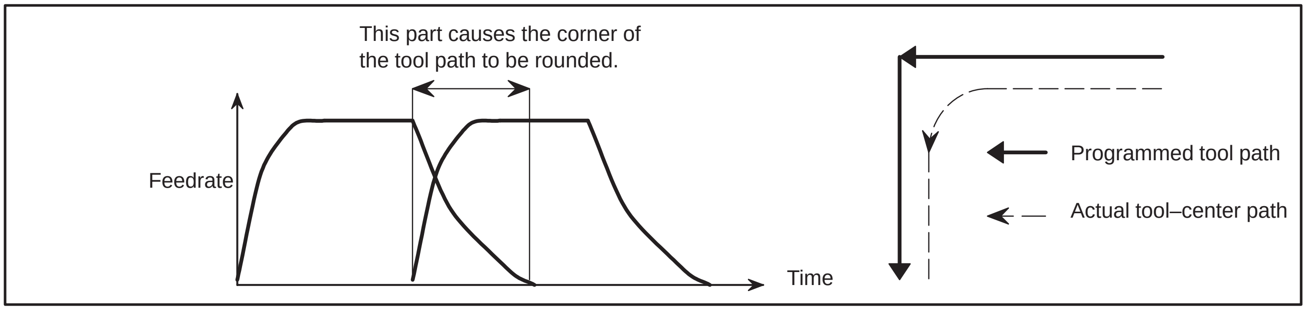
Note

> **NOTE**
> If the NCIPS bit (bit 5 of parameter No.0020) 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.

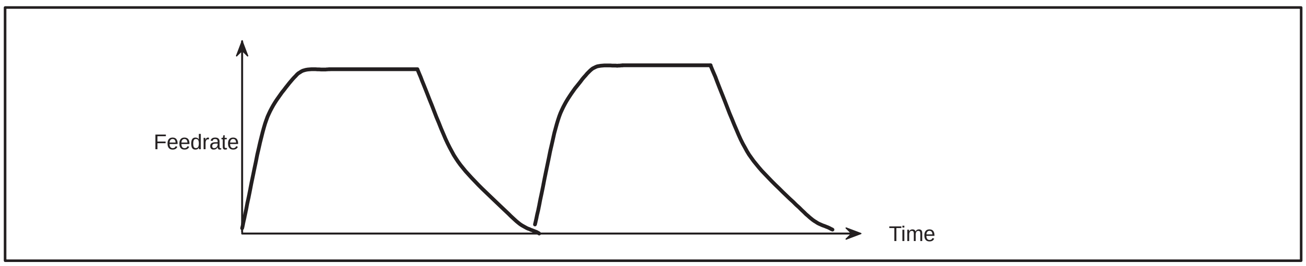
7.2.5.3 Error detect (T series)

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 <G126#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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G126		SMZ						

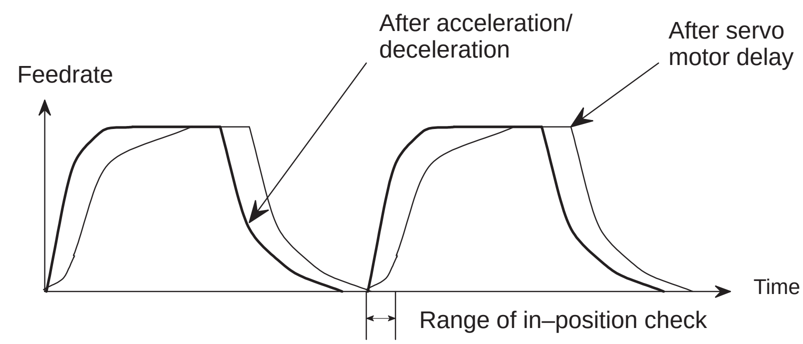
Note

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.

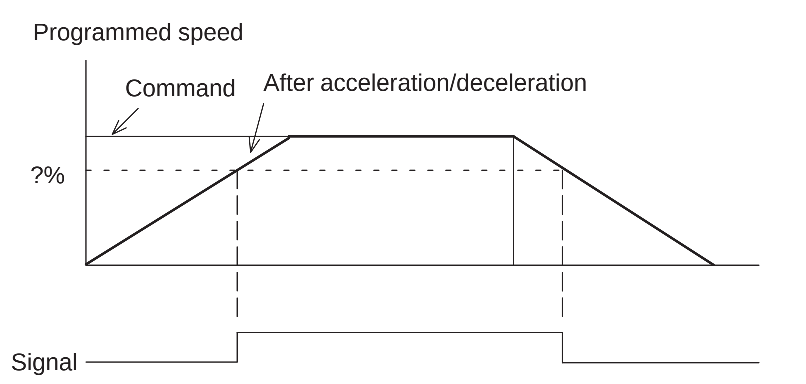
To prevent corner rounding due to delay caused by the servo motor, use the in–position check function together with this function.



7.2.5.4 Signals output according to the speed of travel along an axis (M series)

General

When a programmed value after linear acceleration/deceleration by rapid traverse exceeds the value (percentage) specified in the corresponding parameter, a signal is output.



Signal

Signals output according to the speed of travel along an axis SPDS1 to SPDS4 <G189#0 to #3>

[Function] The signal reports that the speed programmed for any of the first to the fourth axes in direct acceleration/deceleration exceeds the value specified in the corresponding parameter.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F189					SPDS4	SPDS3	SPDS2	SPDS1

Parameter

0243	SPDSP

SPDSP Set a reference speed in %, to enable the function to output a signal according to the speed of travel along an axis. If a value exceeding the range indicated below is set, no signal is output.
Valid data range: 0% to 99%

	#7	#6	#5	#4	#3	#2	#1	#0
0394							SPDS	

SPDS The function used to output a signal according to the speed of travel along an axis is:
1 : Enabled.
0 : Disabled.

Caution

CAUTION
This function is enabled in 1st to 4th axis .
(This function is disabled in 5th and 6th axis)

Note

NOTE
1 The signal is output in rapid traverse mode only.
2 Identical values are set for acceleration and deceleration.

7.2.6 Feed Forward in Rapid Traverse (M series)

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
0395		FDFFEN						

[Data type] Bit

FDFFEN 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 AC SERVO AMPLIFIER Maintenance Manual (B–65005E)” or the “FANUC CONTROL MOTOR α series Maintenance Manual (B–65165E).

8 AUXILIARY FUNCTION

8.1 MISCELLANEOUS FUNCTION/2ND AUXILIARY FUNCTION

General

- **Miscellaneous Function (M code)**

When a numeral of up to 8 digits is specified following address M, 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”)

- **2nd Auxiliary Function (B code)**

When three or six digits (M series) / eight digits (T series) 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. BCD output is performed for the M series while binary output is performed for the T series.

- **Command Format of 2nd Auxiliary Function (T series)**

· **Command range**

–99999999 to +99999999

· **Command method**

1.A decimal point can be used for input by setting AUX (bit 0 of parameter No. 3405).

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 CPRD (No.0015#7).

Command		Output value
When CPRD is 1:	B1	1000
When CPRD 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 BLIN (No.0032#5). When DPI=1.

Command		Output value
When BLIN is 1:	B1	10000
When BLIN 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	M11 to M38	MF	DEN	FIN
Spindle–speed function	S	S11 to S28	SF		
Tool function	T	T11 to T28 (T) T11 to T48 (M)	TF		
Secondary auxiliary function	B	B0 to B31 (T) B11 to B38 (M)	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.

(2) Code signal M11 to M38 is sent. After period TMF, specified in parameter No.0009#4 – #7 (standard value: 16 msec), strobe signal MF is set to 1. The code signal is the binary representation or BCD of the programmed value xxx.(*1) If the move, dwell, spindle–speed, or another function is specified at the same time 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.0009#0 – #3 (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)

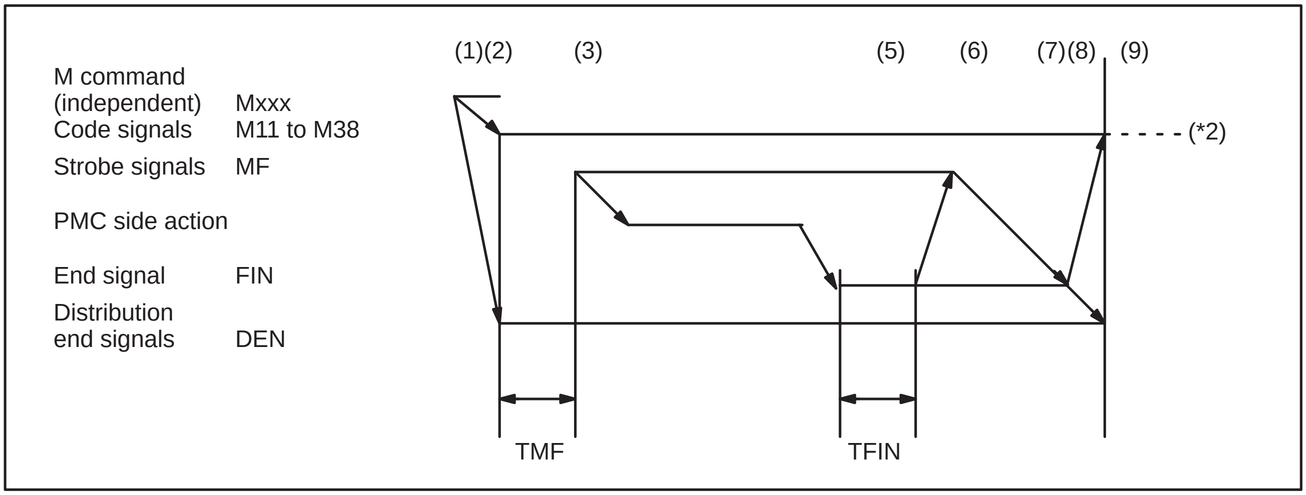
(9) Once all other commands in the same block have been completed, the CNC executes the next block.

*1 When the tool function is executed, the programmed tool number is sent as the code signal (T series).

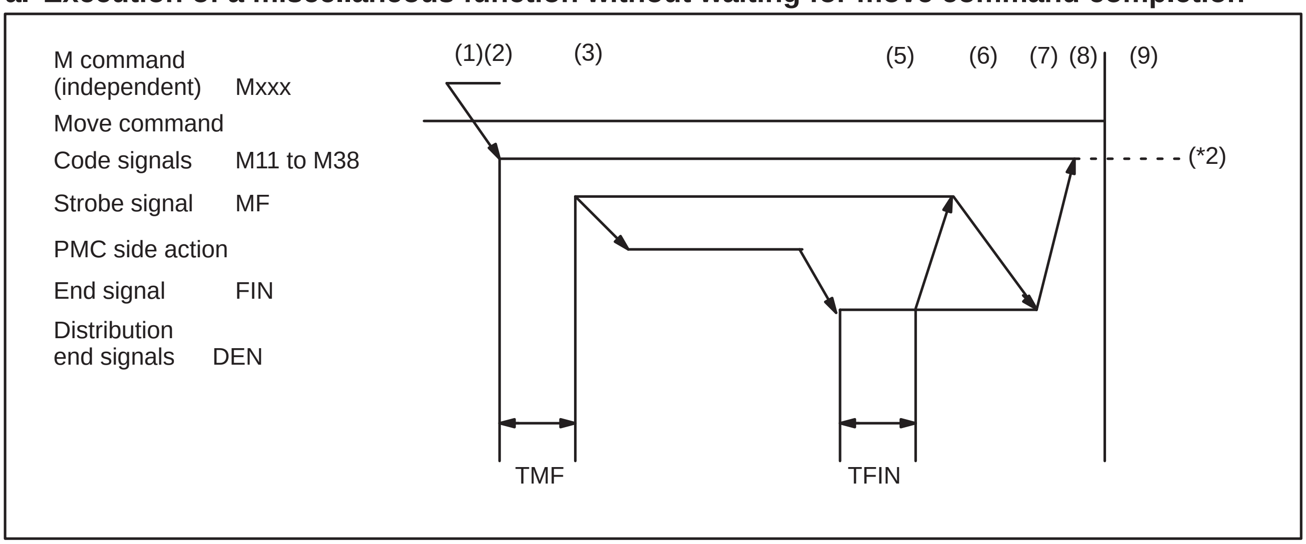
*2 When the spindle–speed function, tool function, or secondary miscellaneous function is executed, the code signal is maintained until a new code for the corresponding function is specified.

The timing diagram is shown below:

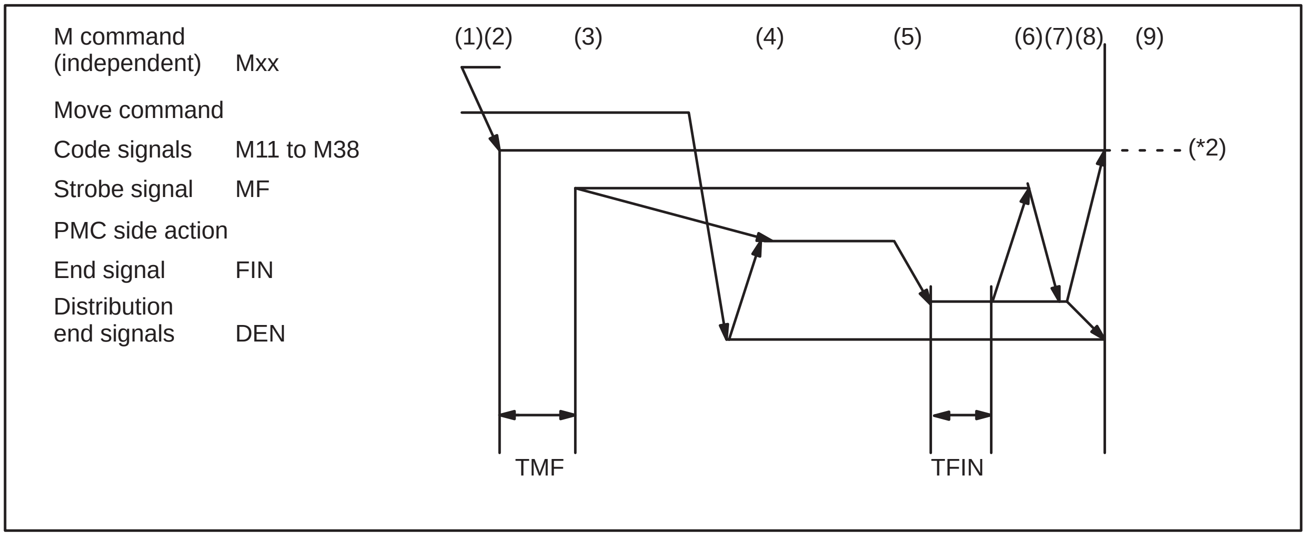
Example 1 Single miscellaneous function specified in a block



Example 2 Move command and miscellaneous function in the same block
a. Execution of a miscellaneous function without waiting for move command completion



b. Execution of a miscellaneous function after move command completion



Signal

Miscellaneous function code signals M11 to M38 <F151, F157#0 to #3> Miscellaneous function strobe signal MF<F150#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 subject to output even when specified:
- M98, M99, M198
- M code that calls a sub program (parameter Nos.0240 to 0242)
- M code that calls a custom macro (parameter Nos.0230 to 0239)

2 Decode signals as well as the code signals and strobe signal are output for the miscellaneous functions listed below. (only for M series)
M00, M01, M02, M30

Decode M signals (M series) M00<F154#7>, M01<F154#6>, M02<F154#5>, M30<F154#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	M00
M01	M01
M02	M02
M30	M30

[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 S11 to S28 <F152> Spindle–speed strobe signal SF<F150#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 T11 to T48<F153#0 to #7, F156#0 to #7>(M series)/ T11 to T28 <F153#0 to #7>(T series) Tool function strobe signal TF<F150#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 B11 to B38<F155#0 to #7, F154#0 to #3>(M series) B0 to B31 <F276 to F279> (T series) Second auxiliary function strobe signal BF1, BF2<F150#7, #6> (M series) BF<150#7>(T 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<G120#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.0009#0 to #3) 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 signals DEN<F149#3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those miscellaneous functions, spindle 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:

· The completion of miscellaneous functions, spindle–speed functions, tool functions, and so forth is being awaited, 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 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.

Passing point signal (T series) DEN2<F150#6>

[Classification] Output signal

[Function] The signal indicates that the position specified by the passing point signal output function has been reached.

[Output condition] The DEN2 signal becomes 1 when:

- The specified position is passed.

The DEN2 signal becomes 0 when:

- Miscellaneous function completion signal FIN is returned.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G120					FIN				

	#7	#6	#5	#4	#3	#2	#1	#0	
F149		DEN2			DEN				
F150	BF				TF	SF		MF	(T series)
F150	BF1	BF2			TF	SF		MF	(M series)
F151	M28	M24	M22	M21	M18	M14	M12	M11	
F152	S28	S24	S22	S21	S18	S14	S12	S11	
F153	T28	T24	T22	T21	T18	T14	T12	T11	
F154	M00	M01	M02	M30	B38	B34	B32	B31	(M series)
F155	B28	B24	B22	B21	B18	B14	B12	B11	(M series)
F156	T48	T44	T42	T41	T38	T34	T32	T31	(M series)
F157					M38	M34	M32	M31	
F276	B7	B6	B5	B4	B3	B2	B1	B0	(T series)
F277	B15	B14	B13	B12	B11	B10	B9	B8	(T series)
F278	B23	B22	B21	B20	B19	B18	B17	B16	(T series)
F279	B31	B30	B29	B28	B27	B26	B25	B24	(T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0009	TMF				TFIN			

[Data type] Bit

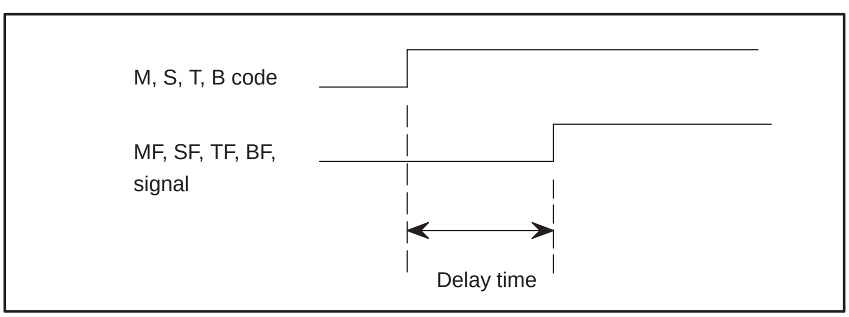
[Unit of data] 16[msec]

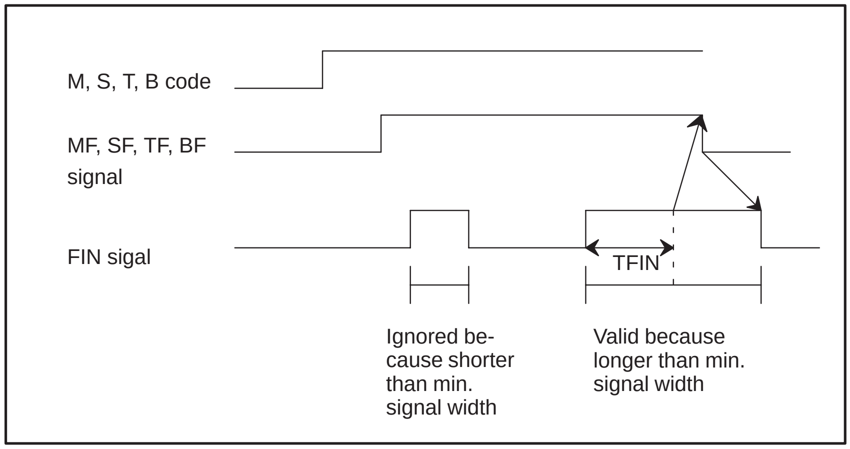
[Valid data range] 1 to 6 (16[msec] to 256[msec])

TFIN Set a time range within which the miscellaneous, spindle–speed, or tool function completion signal FIN (G120, #3) can be accepted.

TMF Set a period starting from when an M, S, or T code is sent until the miscellaneous function code read signal MF (F150, #0), spindle–speed function code read signal SF (F150, #2), or tool function code read signal TF (F150, #3) is sent.

TMF	TFIN	Setting value			
		#3 #7	#2 #6	#1 #5	#0 #4
16msec	16msec or more	0	0	0	0
32msec	32msec or more	0	0	0	1
48msec	48msec or more	0	0	1	0
64msec	64msec or more	0	0	1	1
80msec	80msec or more	0	1	0	0
96msec	96msec or more	0	1	0	1
112msec	112msec or more	0	1	1	0
128msec	128msec or more	0	1	1	1
144msec	144msec or more	1	0	0	0
160msec	160msec or more	1	0	0	1
176msec	176msec or more	1	0	1	0
192msec	192msec or more	1	0	1	1
208msec	208msec or more	1	1	0	0
224msec	224msec or more	1	1	0	1
240msec	240msec or more	1	1	1	0
256msec	256msec or more	1	1	1	1





	#7	#6	#5	#4	#3	#2	#1	#0
0015	CPRD							

[Data type] Bit

CPRD 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
0019			M02NR					

[Data type] Bit

M02NR 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 is executed, starting from the beginning.

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	
0020				BCD3					(M series)

[Data type] Bit

BCD3 The B code is output using:

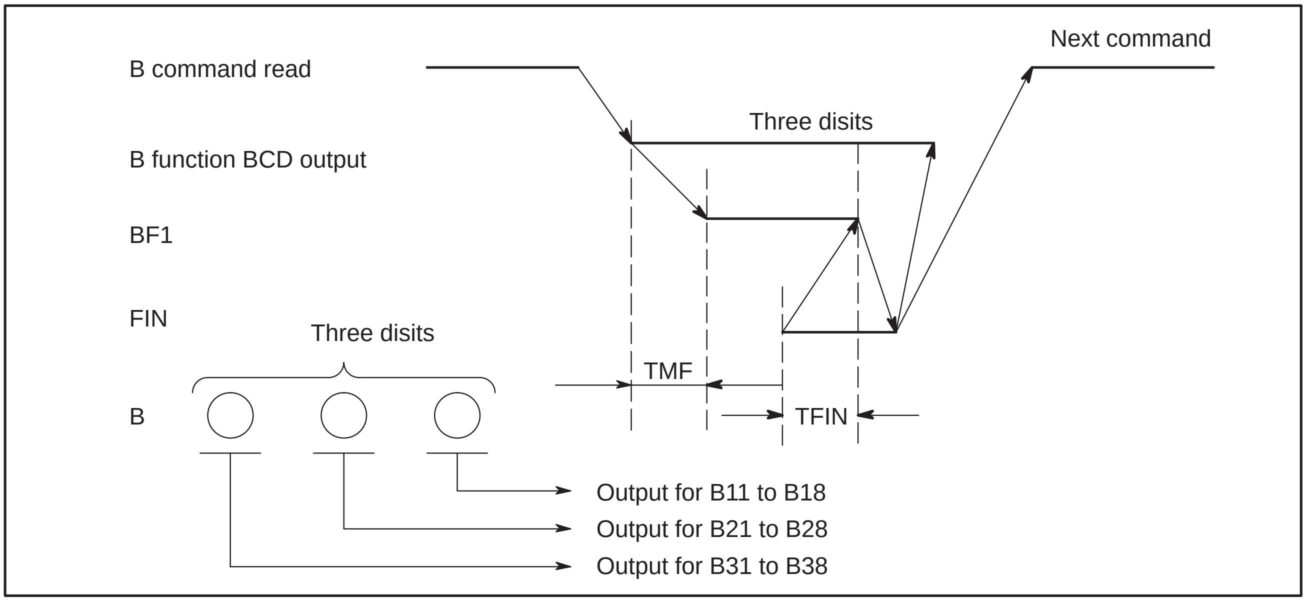
1 : Three digits.

0 : Six digits.

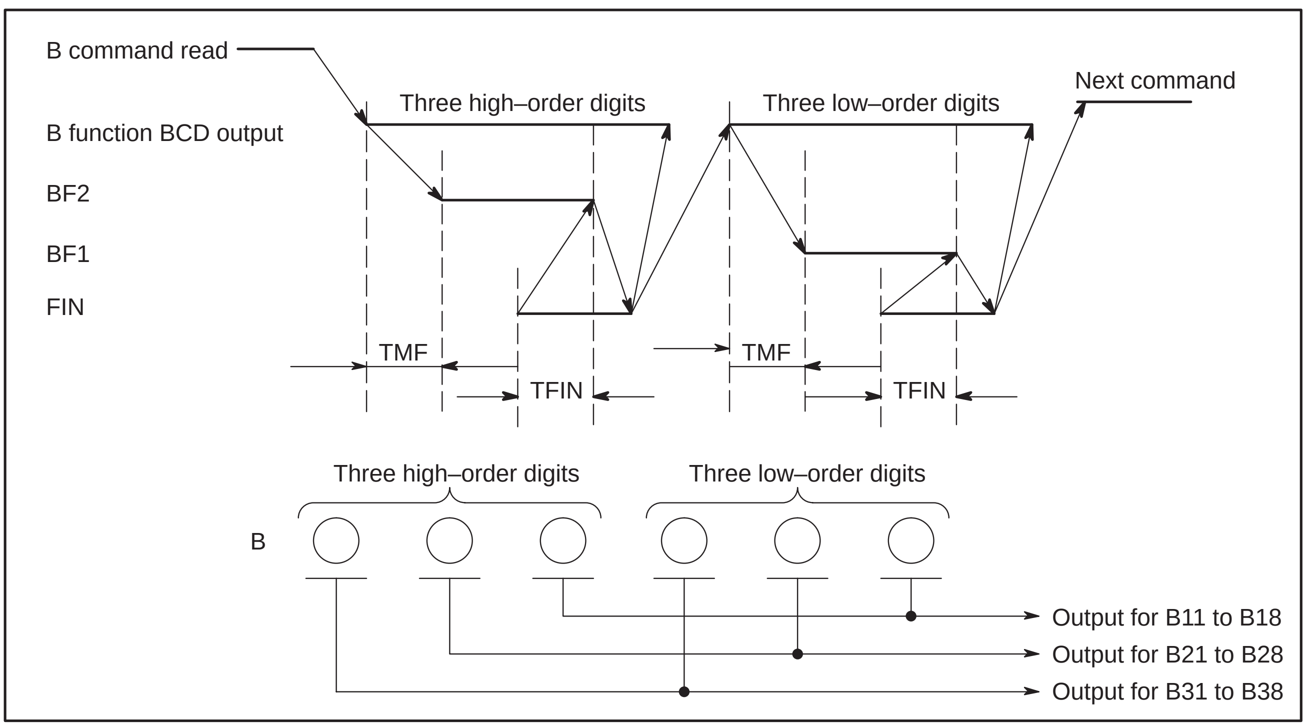
When a B command is programmed for automatic or MDI operation, the value of the B command is sent to the PMC as a BCD code.
The B function can output the BCD code of the B command using three or six digits (BCD3 bit, bit 4 of parameter 0020). When six–digit BCD code output is selected, the three high–order digits and three low–order digits of the BCD code are output separately. The B function outputs the code at the same timing as the miscellaneous function, spindle–speed

function, and tool function. The B function uses the same FIN signal as the miscellaneous function, spindle–speed function, and tool function. For details, see the relevant section.

(1) Output timing of the three–digit BCD code (when the BCD3 bit (bit 4 of parameter 0020) is set to 1)



(2) Output timing of the six–digit BCD code (when the BCD3 bit (bit 4 of parameter 0020) is set to 0)



	#7	#6	#5	#4	#3	#2	#1	#0	
0032			BLIN						(T series)

[Data type] Bit

BLIN 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
0065	M3B							

[Data type] Bit

M3B In a single block,
0: Up to three M codes can be specified.
1: Only one M code can be specified.

0111	M code preventing buffering 1
0112	M code preventing buffering 2

[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.

	#7	#6	#5	#4	#3	#2	#1	#0
0393					M3RQNG			

M3RQNG The command for the three–digit M code is:
1 : Disabled (P/S 003).
0 : Enabled.

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 If the second miscellaneous function is provided, B 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 (Nos. 0111 to 0112).
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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.11.1	AUXILIARY FUNCTION (M FUNCTION)
	II.11.3	THE SECOND AXILIARY FUNCTIONS (B CODE)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.11.1	AUXILIARY FUNCTION (M FUNCTION)
	II.11.3	THE SECOND AXILIARY FUNCTIONS (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<G103#7>

[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 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 (for M series only) 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G103	AFL							

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
OPERATOR’S MANUAL (For Lathe) (B–61394E)	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 M11 to M38 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.0009#4 – #7 (Standard setting: 16 msec).
The second M command (Mbb) sends the code signal M211-M238, the third M command (Mcc) sends the code signal M311-M338, 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.

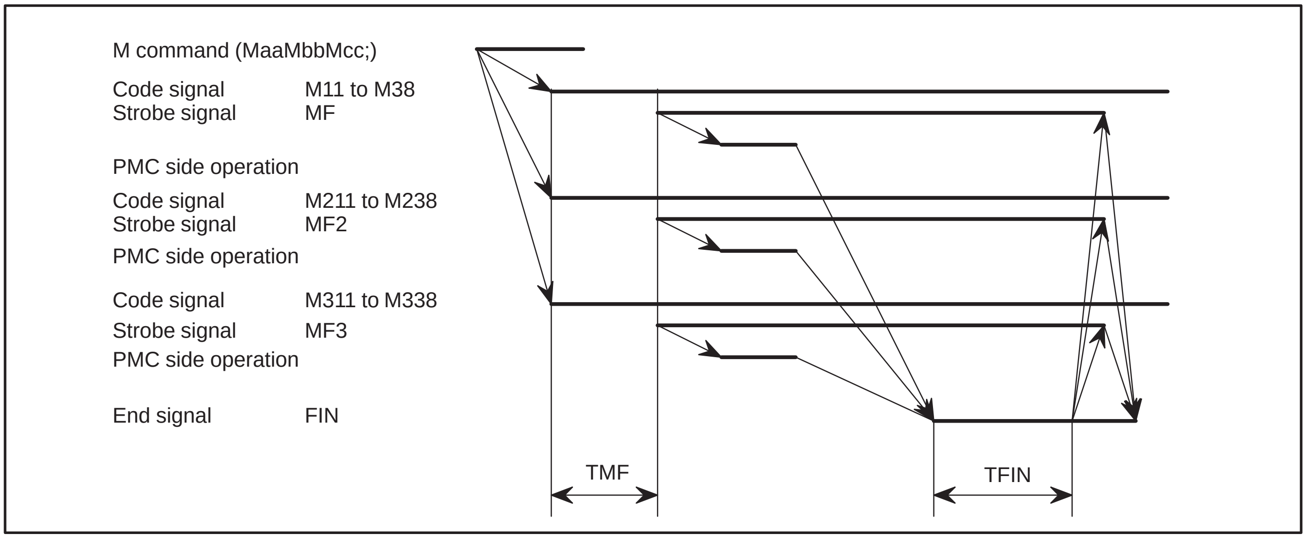
(3) On the PMC side, the code signals corresponding to the respective strobe signals are read when the strobe signals become “1”, and the appropriate operations are performed.

(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.0009#0 – #3 (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 M211 to M238 <F193#0 to F194#3> M311 to M338 <F194#4 to F195#7> 2nd, 3rd M Function strobe signal MF2<F157#4> MF3<F157#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
F157			MF3	MF2				
F193	M228	M224	M222	M221	M218	M214	M212	M211
F194	M318	M314	M312	M311	M238	M234	M232	M231
F195	M338	M334	M332	M331	M328	M324	M322	M321

Parameter

		#7	#6	#5	#4	#3	#2	#1	#0
0065		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 9003 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 send only the M code signals to the PMC side.

3 In MDI operation of type A, only a single M code can be specified.

Note

NOTE

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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.11.2	MULTIPLE M COMMANDS IN A SINGLE BLOCK
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 HSIF (No.0045#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 M11 to M38.
After the expiration of the time set with bits 4 to 7 of parameter 0009, the system inverts the logic level of strobe signal MF. That is, the MF signal is brought to 1 from 0, or to 0 from 1.

(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) 1 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.

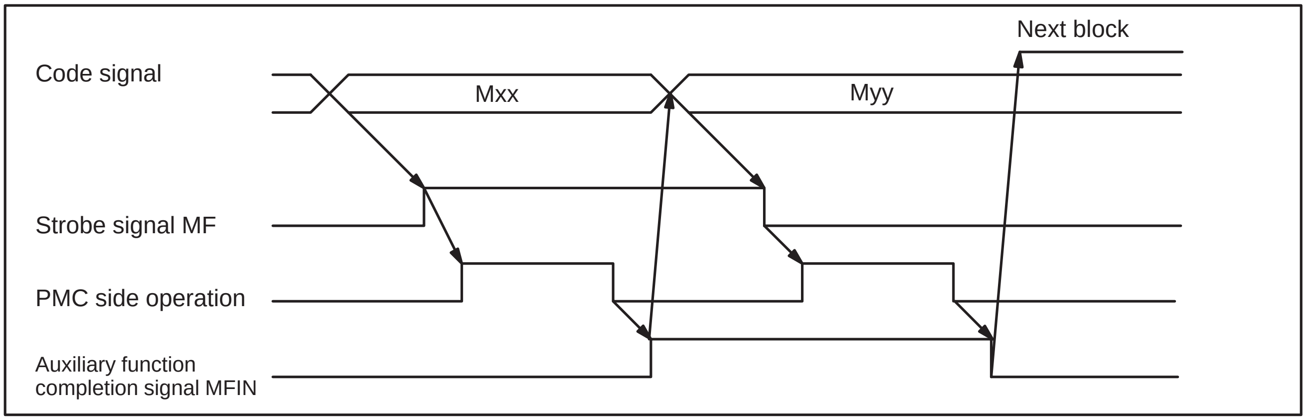


Fig. 8.4 (a) Timing Chart of the High–Speed System

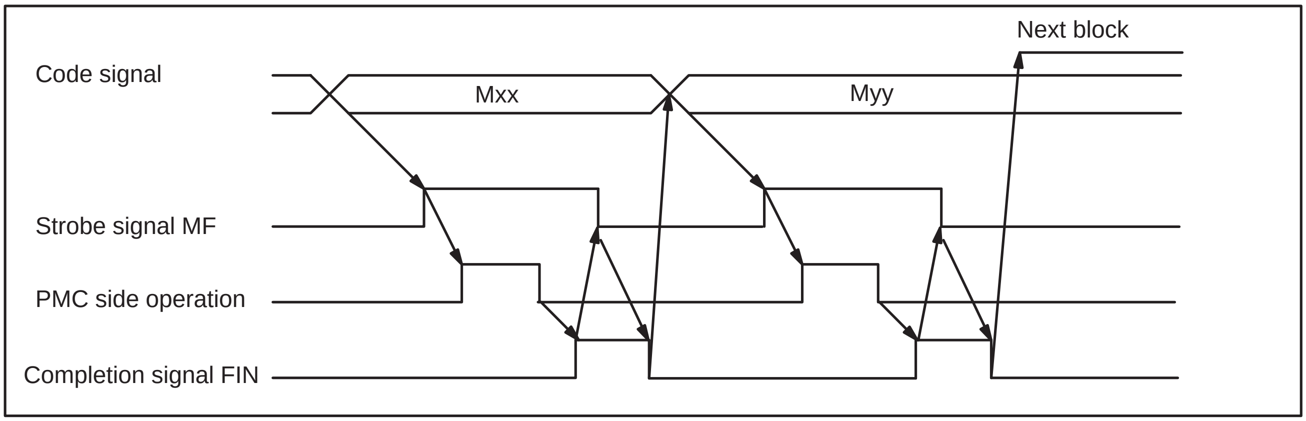


Fig. 8.4 (b) Timing Chart of the Usual System

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<G115#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<G115#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<G115#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<G115#7>(Tseries) BFIN1, BFIN2 <G115#6, #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 <G134#4, #5>

[Classification] Input signal

[Function] Indicate that when the high–speed interface is used for multiple M commands per block, the second or third M function 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) EF<F150#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<G115#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	
G115	BFIN				TFIN	SFIN		MFIN	(T series)
G115	BFIN1	BFIN2			TFIN	SFIN	EFIN	MFIN	(M series)
G134			MFIN3	MFIN2					

	#7	#6	#5	#4	#3	#2	#1	#0	
F150							EF		(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0045	HSIF							

[Data type] Bit

HSIF 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
	11.9	EXTERNAL OPERATION FUNCTION

8.5 WAITING M CODE (0–TTC)

General

Control based on M codes is used to cause one tool post to wait for the other during machining. By specifying an M code in a machining program for each tool post, the two tool posts can wait for each other at a specified block. When an M code for waiting is specified in a block for one tool post during automatic operation, the other tool post waits for the same M code to be specified before staring the execution of the next block. A range of M codes used as M codes for waiting is to be set in the parameters (Nos. 0047#4 and 0243) beforehand.

Signal

No–wait signal NOWT <G133#1>

[Classification] Input signal

[Function] Specifies whether to synchronize the tool posts by the waiting M code.

[Operation] When this signal turns to “1” the paths are not synchronized by the M code. The M code for waiting specified in a machining program is ignored.
When this signal turns to “0”, the tool posts are synchronized by the M code. When the M code for waiting is specified for one tool post, the CNC waits for the corresponding M code of another tool post to be issued, then starts executing the next block.

Waiting signal WATO <F160#6>

[Classification] Output signal

[Function] Indicates that the CNC is waiting for the M code of either tool post 1 or 2.

[Operation] This signal is “1” as long as:

(i) One tool post is waiting for another tool post. That is, the signal stays “1” for the period from when the M code for waiting is issued to one tool post to when the corresponding M code is issued to another tool post.

This signal is “0” as long as:

(ii) Neither of the tool posts are waiting for the other.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G133							NOWT	

	#7	#6	#5	#4	#3	#2	#1	#0
F160		WATO #1						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0047				M3LMT				

M3LMT The range of the waiting M codes to be used with the 0–TTC is determined by value α specified in parameter 0243, as indicated below:

1: M($\alpha \times 100$) to M999.

0: M($\alpha \times 100$) to M($\alpha \times 100 + 99$).

0243	Third (hundreds) digit of the waiting M code

[Valid data range] 1 to 9

[Description] Set the waiting M codes for tool posts 1 and 2 of the FANUC Series 0–TTC. Specify the third (hundreds) digit.

Example Specifying 3

When the M3LMT bit (bit 4 of parameter 0047) is set to 0

Normal M codes: M000 to M299, M401 to M999
Waiting M codes: M300 to M399

When the M3LMT bit (bit 4 of parameter 0047) is set to 1

Normal M codes: M000 to M299
Waiting M codes: M300 to M999

NOTE
In this parameter, set identical values for both tool posts.

Alarm and message

Number	Message	Description
160	MISMATCH WAITING M–CODE	Different M code is commanded in tool post 1 and 2 as waiting M code. Modify the program.

Caution

CAUTION
The waiting M code, unlike other M codes, does not issue code signal nor strobe signal.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.24.2	WAITING FOR TOOL POST

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.

Signal

Refer to 8.1.

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/analog spindle output is used, refer to 9.3.

9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT

9.2.1 General

There are two types of spindle motor control interfaces, spindle serial output and spindle analog output.

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.	Second serial spindle The PC can be used. (*)	Analog spindle The PC cannot be used.
○	×	Serial spindle The PC can be used.	First analog spindle The PC can be used. (*)	Second analog spindle The PC can not be used.
×	○	First analog spindle The PC can be used.	Second analog spindle The PC can not be used. (*)	Third andlog spindle The PC can not 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 of the second spindle (T series only).

· See section 15.4 or 9.10 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 rotation (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	○	×	×	×
Multispindle (T series) (*2)	○ (First spindle)	○ (Second spindle)	×	○ (Third spindle)
Rigid tapping	○	○ (*1)	○	×
Spindle synchronization	○ Master (*3)	○ Slave (*3)	×	×
Spindle control unit functions (*4), such as spindle orientation, spindle output switch, and other types of spindle switching	○	○	○	○
Polygon turning (T series)	○	○ (*1)	○	×
Spindle output control using PMC	○	○	○	○

NOTE

1(*1) The multispindle function (T series) is necessary. The function cannot be used for the first and second spindles simultaneously.

2(*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(*3) For 0–TTC, the first spindle on tool post 1 is the master, and the first spindle on tool post 2 is the slave. The second spindle of either tool post cannot be used in spindle synchronization.

4(*4) 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 (6000 to 6888/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 G229 to G232 and F281 to F284: Addresses for the first spindle G233 to G236 and F285 to F288: 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

<G229 to G232> (input), <F281 to F284>
→ for the first serial spindle
<G233 to G236> (input), <F285 to F288>
→ 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 AC SPINDLE MOTOR series (Serial Interface) Descriptions (B–65042E)
FANUC AC SPINDLE SERVO UNIT Serial Interface S series Maintenance Manual (B–65045E)
FANUC SERVO AMPLIFIER α series Descriptions (B–65162E)

Signal address

- **For 1st SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231						NRROA	ROTAA	INDXA

	#7	#6	#5	#4	#3	#2	#1	#0
F281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282					PCFNA	PCHPA	CFINA	CHPA

- **For 2ND SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G233	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G235						NRROB	ROTAB	INDXB

	#7	#6	#5	#4	#3	#2	#1	#0
F285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286					PCFNB	PCHPB	CFINB	CHPB

Parameter

- **Connection of serial spindle control unit**

	#7	#6	#5	#4	#3	#2	#1	#0
0071				SRLS2P				

[Data type] Bit

SRLS2P 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.

- **Parameters of serial spindle control unit**

Nos.6500 – 6635: S1 → For 1st serieal spindle
Nos.6640 – 6775: 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 series PARAMETER MANUAL (B–65160E)

FANUC AC SPINDLE SERVO UNIT Serial Interface S series Maintenance Manual (B–65045E)

Alarm and message

Number	Message	Description
408	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) 4) The second spindle (when SRL2SP, bit 4 of parameter No.0071, is 1) is in one of the above conditions 1) to 3). Note) This alarm does not occur after the system including the spindle control unit is activated.
409	SPINDLE ALARM DETECTION	A spindle amplifier alarm occurred in a system with a serial spindle. The alarm is indicated as "AL–XX" (where XX is a number) on the display of the spindle amplifier. For details, see Manuals for Serial spindle. Setting bit 7 of parameter No.0397 causes the spindle amplifier alarm number to appear on the screen.

DIAGNOSIS SCREEN

- **Load and speed meter readings for the serial spindle**

750	First serial spindle: Load meter reading (%)
751	First serial spindle: Speed meter reading (rpm)
752	Second serial spindle: Load meter reading (%)
753	Second serial spindle: Speed meter reading (rpm)

To display the load and speed meter readings, the following parameters must be specified correctly.

Maximum motor speed:
Parameter Nos.6520 and 6660 (main)/Nos.6160 and 6340 (sub)

Load meter reading at maximum output:
Parameter Nos.6627 and 6767 (main)/Nos.6238 and 6418 (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**

754	Master spindle motion error during spindle synchronization
755	Slave spindle motion error during spindle synchronization
756	Absolute value of synchronization error during spindle synchronization

The display for parameter Nos. 754 to 756 are in pulse units (one pulse = 360/4096 degrees)

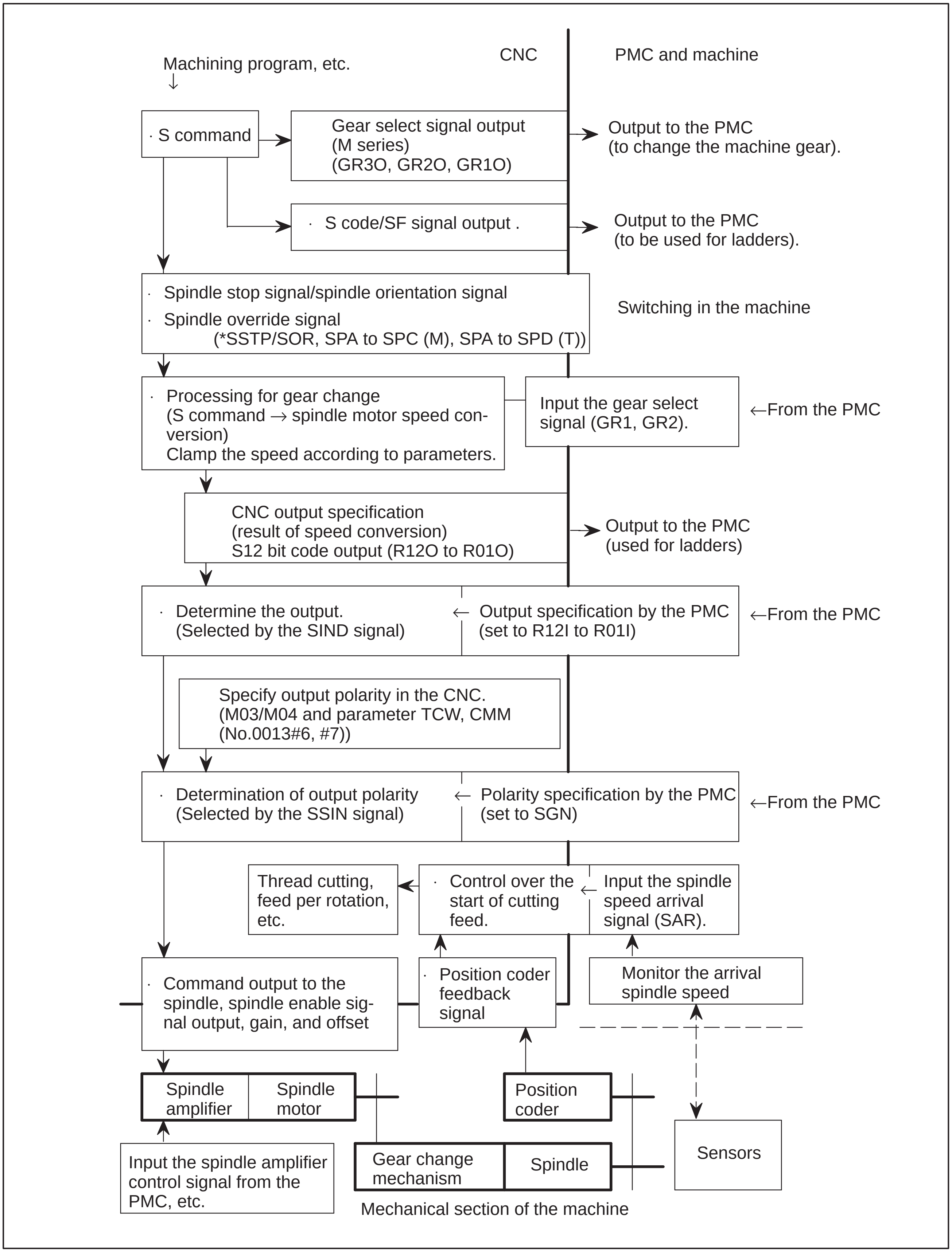
9.3 SPINDLE SPEED CONTROL

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 without the constant surface speed control option, the CNC specifies the gear stage for the desired spindle speed to the PMC according to parameter Nos. 3741, 3742, and 3743, and the S command.

(GR3O, GR2O, GR1O <F152#2, #1, #0>)

- **S code/SF signal output**

With the spindle serial output or spindle analog output option, the spindle control function in the CNC converts the S command value to the output value for the spindle motor. Unlike a system without such options, the system with the options responds to the S command with the S code/SF signals as follows to enable gear change and constant surface speed control:

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.)

- **Spindle stop signal (*SSTP)**

This signal sets the S command value in the CNC to 0. If the CNC has the spindle output specified (see descriptions on the SIND signal), this signal sets the speed command for the spindle to 0.

Even if the function of 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 (ORCW) of parameter No.0013 at a constant speed specified by parameter No.0108.

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.

In the M series, specifying bit 5 (GST) of parameter No.0003 causes the spindle motor to rotate at constant speed. This function can be used for gear shifting because it maintains a constant speed of the gear change mechanism.

- **Spindle speed override signal (SPA to SPD (T series)/ SPA to SPC (M series))**

This signal specifies an override of 50% to 200% (T series)/50% to 120% (M series) 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 (G84, G74) for the M series
Thread cutting (G32, G92, G76) for the T series

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:

M type
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).

T type
The gear stage (one of the four gear stages) being currently used by the machine is input in response to the gear select signal inputs (GR1, GR2). The machine determines which gear to use.
The CNC outputs the spindle speed that corresponds to the gear stage input.

- **Selection of gear change system**

The M series system can use either M or T type.

M type ← Without constant surface speed control option.

T type ← With constant surface speed control.

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. 0541, 0539, and 0555, 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 SFOUT no. 0020#7).
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. 9.3.1 (a))

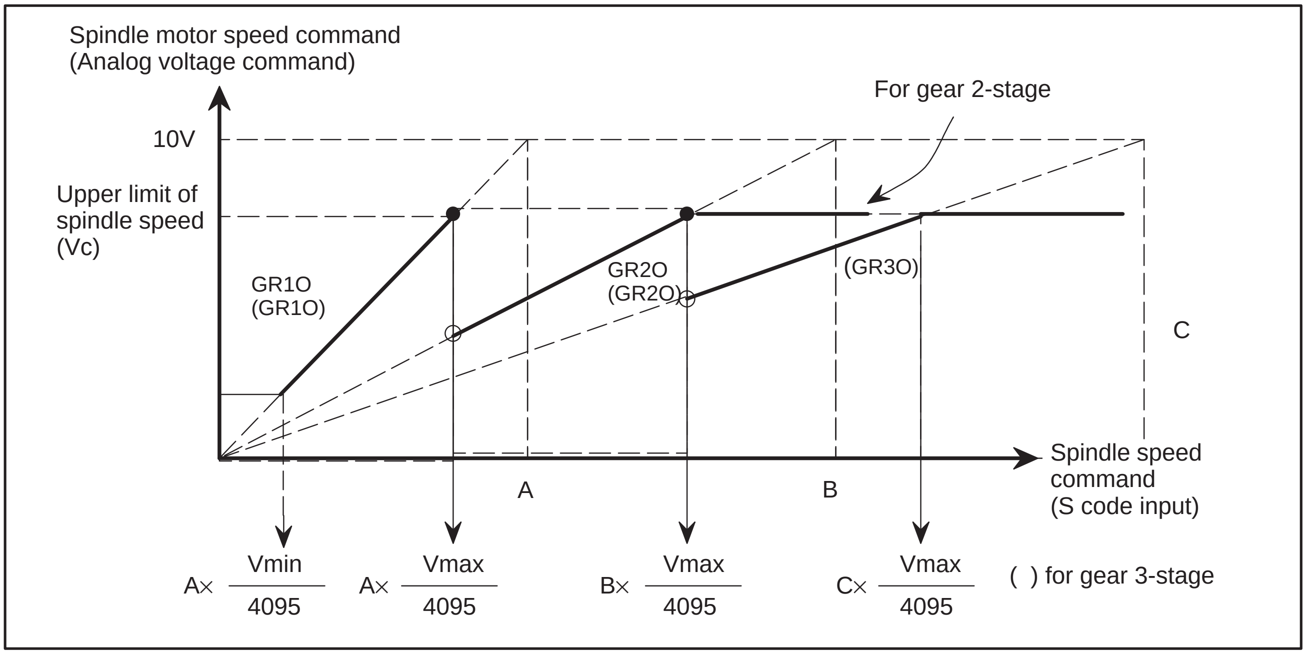


Fig.9.3.1 (a) S code input and output voltage

Set the following values as parameters:

· Constant Vmax: for upper limit of spindle speed (parameter No.0542)

$$\text{Vmax}=4095\times\frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Constant Vmin; for lower limit of spindle speed (parameter No.0543)

$$\text{Vmin}=4095\times\frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Spindle speed A (rpm) ; at command voltage 10V and low gear (parameter No.0541)

· Spindle speed B (rpm) ; at command voltage 10V and high gear (or middle-high gear) (parameter No.0539)

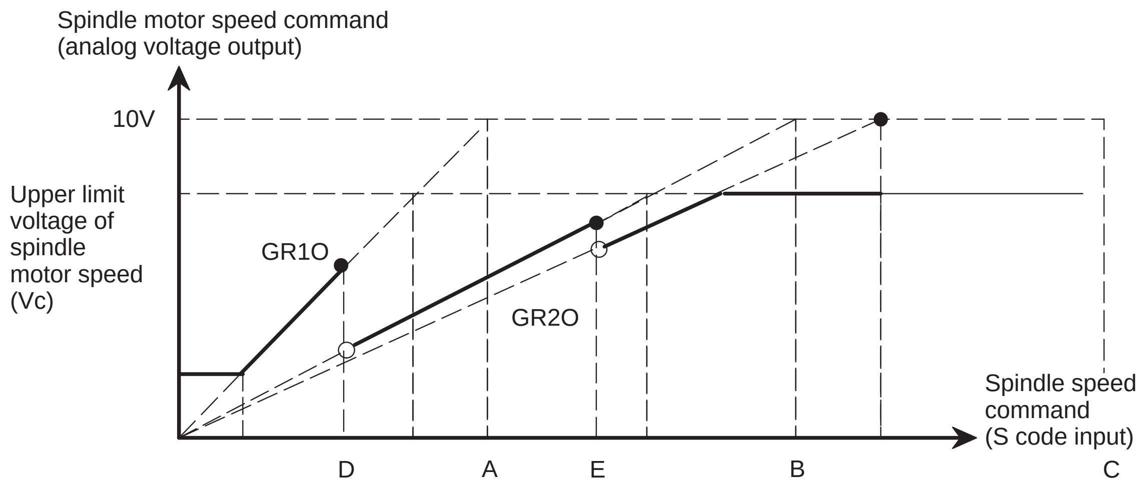
· Spindle speed C (rpm) ; at command voltage 10V and high gear (parameter No.0555) (for gear 3–stage)

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 (G3O, G2O, G1O) are output to the spindle motor as shown in Fig.9.3.1 (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 G84S (No.0012#6). In this case, gear shift (high–speed/low–speed) is performed at the speed set by parameter Nos.0540 and 0556 (Fig. 9.3.1 (b)).



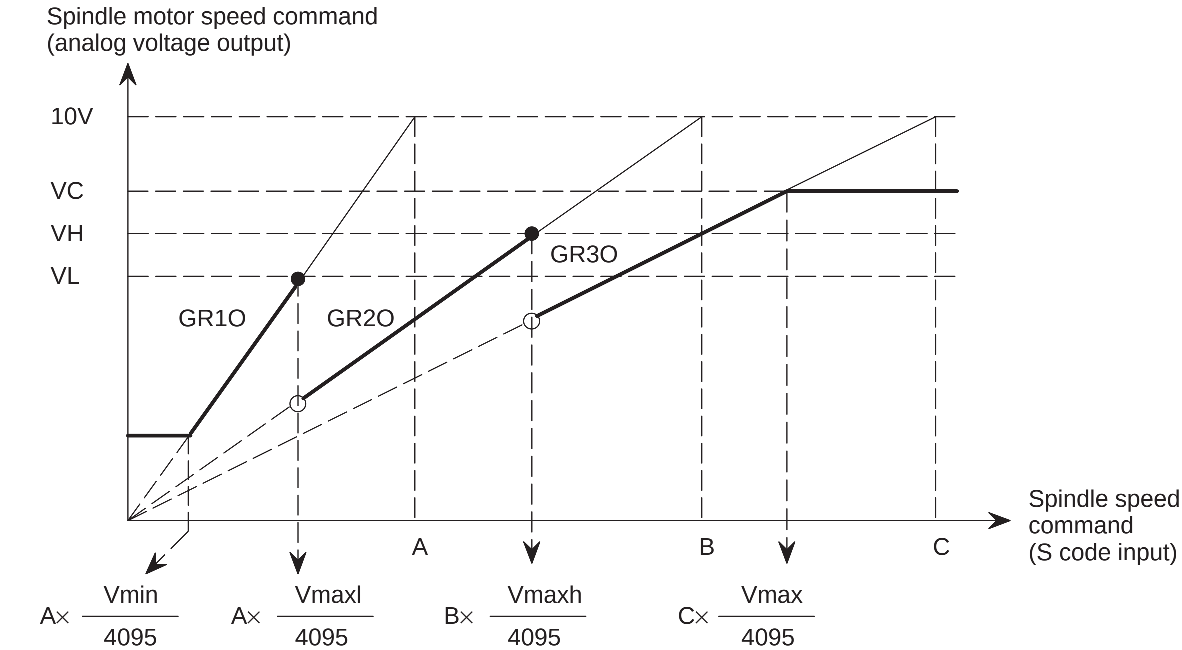
D : Gear shift point (parameter No.0540) for low/high speed change (or low/medium speed change) (set by parameter as rpm value) during tapping.

E : Gear shift point (parameter No.0556) (3-stage gear) for medium/high speed change (set by parameter as rpm value) during tapping.

Fig.9.3.1 (b) S code input and output analog voltage (in tapping)

- M type gear change method B (M series) (Fig. 9.3.1 (c))

The speed (rpm) at which the low-speed and the high-speed gears are changed over can be set as a parameter (No.0585) by setting parameter LGCM No.0035#6. When a 3-step gear is used, it is possible to set the speeds (rpm) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters Nos.0585, 0586.



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-step gear)

VL: Voltage corresponding to the upper limit of output value to spindle motor at low-speed gear

Fig.9.3.1 (c) M type gear change B

When using this function, set the following parameters:

· Constant Vmax (Parameter No.0542) related to the upper limit of spindle motor speed (rpm)

$$\text{Vmax} = 4095\times \frac{\text{Upper limit of spindle motor speed (rpm)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmin (Parameter No.0543) related to the lower limit of spindle motor speed (rpm)

$$\text{Vmin} = 4095\times \frac{\text{Lower limit of spindle motor speed (rpm)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmaxl (Parameter No.0585) related to the upper limit of spindle motor speed (rpm) with low-speed gears

$$\text{Vmaxl} = 4095\times \frac{\text{Upper limit of spindle motor speed (rpm) with low-speed gears}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Constant Vmaxh (Parameter No.0586) related to the upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-step gear)

$$\mathrm{Vmaxh} = 4095 \times \frac{\text{Upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-step gear)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Spindle speed A (Parameter No.0541) (rpm) with low-speed gears when the command voltage is 10V

· Spindle speed B (Parameter No.0539) (rpm) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-step)

· Spindle speed C (Parameter No.0555) (rpm) with high-speed gears when the command voltage is 10V (3-step 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 G84S (No.0012#6) is set, the gears are changed over at the gear changing point for tapping.
> 2 For this function (parameter LGCM=1 (No.0035#6)), when only one-step gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-step gear is used, it is calculated according to Vmaxh. Therefore, set Vmaxl when only one-step gear is used, Vmaxl and Vmaxh when 2-step gear is used.

- **Time chart**

When S code is commanded, the I/O signal time chart is :

- When Gear select signal does not change

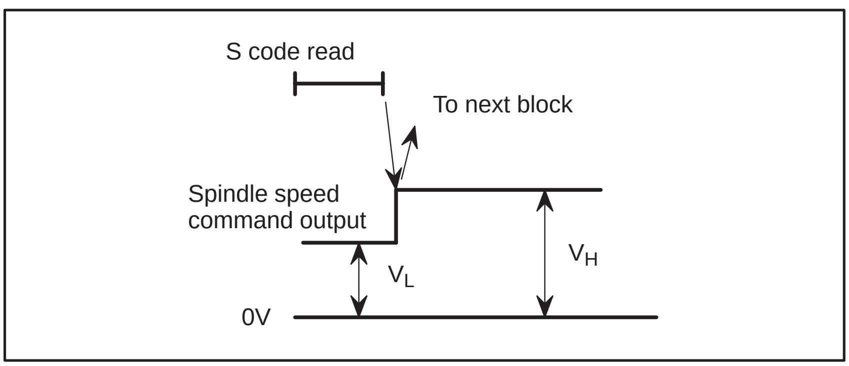


Fig. 9.3.1 (d) Time chart when gear select signal does not change

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

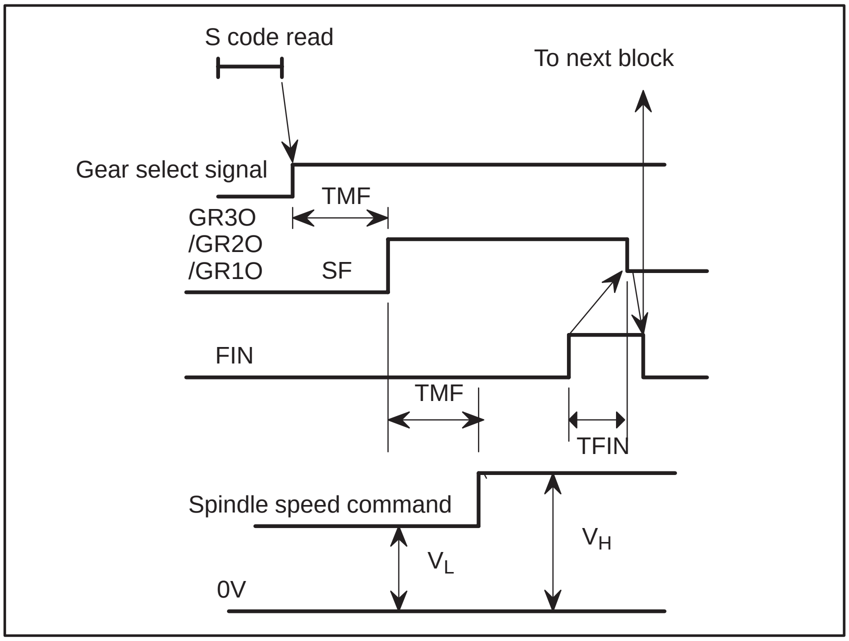


Fig.9.3.1 (e) Time chart when gear select signal changes

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.0009#4 to #7, is common to M, S and T functions.

Moreover, specifying bit 7 (SFOUT) of parameter No.0020 can specify that the SF signal be output even if no gear change is used.

- **Details of T type (Input of GR1, GR2)**

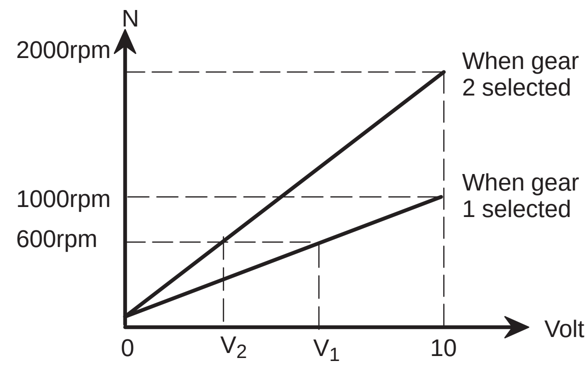
To perform the T type gear changing, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameter Nos. 0540 – 0543.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear No.	Parameter No. for max. spindle speed
0	0	1	No.0540
1	0	2	No.0541
0	1	3	No.0542
1	1	4	No.0543

The following descriptions exemplify 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 rpm for the low speed gear (G1) and 2000 rpm for the high speed gear (G2), set these speeds by the parameter Nos.0540, 0541. 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.

V1: 6(V)
V2: 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

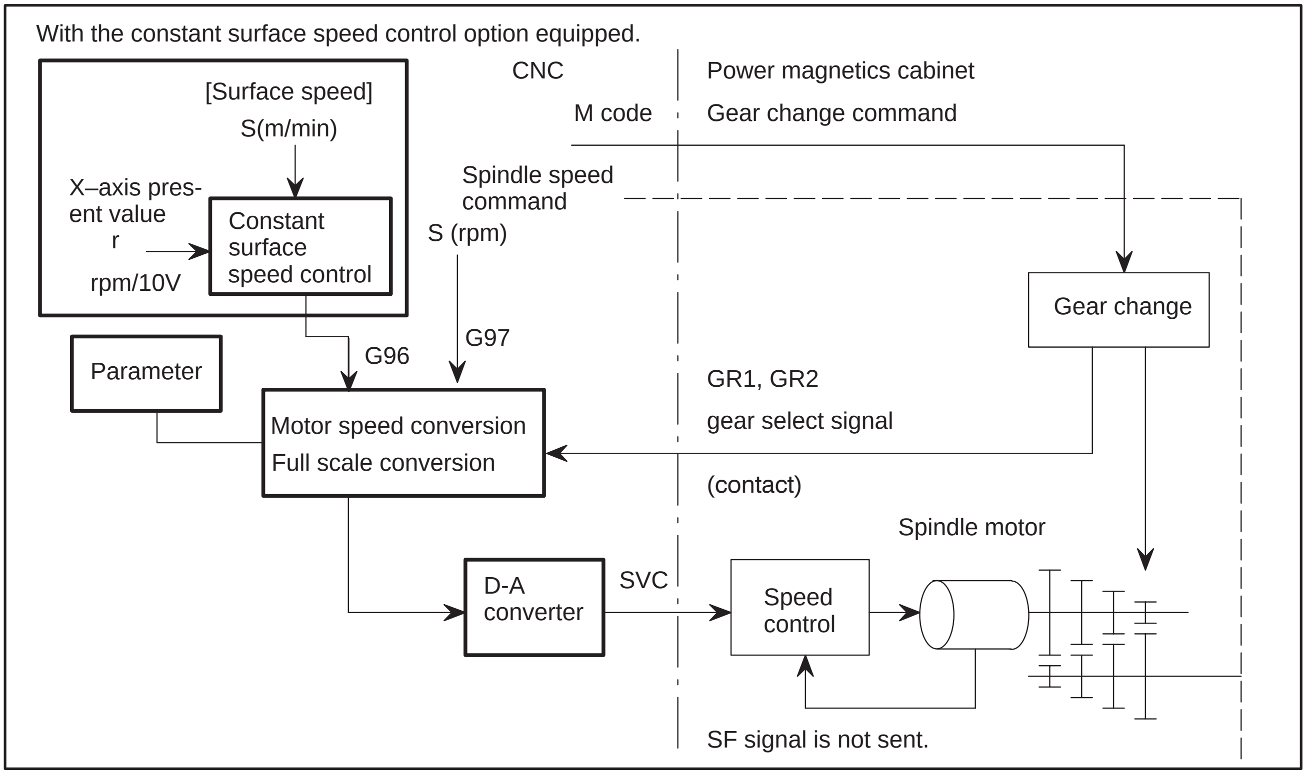
N: Spindle speed given by S5-digits

This is equivalent to the G97 mode for constant surface speed control.

See Section 9.4 for operations during the constant surface speed control mode (G96).

In addition, parameter No.0556 (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.

(R01O to R12O<F172#0 to F173#3>)

After the calculation results are received, the SIND signal <G125#7> determines which is to be used, the speed command output calculated by the CNC or the data specified in the PMC. Thus speed command output control for 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 actually used output polarity is determined by the CNC as follows:

· If bit 7 (TCW) of parameter No.0013 = 0
  → Determined according to bit 6 (CWM) of parameter No.0013

· If bit 7 (TCW) of parameter No.0013 = 1
  → Determined according to bit 6 (CWM) of parameter No.0013 and M03/M04 given to the CNC

After that, the SSIN signal <G125#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 output polarity of the speed command output to the spindle motor is determined. (See also Section 15.4.)

Keep in mind the following: Even with bit 7 (TCW) of parameter No.0013 = 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 output 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.0013 = 1, no command output is sent to the spindle, because the output polarity is not determined until an M03/M04 is issued.

- **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 <F149#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.0516
Offset voltage compensation: Parameter No.0577 (M)/0539 (T)

· Analog spindle as the second spindle
→ Gain adjustment data: Parameter No.0613
Offset voltage compensation: Parameter No.0614

· Analog spindle as the third spindle
→ Gain adjustment data: Parameter No.0617
(valid for multispindle control)
Offset voltage compensation: Parameter No.0618

- **Position coder feedback signal**

The position coder is necessary for thread cutting or feed per rotation. (For the M series, a software option must also be purchased.)

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 6 and 7 (PSG1, PSG2) of parameter 0028 (M series)/bits 6 and 7 (PSG1, PSG2) of parameter 0003 (T series).

See Section 9.11 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 to cause 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 using the PC ladder regularly, 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.0110 after the S command or cutting feed command was issued.

When using the SAR signal, set bit 2 (SCTO) of parameter No.0024 to 1.

Bit 6 (CSCT) of parameter No.0700 on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

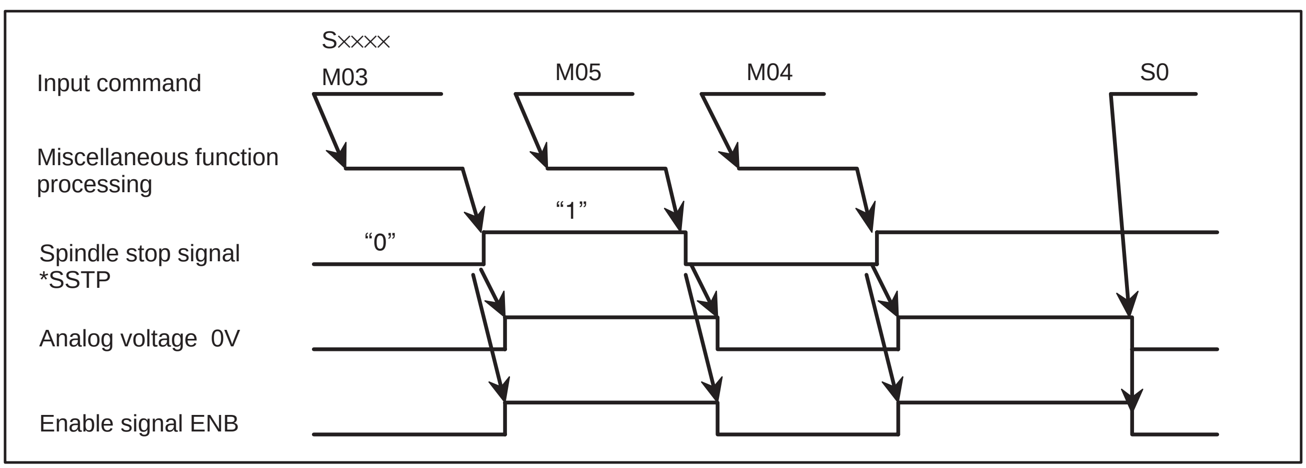
Signal

Spindle stop signal *SSTP<G120#6>

[Classification] Input signal

[Function] The command output to the spindle is held.

[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<G120#5>

[Classification] Input signal

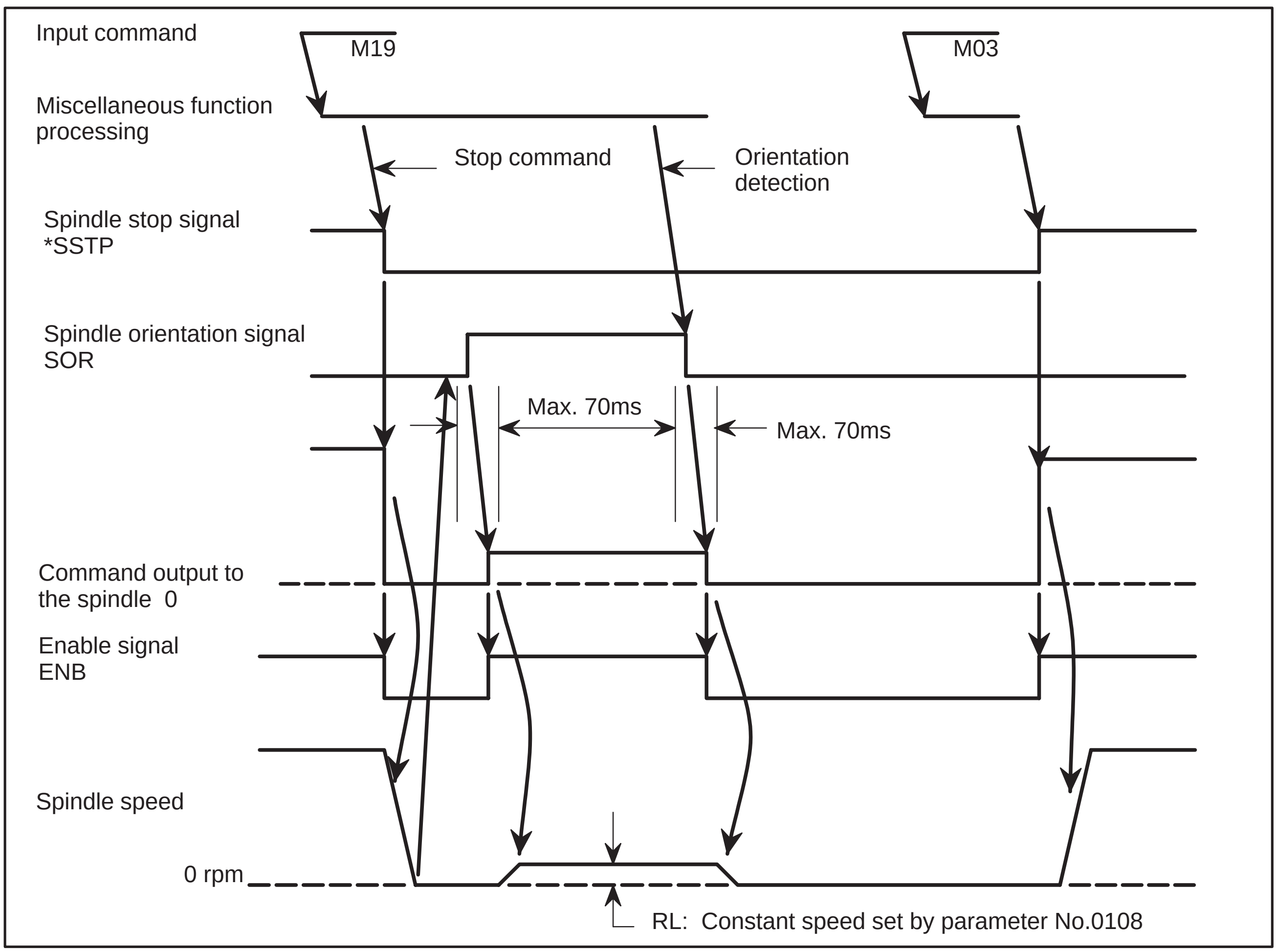
[Function] The spindle or the spindle motor is rotated at a constant speed.

[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.0108 is output. The enable signal ENB also turns to "1". This signal is disabled when the spindle stop signal is "1".

In M series, when the spindle speed for orientation is set by parameter GST No.0003#5 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 0108 with an output polarity set by parameter ORCW (No.0013#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.0108 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.0003#5)=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 SPA to SPD<G103#2 to #5>(T series)/ SPA to SPC<G103#3 to #5>(M series)

[Classification] Input signal

[Function] The spindle speed override signal specifies an override from 50% to 200% (T)/from 50% to 120% (M) in 10% units for the S command sent to the CNC.

[Operation] (M series)

Spindle override value	Machine contact status		
	SPA	SPB	SPC
50%	1	1	1
60%	0	1	1
70%	0	1	0
80%	1	1	0
90%	1	0	0
100%	0	0	0
110%	0	0	1
120%	1	0	1

(T series)

Spindle override value	Machine contact status			
	SPA	SPB	SPC	SPD
50%	1	1	1	0
60%	0	1	1	0
70%	0	1	0	0
80%	1	1	0	0
90%	1	0	0	0
100%	0	0	0	0
110%	0	0	1	0
120%	1	0	1	0
130%	1	0	1	1
140%	1	1	1	1
150%	0	1	1	1
160%	0	1	0	1
170%	1	1	0	1
180%	1	0	0	1
190%	0	0	0	1
200%	0	0	1	1

1 : Contact is closed.
0 : Contact is opened.

The spindle speed override function is disabled (an override of 100% is applied) under the following conditions:

· Tapping cycle (G84, G74), M series
· Thread cutting (G32, G92, G78), T series

→ When this function is not in use, specify an override of 100%.

Spindle speed arrival signal SAR <G120#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.0110 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 2 (SCTO) of parameter No.0024 to 1.

The CNC checks the SAR signal under the following conditions:

(a) Bit 2 (SCTO) of parameter No.0024 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.0110 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 gated with 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<F149#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 ENB signal can be used also for serial spindle control.

Gear selection signal GR1O, GR2O, GR3O <F152#0 to #2>(M series)

[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 <G123#2, #3>(M series)/ <G118#2, #3>(T series)

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

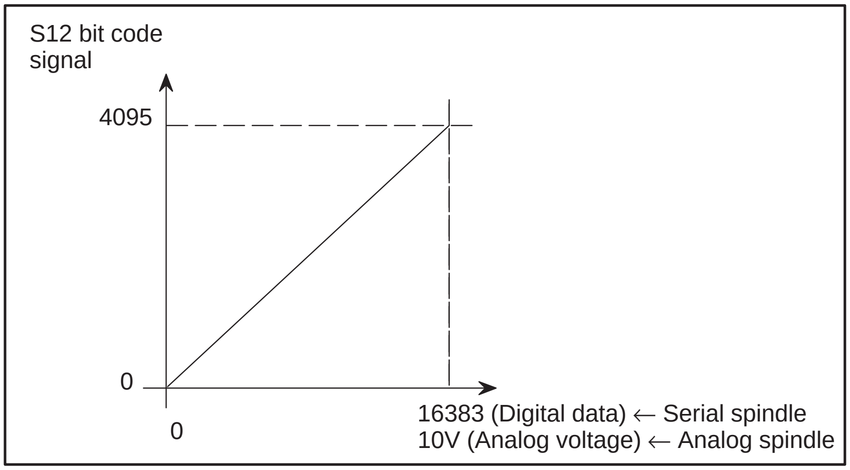
[Output condition] For details of this signal, see descriptions on the T type gear selection method in General.

S12–bit code signal R01O to R12O <F172#0 to F173#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 S11 to S28<F152> (Output) Spindle speed function strobe signal SF<F150#2>(Output)

See Sections 9.1 and 15.4 for these signals.

Spindle speed output control signal by PMC SIND<G125#7>(Input) R01I to R12I <G124#0 to G125#3> (Input) SSIN<G125#6>(Input) SGN<G125#5>(Input)

See Section 15.4 for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G103			SPC	SPB	SPA	SPD			(T series)
G103			SPC	SPB	SPA				(M series)
G120		*SSTP	SOR	SAR					
G123					GR2	GR1			
G124	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I	
G125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I	
G145			*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1	(T series)

	#7	#6	#5	#4	#3	#2	#1	#0
F149				ENB				
F150						SF		
F152	S28	S24	S22	S21	S18	S14	S12	S11
F172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0003			GST						(T series)

[Data type] Bit

GST The SOR signal is used for:
0 : Spindle orientation
1 : Gear shift

	#7	#6	#5	#4	#3	#2	#1	#0	
0003	PG2	PG1							(T series)
0028	PG2	PG1							(M series)

[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

$$\text{Magnification} = \frac{\text{Spindle speed}}{\text{Number of position coder revolutions}}$$

	#7	#6	#5	#4	#3	#2	#1	#0	
0012		G84S							(M series)

[Data type] Bit

G84S 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 0540 and 0556).

	#7	#6	#5	#4	#3	#2	#1	#0
0013	TCW	CWM	ORCW					

[Data type] Bit

ORCW 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	
0020	SFOUT								(M series)

[Data type] Bit

SFOUT 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	
0029				SFOB					(M series)

[Data type] Bit

SFOB When an S code command is issued in constant surface speed control,
0 : SF is output.
1 : SF is not output.

	#7	#6	#5	#4	#3	#2	#1	#0	
0035		LGCM							(M series)

[Data type] Bit

LGCM Gear switching method
0 : Method A (Parameters 0542 and 0543 for the maximum spindle speed at each gear are used for gear selection.)
1 : Method B (Parameters 0585 and 0586 for the spindle speed at the gear switching point are used for gear selection.)

0049	#7	#6	#5	#4	#3	#2	#1	#0	
								EVSF	(T series)

[Data type] Bit

EVSF 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.

0071	#7	#6	#5	#4	#3	#2	#1	#0	
							HISSC		(T series)

[Data type] Bit

HISSC The sampling frequency to obtain the average spindle speed
0 : 4 (Normally, set to 0.)
1 : 1

0108	The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift

[Data type] Byte

[Valid data range] 0 to 255

Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.

When GST, #5 of parameter 0003, is set to 0, set the spindle speed during spindle orientation in rpm.

When GST, #5 of parameter 0003, is set to 1, set the spindle motor speed during spindle gear shift calculated from the following formula.

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 4095 \text{ (For an analog spindle)}$$

0110	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.

0516	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) Specify the spindle speed so that the analog output of the spindle speed is the maximum voltage (10 V).

(3) Measure the output voltage.

(4) Assign the value obtained by the following equation to parameter No.0516.

$$\text{Set value} = \frac{10\ (V)}{\text{Measured data } (V)} \times 1000$$

(5) After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is the maximum voltage. Confirm that the output voltage is 10V.

> **NOTE**
> This parameter needs not to be set for serial spindles.

0539	Compensation value for the offset voltage of the analog output of the spindle speed	(T series)
0577	Compensation value for the offset voltage of the analog output of the spindle speed	(M series)

[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) Specify the spindle speed so that the analog output of the spindle speed is 0.

(3) Measure the output voltage.

(4) Assign the value obtained by the following equation to parameter No.0539 (T)/0577 (M).

$$\text{Set value} = \frac{-8191 \times \text{Offset voltage } (V)}{12.5}$$

(5) After setting the parameter, specify the spindle speed so that the analog output of the spindle speed is 0. Confirm that the output voltage is 0V.

> **NOTE**
> This parameter need not to be set for serial spindles.

0540	Maximum spindle speed for gear 1
0541	Maximum spindle speed for gear 2
0542	Maximum spindle speed for gear 3
0543	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed corresponding to each gear.

> **NOTE**
> These parameters are valid when a T type gear is selected.

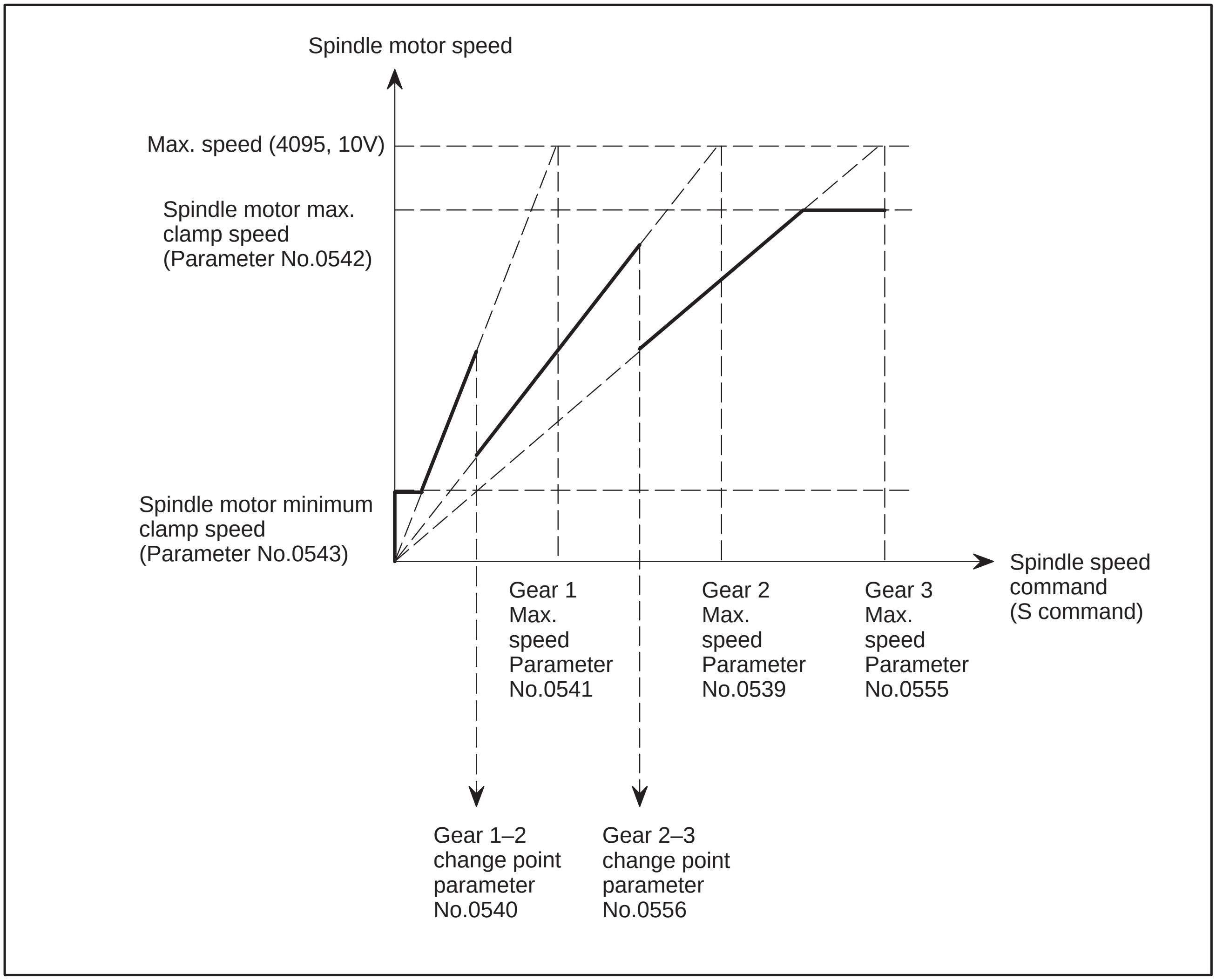
0540	Spindle speed when switching from gear 1 to gear 2 during tapping	(M series)
0556	Spindle speed when switching from gear 2 to gear 3 during tapping	(M series)

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (G84S, #6 of parameter 0012, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



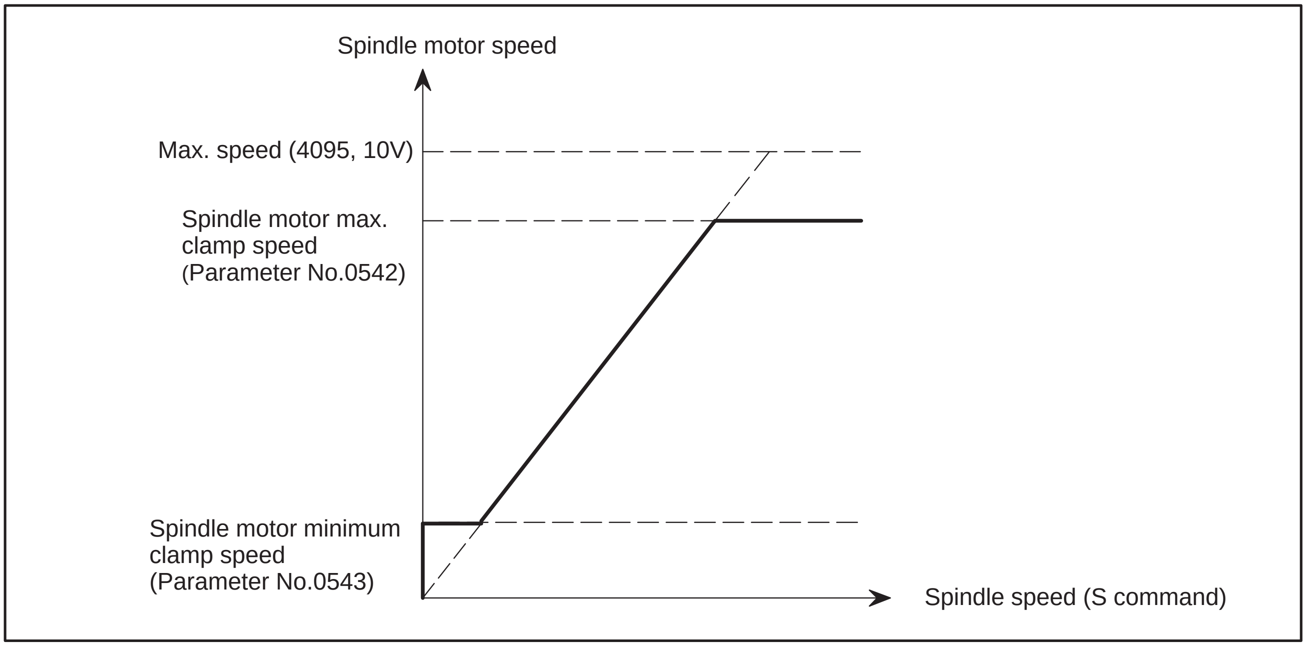
0542	Maximum clamp speed of the spindle motor	(M series)

[Data type] Word

[Valid data range] 0 to 4095

Set the maximum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Maximum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$



0543	Minimum clamp speed of the spindle motor	(M series)

[Data type] Word

[Valid data range] 0 to 4095

Set the minimum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Minimum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$

0556	Maximum spindle speed

[Data type] Word

[Unit of data] rpm

[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 at the maximum speed.

NOTE
1 In the M series, this parameter is valid when the constant surface speed control option is selected.
2 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.

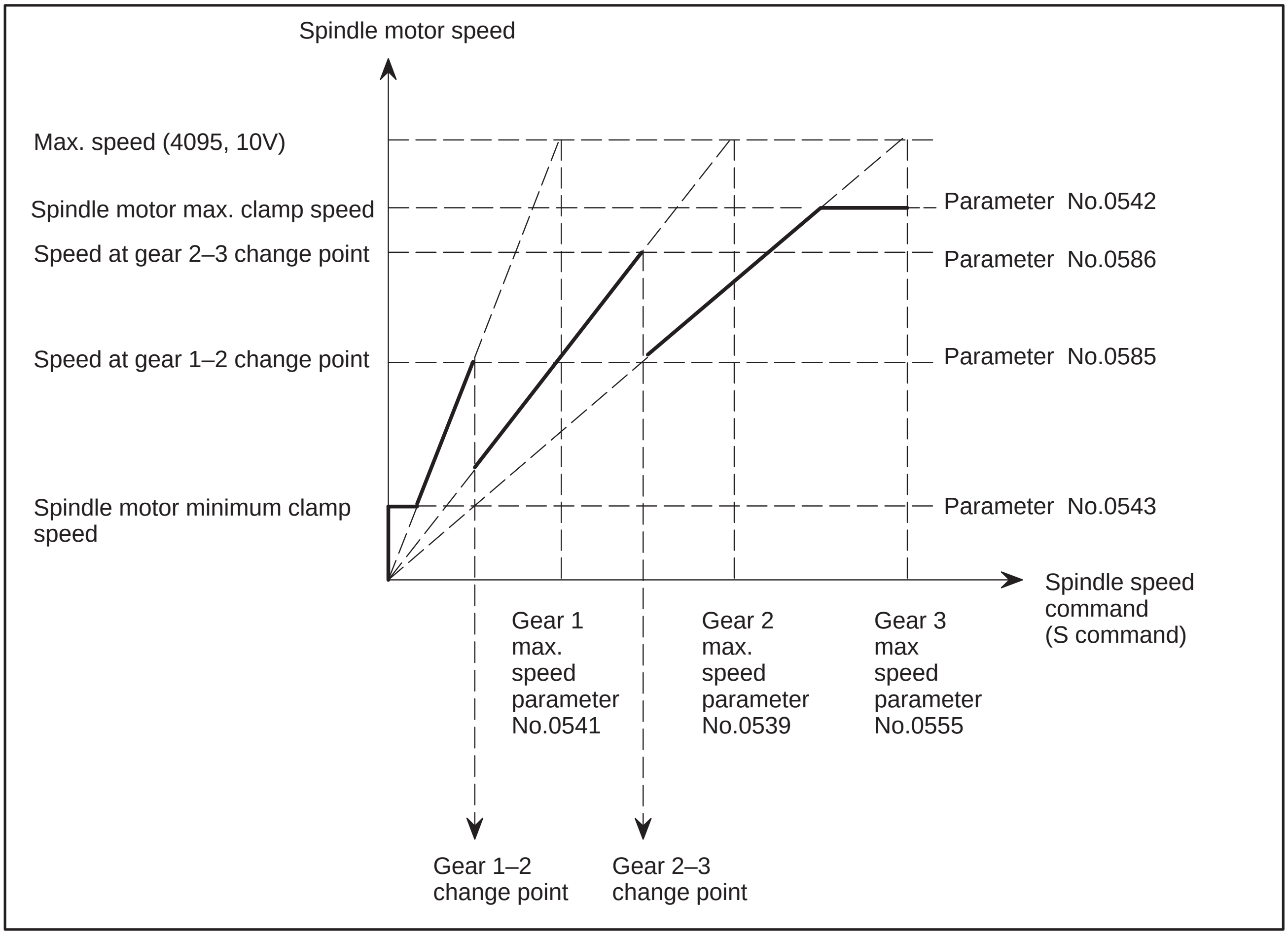
0585	Spindle motor speed when switching from gear 1 to gear 2	(M series)
0586	Spindle motor speed when switching from gear 1 to gear 3	(M series)

[Data type] Word

[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.

$$\text{Set value} = \frac{\text{Spindle motor speed when the gears are switched}}{\text{Maximum spindle motor speed}} \times 4095$$



0613	PSANG2

PSANG2 Set data used for adjusting the gain of the second spindle.
The setting method is the same as that for the corresponding parameter for the first spindle (parameter 0516).

0614	SPDLC2

SPDLC2 Set an offset compensation value for the spindle speed of the second spindle.
The setting method is the same as that for the corresponding parameter for the first spindle (parameter 0539).

0615	Maximum spindle speed for gear 1 of the second spindle	(T series)
0616	Maximum spindle speed for gear 2 of the second spindle	(T series)

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set maximum spindle speeds for the gears of the second spindle.

> **NOTE**
> These parameters are provided for multiple–spindle control.

0617	PSANG3

PSANG3 Set the data used for adjusting the gain of the third spindle.
The setting method is the same as that for the corresponding parameter for the first spindle (parameter 0516).

0618	Offset–voltage compensation value of the analog output of the third–spindle speed	(T series)

[Data type] Word

[Unit of data] Velo

[Valid data range] –1023 to 1024

Set a compensation value for the offset voltage of the analog spindle speed output of the second spindle when multiple–spindle control is applied.

0619	G3MAX1
0620	G3MAX2

G3MAX2, G3MAX1 Speeds of the first and second gears of the third spindle, respectively, when 10 V is specified in the spindle speed command

[Valid data range] 1 to 19999

[Unit of data] rpm

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 SPINDLE SPEED CONTROL FOR 0–TTC

General

In 0–TTC, the additional tool post section (tool post No. 2) can have the same spindle interface as a one–tool post lathe (see Section 9.2.).

Each spindle is controlled by a command issued by tool post 1 or 2. Which spindle is controlled by which tool post can be switched by signals.

This section describes the configuration of 0–TTC and how it is controlled.

One–spindle control and two–spindle control

In 0–TTC, there are two selectable configurations, a configuration in which the entire system uses one spindle (one–spindle control) and a configuration in which each spindle is controlled separately (two–spindle control). Parameter No. 0047#3 (2SP) is used to select a configuration.

One–spindle control

The spindle interface for tool post 2 is not used.

- **Selection of the spindle command**

The spindle command select signal SLSPA<G133#2> (input) specifies the tool post whose spindle command is to be followed by the spindle.

- **Position coder feedback signal (serial spindle)**

When a serial spindle is used, a position coder feedback signal is input to both tool posts of the NC. Either tool post can perform threading or feed per rotation.

The optical cable from the spindle amplifier of the spindle (S1/P1) is connected to connector COP5. Connector COP6 is left unconnected.

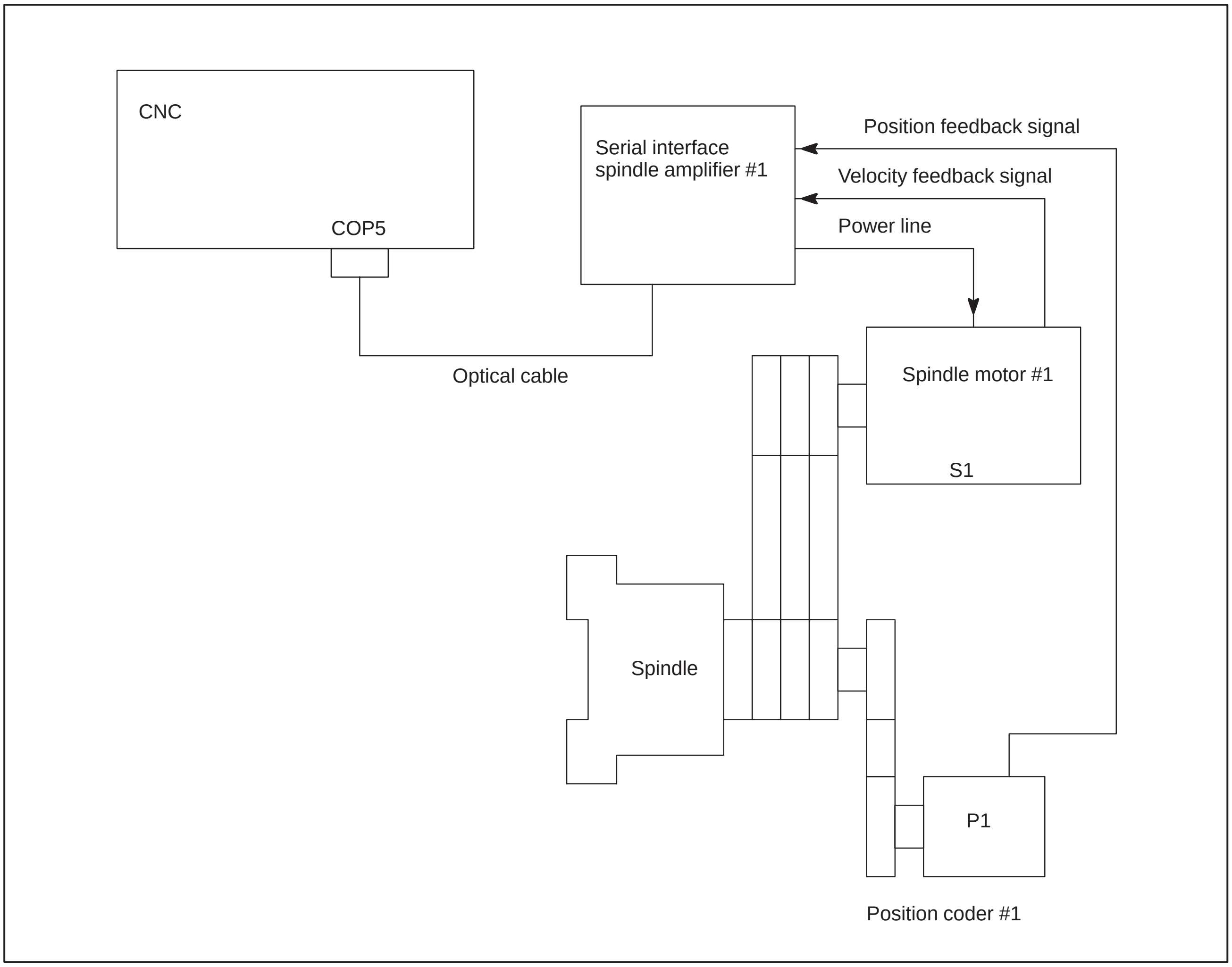


Fig 9.4 (a) One spindle control (Serial spindle)

In addition, spindle amplifier #3 can be connected to spindle amplifier #1.

- **Position coder feedback signal (Analog)**

When an analog spindle is used, supplying the position coder feedback signal to the position coder interface of tool post 2 via an external distribution circuit makes it possible to use either tool post for thread cutting and feed per rotation.

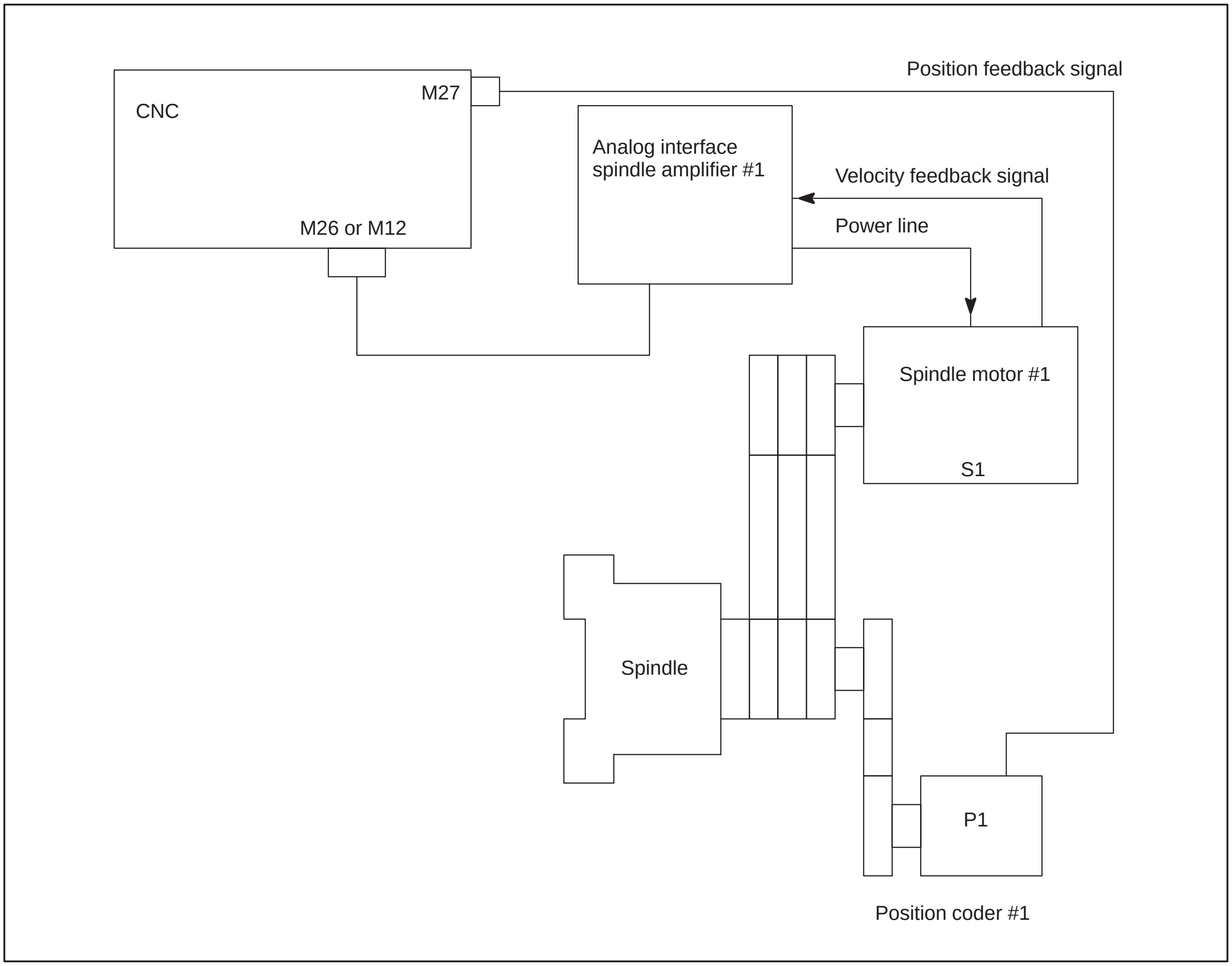


Fig 9.4 (b) One spindle control (Analog spindle)

Two–spindle control

The spindle interface for either tool post is used.

- **Selection of spindle command**

The spindle command select signals SLSPA <G133#2> and SLSPB <G133#3> (input) specify the tool post whose spindle command is to be followed by each spindle.

- **Position coder feedback signal (When both are serial spindle)**

When the serial spindles are used on both tool posts, the position coder feedback signals SLPCA<G1333#2> and SLPCB<G1333#3> (input) direct each tool post to select which spindle's position coder feedback signal is used. Therefore, it is possible to use the spindle of the other system; for example, tool post 1 can perform thread cutting or feed per rotation using the spindle connected to tool post 2.

The optical cable from the spindle amplifier of the first spindle (S1/P1) is connected to connector COP5. The optical cable from the spindle amplifier of the second spindle (S2/P2) is connected to connector COP6.

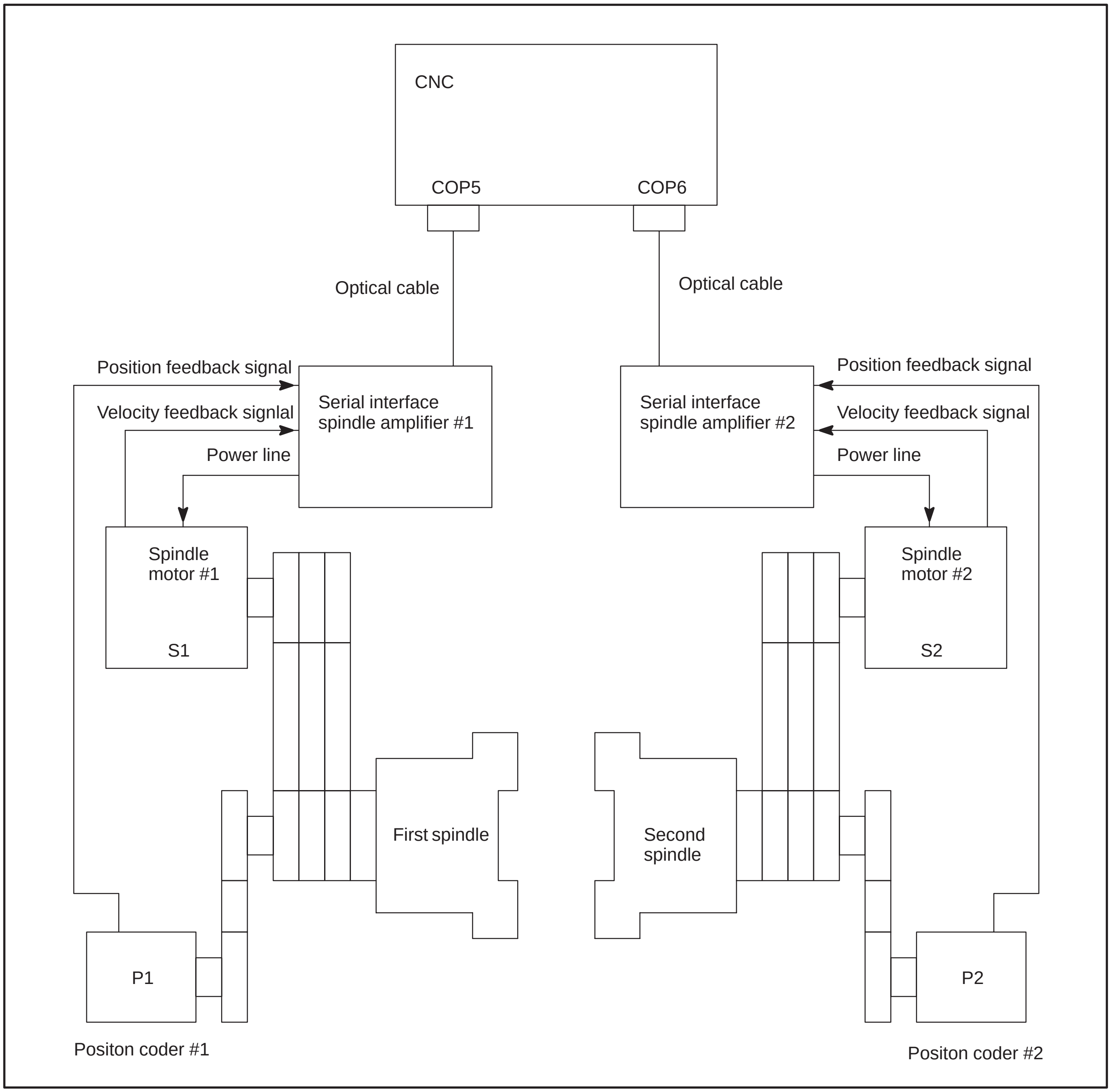


Fig 9.4 (c) Two spindle control (Serial spindle)

In addition, spindle amplifier #3 can be connected to spindle amplifier #1, and spindle amplifier #4 can be connected to spindle amplifier #2.

- **Position coder feedback signal (when an analog spindle is used)**

If either tool post uses an analog spindle as the first spindle, the spindle feedback signals SLPCA<G1333#2> and SLPCB<G1333#3> (input) cannot cause the NC to select a position coder feedback signal.

If both tool posts use an analog spindle, switching the position coder feedback signal inputs to the NC using an external circuit makes it possible to use the spindle of the other system.

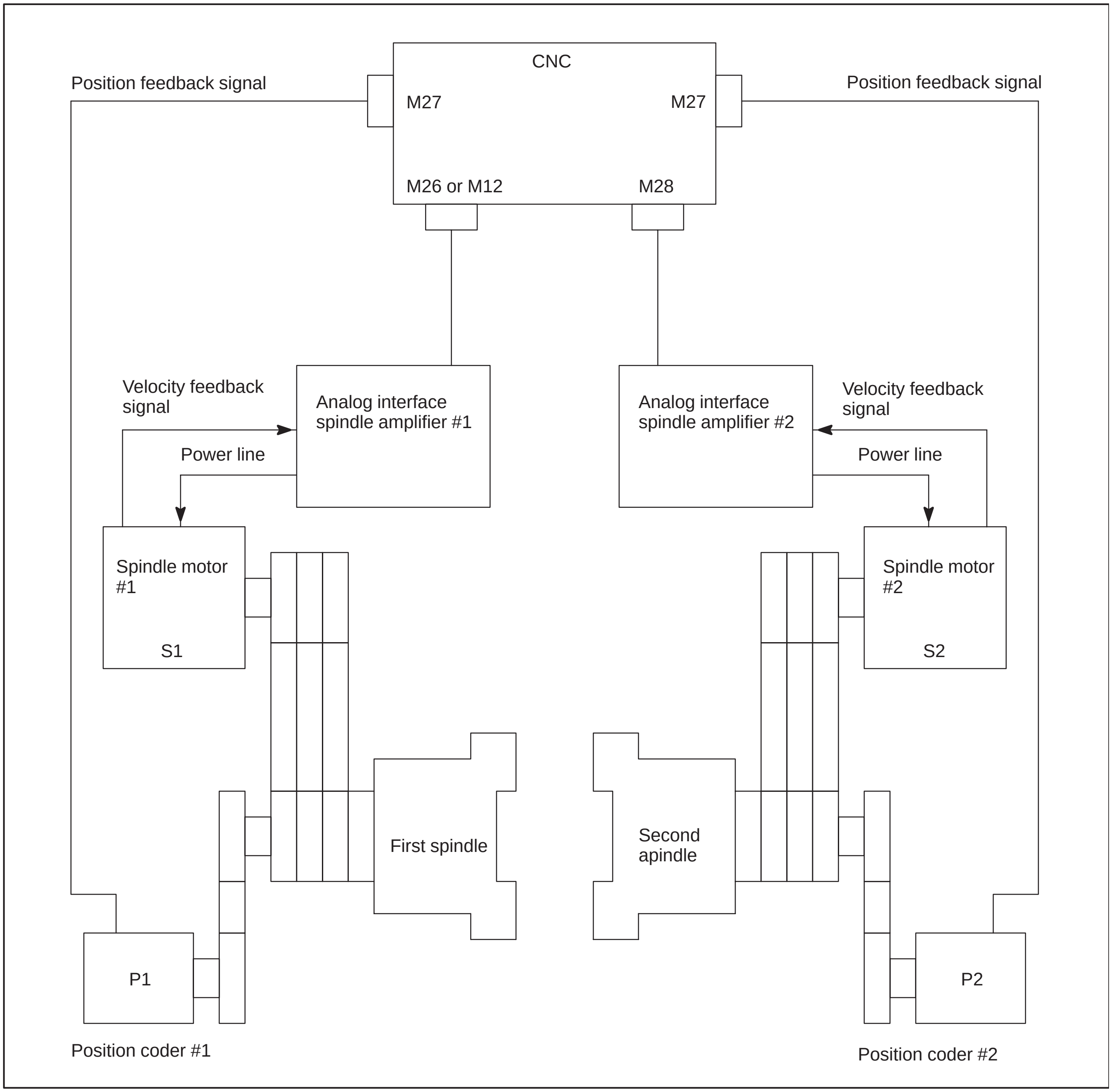


Fig 9.4 (d) Two spindle control (Analog spindle)

2nd and 3rd spindles

If the first spindle is a serial spindle, the second and third spindles can also be used in the 0–TTC. (See Section 9.2.)

In the following chart, all spindles are connected under two–spindle control.

Under one–spindle control, any spindle ($SP1^{\#2}$, $SP2^{\#2}$, $SP3^{\#2}$) of tool post 2 cannot be used.

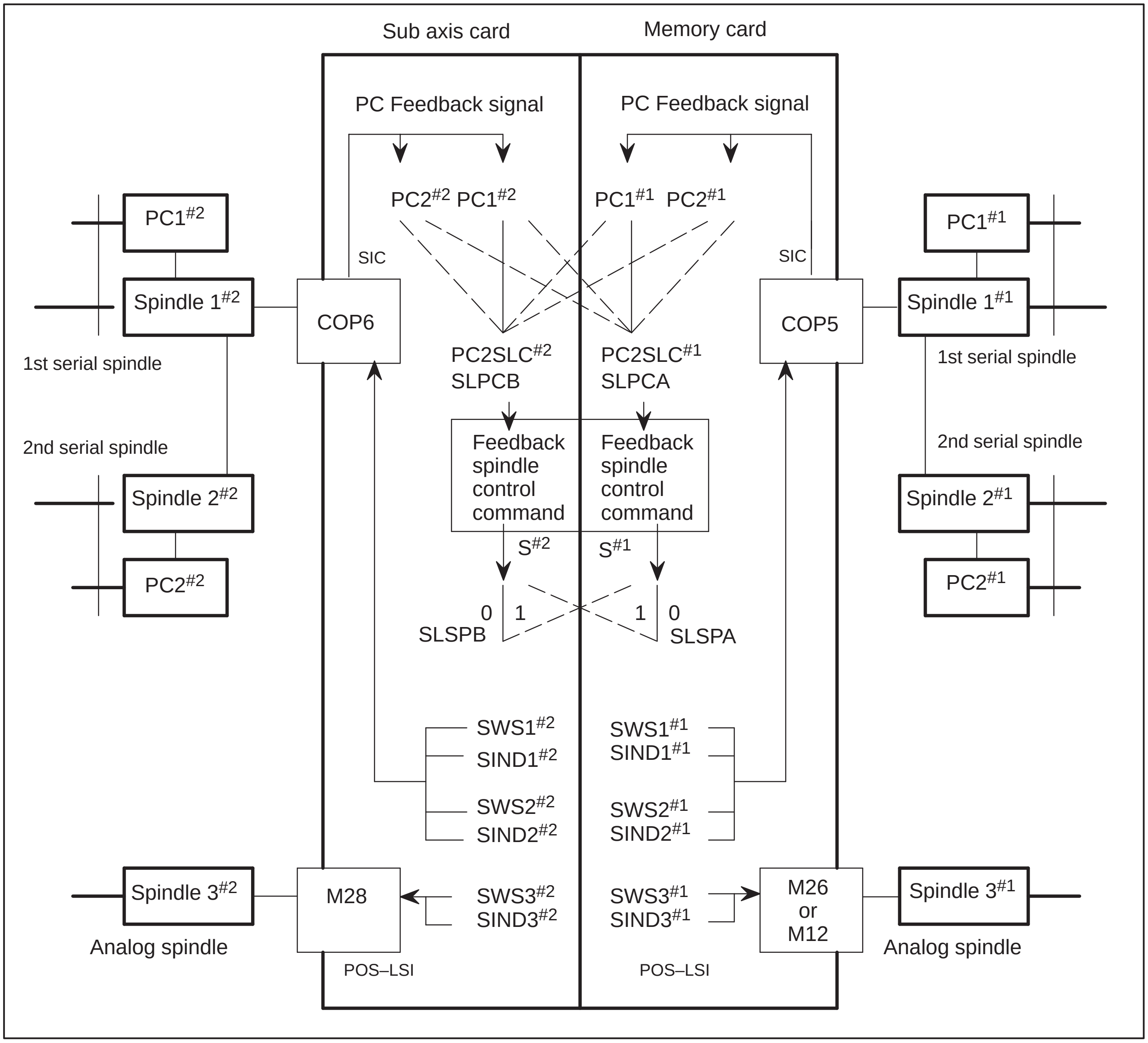


Fig. 9.4 (e) Spindle Configuration and Flow of Commands and Feedback Signals in the 0–TTC (with All Spindles under Two–Spindle Control)

The second and third spindles should be controlled using the PMC or be under multi–spindle control. (See Section 15.4 or 9.10.)

If multispindle control is applied to both tool posts, the position coder feedback signal for the second spindle of each tool post also becomes usable.

Options related to spindles

Optional functions for spindles are valid for both tool posts. However, you may want to use the optional functions for only one of the tool posts because of relationships with the interface and PMC ladder.

Parameters are available to disable the following functions for individual tool posts.

· Spindle serial output

· Spindle analog output

· Spindle positioning

Signal

Spindle command select signals SLSPA <G133#2>, SLSPB <G133#3>

[Classification] Input signal

[Function] Selects which tool post receives spindle command of which spindle.

SLSPA: Selects the spindle command for spindle connected to tool post 1.

SLSPB: Selects the spindle command for spindle connected to tool post 2.

The spindle command select signals are associated with the spindle commands as follows:

(1) In the 1-spindle control mode

Signal input SLSPA	Command to spindle
0	Spindle command of tool post 1
1	Spindle command of tool post 2

NOTE
SLSPB is ineffective.

(2) In the 2-spindle control mode

Signal input		Command to the spindle connected to tool post 1	Command to the spindle connected to tool post 2
SLPCA	SLPCB		
0	0	Spindle command of tool post 1	Spindle command of tool post 2
0	1	Spindle command of tool post 1	Spindle command of tool post 1
1	0	Spindle command of tool post 2	Spindle command of tool post 2
1	1	Spindle command of tool post 2	Spindle command of tool post 1

Spindle feedback select signals SLPCA <G1333#2> SLPCB <G1333#3>

[Classification] Input signal

[Function] Selects which spindle sends the feedback signal of the position coder to which tool post.

NOTE
This function is effective only in the 2-spindle control system in which both tool posts use serial spindle.

SLPCA: Selects the feedback signal for tool post 1.
SLPCB: Selects the feedback signal for tool post 2.

The spindle feedback select signals are associated with the feedback signals of the position coder as follows:

(1) In 1-spindle control mode
The feedback signal of the spindle connected to tool post 1 is always sent to both tool posts; the SLPCA and SLPCB signals have no meaning.

(2) In 2-spindle control mode

Signal input		Tool post 1	Tool post 2
SLPCA	SLPCB		
0	0	PC#1	PC#2
0	1	PC#1	PC#1
1	0	PC#2	PC#2
1	1	PC#2	PC#1

PC#1= Position coder feedback signal for the spindle connected to tool post 1.

PC#2= Position coder feedback signal for the spindle connected to tool post 2.

NOTE
The SLPCA and SLPCB signals are effective only in the 2-spindle control mode using two serial spindles. In the 2-spindle control mode using analog spindles, the feedback signal of spindle 1 is input to tool post 1, and the feedback signal of spindle 2 is input to tool post 2, regardless of the setting of the SLPCA and SLPCB signals.

Spindle command signal COSP <F180#5>

[Classification] Output signal

[Function] Indicates which tool post issued the latest spindle command.

[Output condition] The COSP signal turns to “1” when:

· Tool post 2 issues the spindle command.

The COSP signal turns to “0” when:

· Tool post 1 issues the spindle command, or neither of the tool posts issues the spindle command.

[Use] In 1-spindle control mode, if this signal is input to the SLSPA signal (spindle command select signal), the spindle speed specified by the latest spindle command can always be output to the spindle, regardless of whether it is from tool post 1 or 2.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G133					SLSPB	SLSPA		
G1333					SLPCB	SLPCA		
F180			COSP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0047					2SP				(T series)

[Data type] Bit

2SP Specifies whether one or two spindles are controlled (0–TTC).
0 : One spindle (two tool posts)
1 : Two spindle (two tool posts)

	#7	#6	#5	#4	#3	#2	#1	#0	
0381	IGNSIC								(0–TTC)

[Data type] Bit

IGNSIC For tool post 2, the S analog output function is:
0 : Used.
1 : Not used.

	#7	#6	#5	#4	#3	#2	#1	#0	
0384	EFCS	EFSS							(0–TTC)

[Data type] Bit

EFSS For tool post 2, the S serial output function is:
0 : Used.
1 : Not used.

EFCS For tool post 2, the Cs contour control function is:
0 : Used.
1 : Not used.

NOTE
Set the parameters IGNSIC, EFSS, and EFCS for tool post 2.

· Selecting position coder feedback signals for both tool posts in the 0–TTC under multi–spindle control.

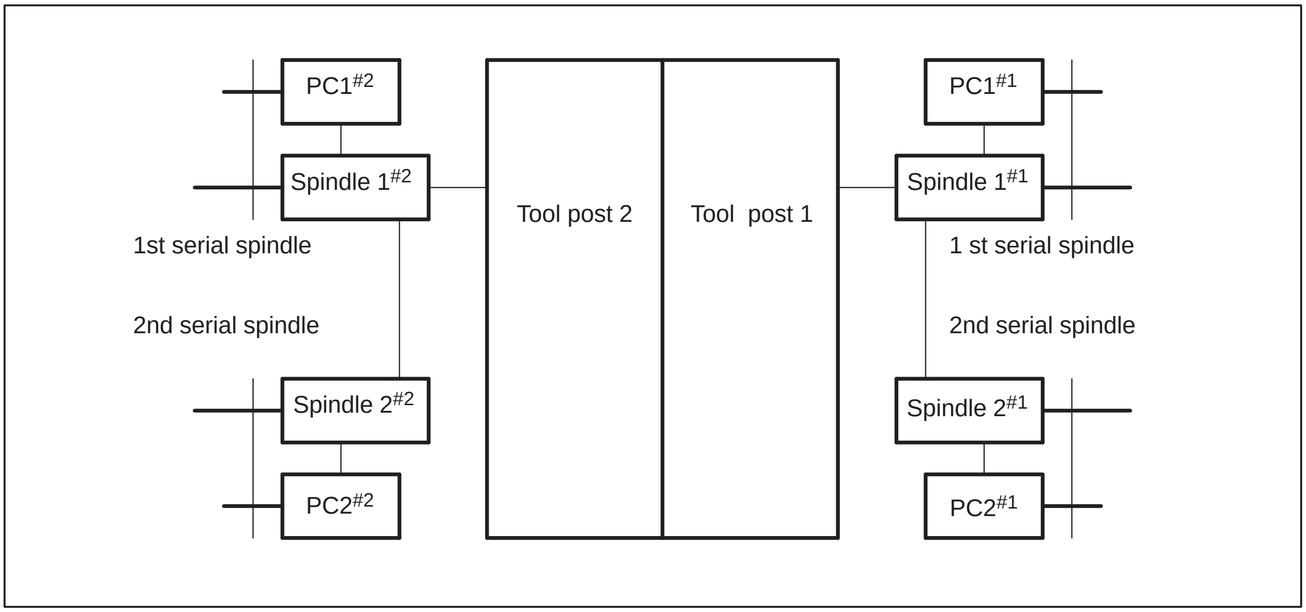


Table 9.4 lists the position coder feedback signals used for each tool post in the above configuration. These position coder feedback signals are selected according to the following:

· Spindle feedback select signals SLPCA <G1333#2> and SLPCB <G1333#3>

· Multi–spindle control
Second position coder select signals PC2SLC#1 <G146#7> and PC2SLC#2 <G1346#7>

Table 9.4 Selection of Position Coder Feedback Signal in the 0–TTC
(— means position coder selection is indifferent on the tool post side)

	Position coder selected	SLPCA	SLPCB	PC2SLC$^{\#1}$	PC2SLC$^{\#2}$
Tool post 1	PC1$^{\#1}$	"0"	—	"0"	—
	PC2$^{\#1}$	"0"	—	"1"	—
	PC1$^{\#2}$	"1"	—	—	"0"
	PC2$^{\#2}$	"1"	—	—	"1"
Tool post 2	PC1$^{\#1}$	—	"1"	"0"	—
	PC2$^{\#1}$	—	"1"	"1"	—
	PC1$^{\#2}$	—	"0"	—	"0"
	PC2$^{\#2}$	—	"0"	—	"1"

Note

NOTE

1 The spindle commands include S code commands, maximum speed command (G50S__), M03, M04, M05, and constant surface speed control commands (G96 and G97)

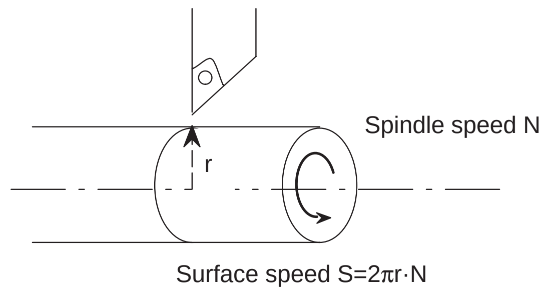
2 Signals to operate the spindle control unit are not affected by the spindle command select signals SLSPA<G133#2> or SLSPB<G133#3>. They may be processed in the PMC ladder, as required.
(Example: G229#5SFRA is always a forward rotation command for the first spindle control amplifier of tool post 1.)

3 The machine tool builder should prepare an external circuit to distribute and select position coder feedback signals for the analog spindle.

9.5 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.

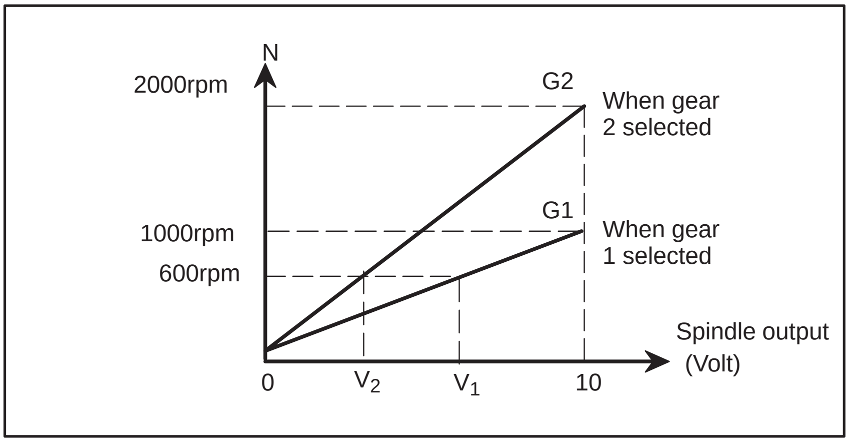
G97: Constant surface speed control not performed. S in the G97 mode is rev/min.

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 Nos.0540 – 0543.
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 rpm for the low speed gear (G1) and 2000 rpm for the high speed gear (G2), set these speeds to the parameter nos. 0540, 0541, 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 rpm ($S = 2\pi r N$). Therefore, V_1 (for G1) or V_2 (for G2) is calculated inside the CNC and output to the machine side.

V1: 6(V)

V2: 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 (rpm) at 10V output voltage (that is , spindle speed set by parameter Nos.0540 to 0543)

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 (rpm)

N: Spindle speed given by S command (rpm)

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 (rpm) at maximum spindle motor speed (that is , spindle speed set by parameter Nos.0540 to 0543)

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 (rpm)

N: Spindle speed given by S command (rpm)

Signal

Gear selection signal (Input) GR2, GR1 <G118#3, #2> (T series) <G123#3, #2>(M series)

Refer to 9.3 “Spindle Control”.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0041			SSCA1	SSCA0					(M series)

[Data type] Bit

SSCA, SSCA1 Set a reference axis for the calculation of constant surface speed control.

SSCA1	SSCA0	Axis
0	0	X
0	1	Y
1	0	Z
1	1	4

0540	Maximum spindle speed for gear 1
0541	Maximum spindle speed for gear 2
0542	Maximum spindle speed for gear 3
0543	Maximum spindle speed for gear 4

[Data type] Word
[Unit of data] rpm

0551	Minimum spindle speed in constant surface speed control mode (G96)

[Data type] Word
[Unit of data] rpm
[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.

0556	Maximum spindle speed

[Data type] Word
[Unit of data] rpm
[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 option is selected.
2 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.

Alarm and message

Number	Message	Description
190	ILLEGAL AXIS SELECT (M series)	In the constant surface speed control, the axis specification is wrong. (See bits 4 and 5 of parameter No. 0045.) 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. 0551, the actual spindle speed is clamped at the specified speed.

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 (T series) enables constant surface speed control for spindles other than the first spindle. (See Section 9.10.)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.10.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.10.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)

9.6 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES)

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 0564.)

Sr: The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r is set with parameter 0532.)

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)

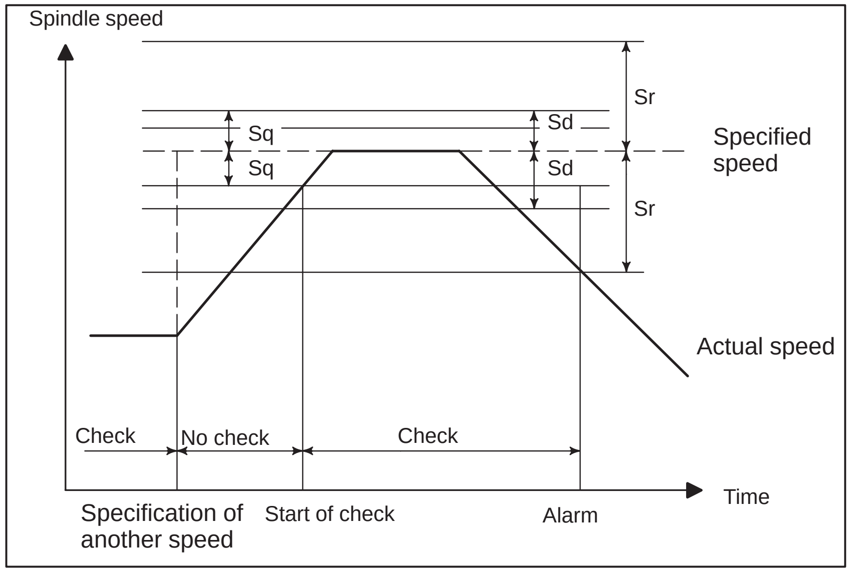
where $Sq = Sc \times q/100$

q: Percent tolerance of the target spindle speed, specified in parameter No. 0531. 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.

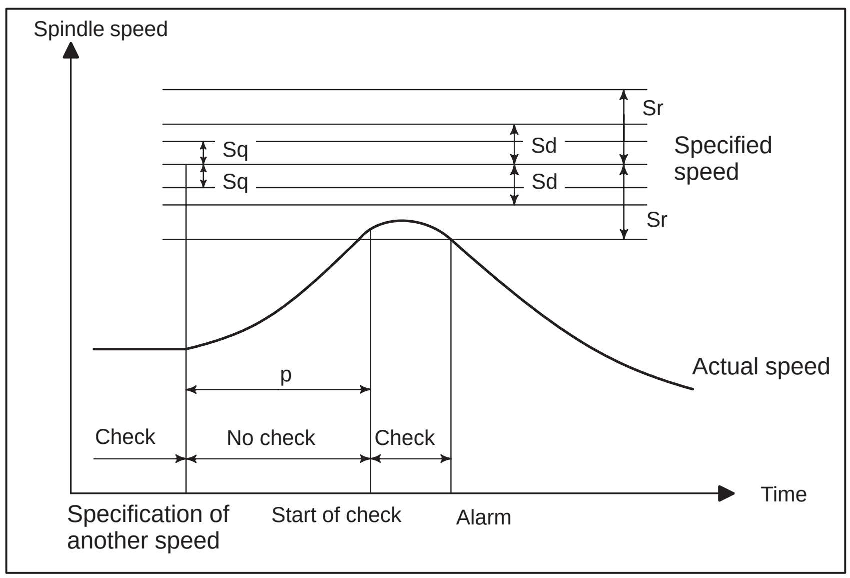
· When time p specified in parameter No. 0712 elapses after the specified speed Sc changes.

Parameter Nos. 0712, 0531, and 0532 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 <F154#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
0049						CHKSP		

[Data type] Bit

CHKSP 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
0397								SPLME

[Data type] Bit

SPLME When the spindle speed fluctuation detection function is used, the rates of allowance (q) and fluctuation (r) those are set in parameter No. 0531 and No. 0532, respectively are set in steps of:
0 : 1%
1 : 0.1%

0531	Ratio (q) of the fluctuation of spindle speed which is assumed to be the specified spindle speed

[Data type] Word

Unit of data	1%	0. 1%
Data range	1 to 100	1 to 1000

NOTE
Unit of data depends on parameter No. 0397#0 SPLME (T series only)

Set the ratio (q) of the spindle speed which is assumed to be the specified spindle speed in the spindle speed fluctuation detection function.

0532	Spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

Unit of data	1%	0. 1% (T series)
Data range	1 to 100	1 to 1000

NOTE
Unit of data depends on parameter No. 0397#0 SPLME.

Set the spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function.

0564	Spindle speed fluctuation value for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the allowable fluctuation speed for which no alarm is activated in the spindle speed fluctuation detection function.

0712	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] ms

[Valid data range] 0 to 999999

Set the time elapsed from when the specified spindle speed is changed to the start of spindle speed fluctuation detection in the spindle speed fluctuation detection function. That is, 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	OVERHEAT: 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 rpm.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.9.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)

9.7 ACTUAL SPINDLE SPEED OUTPUT (T SERIES)

General

The PMC can read actual spindle speed.

Signal

Actual spindle speed signal AR0 to AR15 <F158 to F159>

[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]

$$\text{Spindle speed} = \sum_{i=0}^{15} \{2^i \times V_i\} \text{ rpm}$$

where Vi = 0 when ARi is “0” and Vi = 1 when ARi is “1”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
F158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0	(T series)
F159	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08	(T series)

Note

NOTE

1 The AR0 - AR15 signals are always output. Their values change every 64 msec.

2 An absolute error of about 0.5 rpm exists as a measuring error.

9.8 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 with ease 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.

· Least command increment

$$\frac{360}{4096} \doteq 0.088 \text{ deg}$$

· Least 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 third axis.

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 $\pm$180deg for analog spindle and in the range from 0 to 360° for serial spindle.

To specify orientation, use the M code whose parameter no. 0587.
The orientation direction is specified by using parameters ZM3 no. 0003 #2 for analog or RETRN no. 6500#3 for serial spindles.

- **Orientation speed**

The spindle moves at rapid traverse set by parameter No. 0520 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. 0586. When a serial spindle is used, orientation speed depends on the spindle.

· Orientation enable speed
RPD>9$\times$(loop gain) KPPM
Loop gain: Parameter no. 0584 (unit: 1/sec)
Set rapid traverse speed at above value.

(Example)

When the loop gain parameter no. 0584 is set to 20 [1/sec], the orientation speed is:

RPD>9$\times$20$\times$1000$\times$(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 or automatic coordinates system settings (parameter APRS no. 0010#7)

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. 0589. Indexing angle β also requires prior parameter setting data no. 0590. Rotation can be done in any direction, using parameter SDRT data no. 0031#1.

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

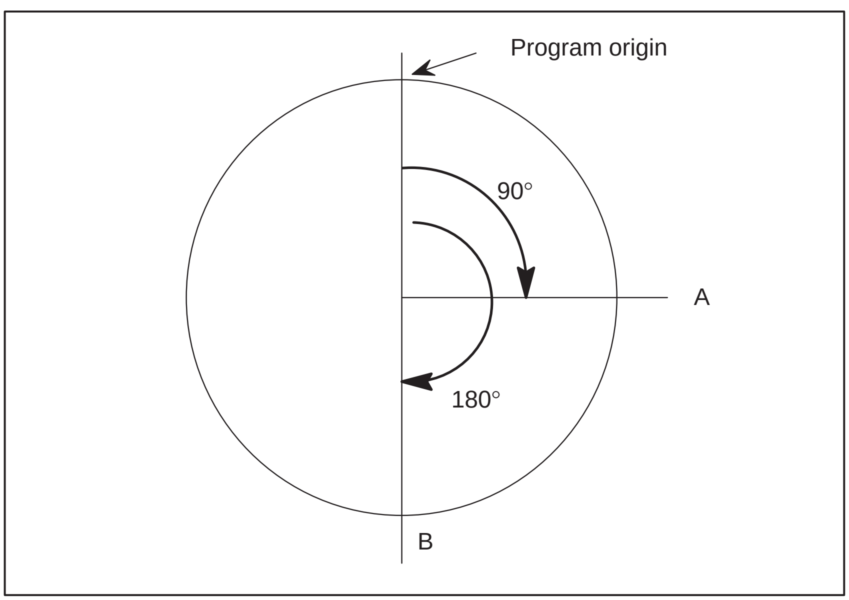
(iii) Decimal point input: A numerical value with decimal point can be entered. The decimal point location is in “degrees”, for instance: C35.0=C35 degrees

When address C is used, the ADRC bit (bit 4 of parameter 0029) must be set to 1.

- **Absolute and incremental commands**

Semi–fixed angle positioning (specified with M code) must always be specified using incremental values.

For arbitrary angle positioning, use address C to specify the position of the end point relative to the program origin (absolute) or use address H to specify the distance between the start and end points (incremental).



Command method		G code system A		G code system B, C	
		Address used	Command of A–B on the above illustration	Address used and G–code	Command of A–B on the above illustration
Absolute command	Direct the end 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 positions.	H	H90.0 ;	G91,C	G91C90.0 ;

- **Spindle positioning feedrate**

Spindle positioning is done at the rapid traverse rate specified by parameter No. 0520, to which linear acceleration deceleration are applied. Overrides of 100%, 50%, 25% and F0 (parameter No. 0585) are also applied.

- **Spindle positioning reset**

A specific M code (parameter no. 0588) must be set when the mode is changed from spindle positioning to normal spindle rotation.

Signal

Spindle stop complete signal SPSTP <G123#6>

[Classification] Input signal

[Function] When this signal is 1, the CNC orients and positions the spindle.

Spindle unclamp signal SUCLP <F164#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 <G123#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 <F164#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 <G123#5>

[Classification] Input signal

[Function] This signal indicates that clamping the spindle is complete in response to the spindle clamp signal SCLP.

Other related signals

Gear selection signal GR1, GR2, <G118#2, #3>

Refer to 9.3 “Spindle Control.”

CTH1A, CTH2A <G229#3, #2>

Refer to the manual of 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 ZP3 <F148#2>

[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.

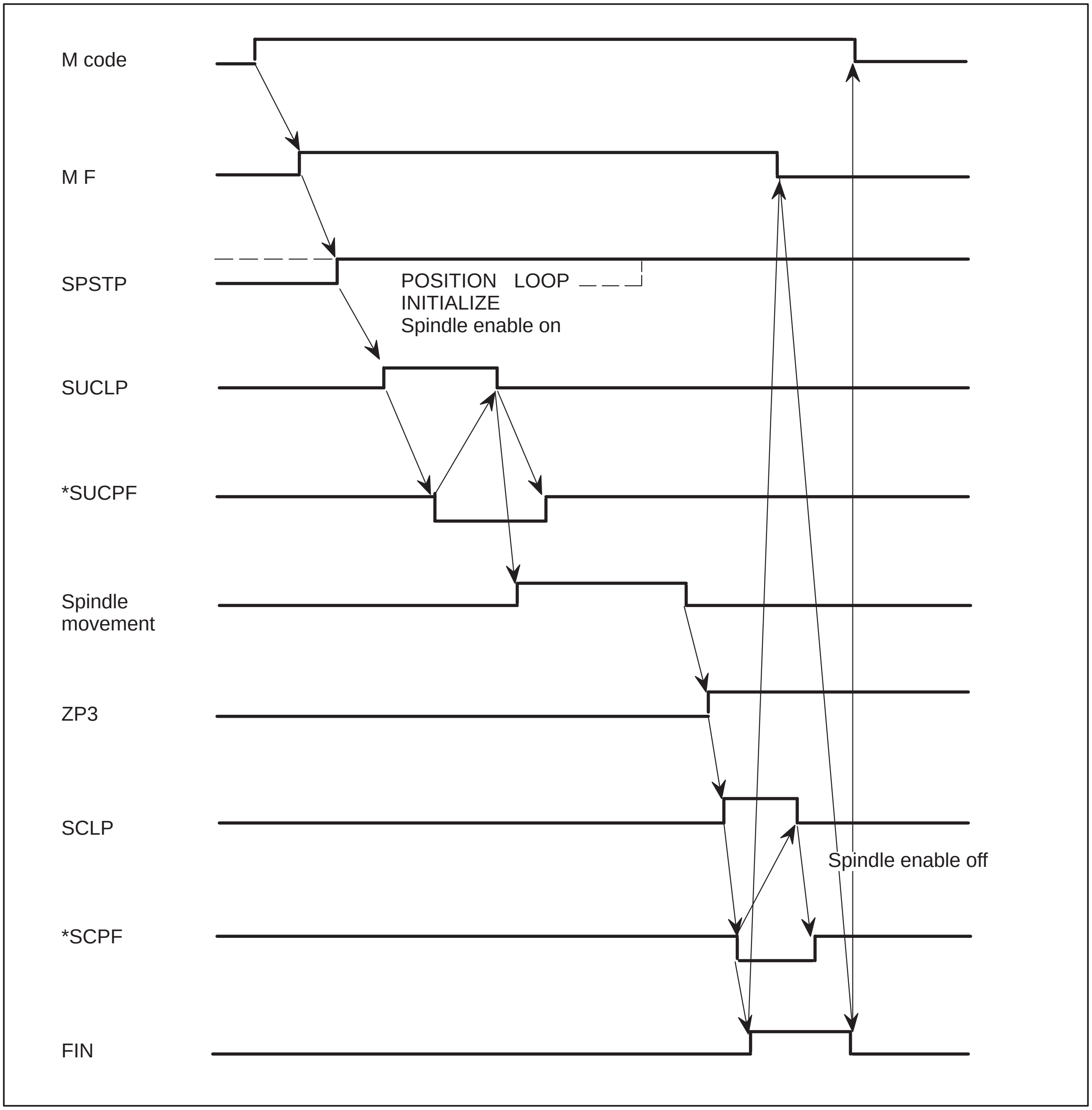
Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G123		SPSTP	*SCPF	*SUCPF				
G229					CTH1A	CTH2A		

	#7	#6	#5	#4	#3	#2	#1	#0
F148						ZP3		
F164							SUCLP	SCLP

Sequence (Time chart)

❑ Spindle Orientation



⇒ POSITION LOOP INITIALIZE is performed within the CNC.

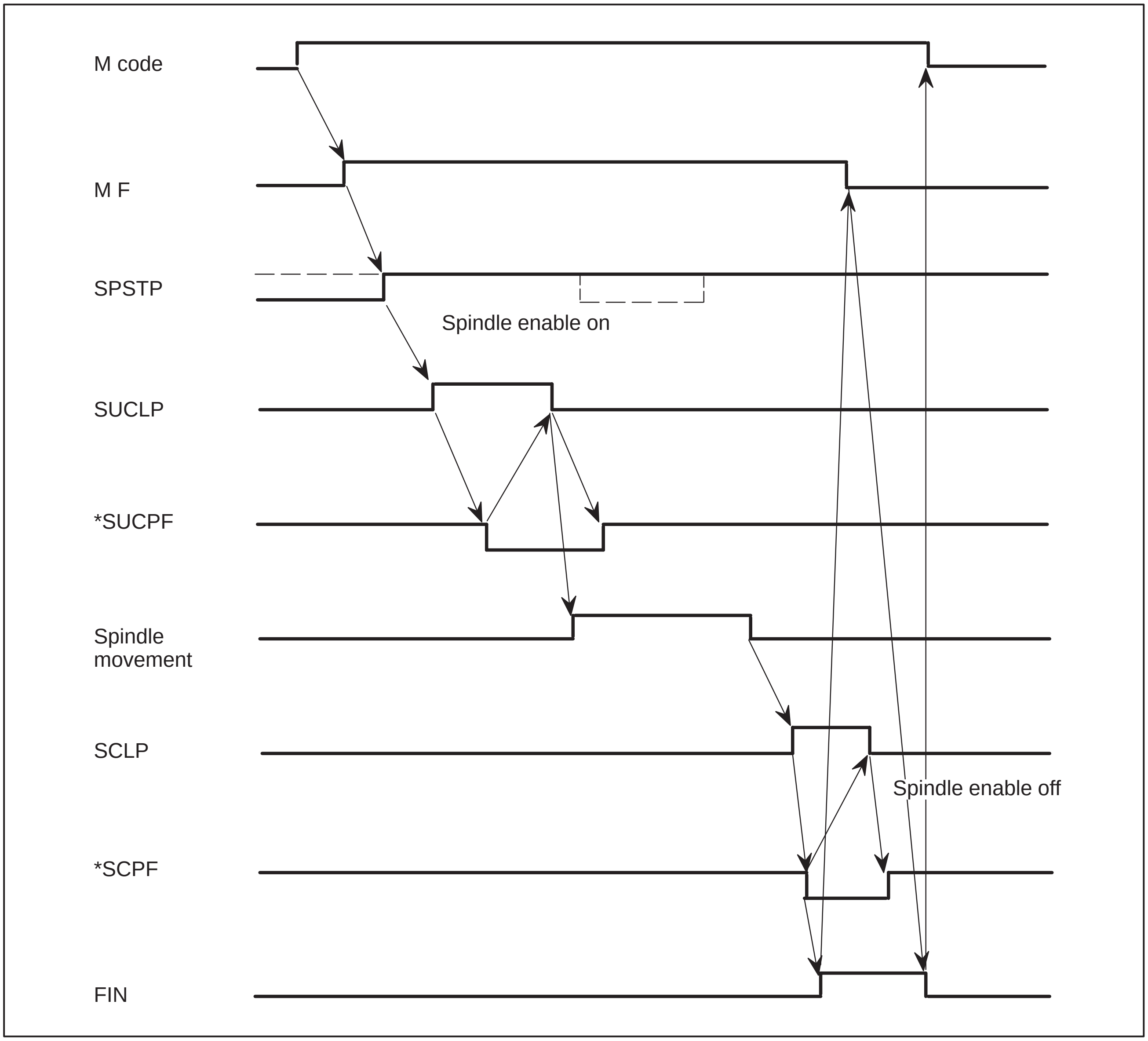
⇒ Spindle ENABLE ON/OFF specifies that the PMC ladder direct the spindle control unit to run or stop the spindle motor.

(Example) For serial spindles, the ladder should contain the following command or something like that:

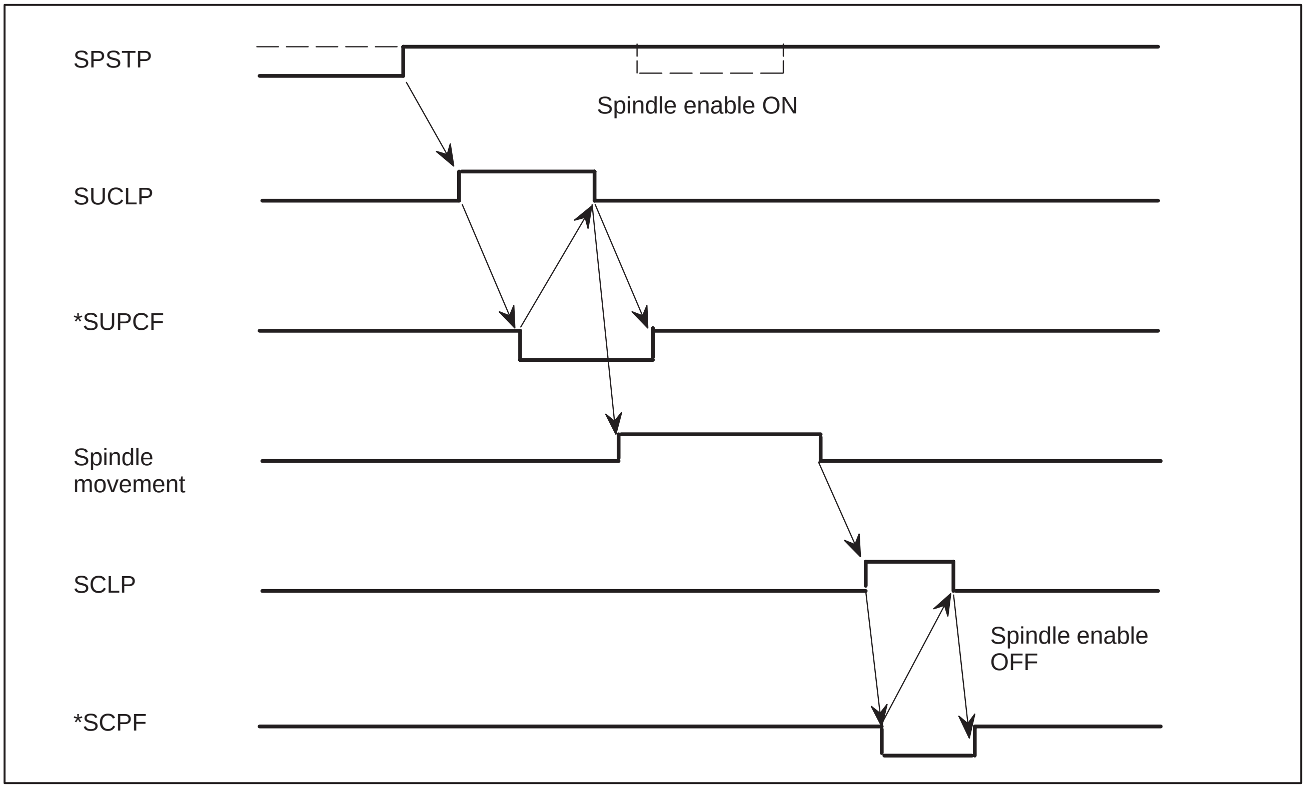
ENABLE ON, and SFRA<G229#5> ⇒ 1
ENABLE OFF, and SFRA<G229#5> ⇒ 0

For details, refer to the manual for the spindle control unit you actually use.

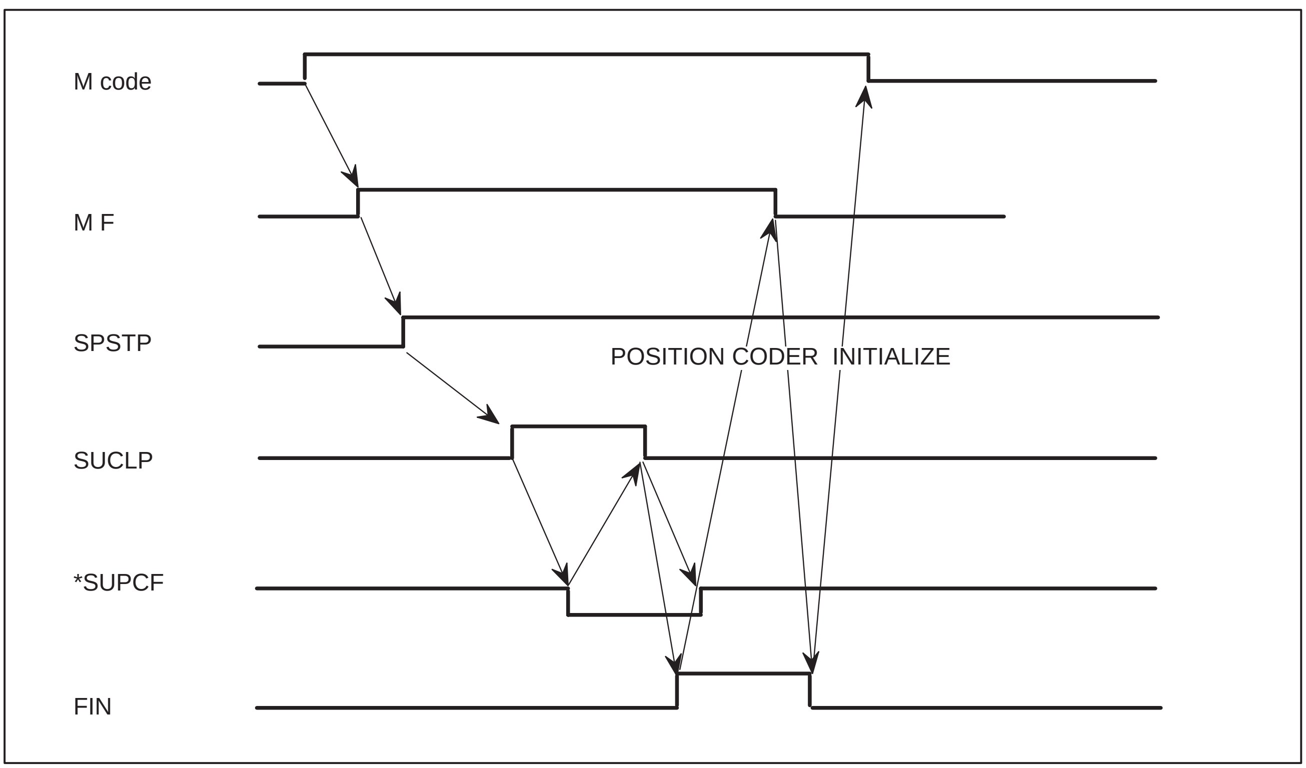
❑ Spindle Positioning by M code



❑ 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
0003					ZM3			

[Data type] Bit axis

ZM3 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
0006		DMR3			GRD3			

[Data type] Bit

DMR3 Setting of delection multiply
Set this parameter to “111” (=4) for spindle positioning.

GRD3 Set the size of the reference counter
Set this parameter to “1001” (=10000) for spindle positioning.

	#7	#6	#5	#4	#3	#2	#1	#0
0010	APRS							

[Data type] Bit

APRS Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

	#7	#6	#5	#4	#3	#2	#1	#0
0029				ADRC				

ADRC 0: For chamfering, corner R, or direct drawing dimension programming, address C or R is used as in the standard specifications. (Address C cannot be used for the third axis.)
1: For chamfering or corner R, address 1 or K is used instead of address C. For direct drawing dimension programming, address C or R, with a comma, is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0030								ADW30

ADW30 Names the third axis:
0: B.
1: C.

> **NOTE**
> When a serial spindle is used, this parameter is invalid for the spindle positioning axis.

	#7	#6	#5	#4	#3	#2	#1	#0
0031							SDRT	SNRL

[Data type] Bit

SNRL Resetting the system in the spindle positioning mode
0 : Does not releases the mode.
1 : Releases the mode

SDRT The positioning direction for the spindle using a M code is
0 : The positive direction
1 : The negative direction

> **NOTE**
> 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.
> Operations (1) to (3) are specified using separate M codes.
> (1) Specified using M codes for performing spindle orientation.
> (See parameter No. 0587)
> (2) Specified using M codes for specifying a spindle positioning angle. (See parameter No. 0588)
> (3) Specified using M codes for clearing spindle positioning operation. (See parameter No. 0589.)

0102	Command multiply for third axis (CMR)

[Data type] Byte

- 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]

<u>Set this parameter to 2 for spindle positioning.</u>

0502	In–position width for third axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for third axis.

0506	Positioning deviation limit for third axis in movement

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for third axis.

0510	Grid shift for third axis

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] –32767 to + 32767

Set a grid shift for third axis.

> **NOTE**
> Set this parameter when the analog spindle is used. When the serial spindle is used, set the value to No. 6573.

0520	Rapid traverse rate for third axis

[Data type] Word

Set the rapid traverse rate when the rapid traverse override is 100% for third axis.

For spindle positioning.

[Unit of data] 10 deg/min

[Valid data range] 30 to 12000

0524	Time constant of rapid traverse linear acceleration/deceleration for third axis

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set time constant of rapid traverse linear acceleration/deceleration for third axis.

0537	Backlash compensating value for third axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to +255

Set the backlash compensation value for third axis.

0580	Servo loop gain multiplier of the spindle for gear 1
0581	Servo loop gain multiplier of the spindle for gear 2
0582	Servo loop gain multiplier of the spindle for gear 3
0583	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 rpm 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 rpm, E is regarded as 2.2 V.
> 2 The above parameters No. 0580 to No. 0584 are for analog spindles

0584	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.

0585	F0 rate of rapid traverse override for spindle

[Data type] Word

Set the F0 rate of the rapid traverse override for spindle.

<u>For spindle positioning.</u>

[Unit of data] deg/min

[Valid data range] 600 to 15000

0586	FL rate at the spindle orientation

[Data type] Word

Set feedrate (FL rate) after deceleration when the reference position return is performed for spindle.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

> **NOTE**
> When serial spindle is used, this parameter becomes invalid.

0587	M code specifying the 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.

0588	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.

0589	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 92

Two methods are availablel for specifying spindle positioning. One method uses address C for arbitrary–angle positioning. The other use an M code for half–fixed angle positioning. This parameter sets an M code for the latter method.

Six M code from M α to M(α+5) are used for half–fixed angle positioning, when α is the value of this parameter.

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°

NOTE
θ represents the basic angular diplacement set in pamrameter No. 0590.

0590	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 half–fixed angle positioning using M codes.

0595	Positioning deviation limit for third axis in the stopped state

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

0710	Coordinate value of the reference position used when automatic coordinate system setting is performed for third axis

[Data type] Two–word

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

For spindle positioning.

[Unit of data] 0.001 deg

[Valid data range] –99999999 to 99999999

	#7	#6	#5	#4	#3	#2	#1	#0
6500					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.

6544	Velocity loop proportion gain in servo mode (High gear)
6545	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.

6552	Velocity loop integral gain in servo mode (High gear)
6553	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.

6556	Gear ratio (HIGH)
6557	Gear ration (MEDIUM HIGH)
6558	Gear ratio (MEDIUM LOW)
6559	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, CTH2A.

6565	Position gain in servo mode (HIGH)
6566	Position gain in servo mode (MEDIUM HIGH)
6567	Position gain in servo mode (MEDIUM LOW)
6568	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. 0584. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH2A.

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. 0029#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.
195	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.
409	FIRST SPINDLE ALARM DETECTION (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.

Warning

WARNING

1 Feed hold is invalid during spindle positioning.
2 Dry run, machine lock, and auxiliary function lock are not available during spindle positioning.
3 Specify parameter no. 0589 even if semi-fixed angle positioning is not used; otherwise M codes (M00 to M05) do not work.

Caution

CAUTION

1 Spindle positioning stops when emergency stop is applied; restart with orientation operation.
2 The spindle positioning function and the serial spindle Cs contour control function cannot be used together. If both functions are specified, positioning has priority.

Note

NOTE

1 Direct spindle positioning with an independent block. X- and Y-axis positioning cannot be commanded to the sample 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 0 V. (parameter no. 0539). 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

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.9.5	SPINDLE POSITIONING FUNCTION

9.9 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**

±9999.999 [deg]

- **Feedrate**

Rapid traverse rate: 30 to 2400 [deg/min]
(parameter no.0520(T)/0521(M))

Cutting feedrate (feed per minute):

1 to 15000 [deg/min] (for machines that use millimeters)
0.01 to 600 [deg/min] (for machines that use inches)

> **NOTE**
> The units in which the following parameters are specified, in inch output, can be multiplied by ten for the rotation axis alone, provided the ROT10 bit (bit 7 of parameter No. 0032) is specified accordingly: jog feedrate, cutting feed clamp value, F0 feedrate for rapid traverse override, and the FL feedrate at reference position return. (T series)

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. For Cs contour control, the T series uses the third axis while the M series uses the fourth axis. The axis of Cs contour control must be set as a rotation axis (T series: by specifying the LIN3 bit (bit 2 of parameter 0032) to 0; M series: by specifying the ADLN bit (bit 2 of parameter 0011) to 0).

Only one set of this setting can be used for each control path. 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. 0030#0, 0069#5, #6 for T series/No.0008#2 to #4 for M series). This address is arbitrary.

When the second auxiliary function option is provided, address B cannot be used for the name of the contour axis. For the T series machines, when either address C is used for the name of the contour axis, clear ADRC (parameter no. 0029#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.

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 (G123#7) for M series to “1” or COFF(G123#0) for T series to “0”. 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 (G123#7) for M series to “0” or COFF (G123#0) for T series to “1” 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. The spindle starts the movement to the reference position; when it reaches that position, the reference position return completion signal (ZPn (F148)) 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 G00 or G28 is specified. Under certain conditions, the G00 command returns the spindle to the reference position, depending upon the setting of parameter CZRN no. 0065#1:

(i) G00 command

When parameter CZRN no. 0065#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(F148)) 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.

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.

(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 “0” 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

DGN No.

0802	Position deviation amount of 1st spindle	(T series)
0803	Position deviation amount of 1st spindle	(M series)

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.

Signal

Spindle contour control change signal CON <G123#7> (M series) COFF <G123#0>(T series)

[Classification] Input signal

[Function] This signal specifies that with the Cs contour control function, the first serial spindle be switched between the spindle speed control and Cs contour control modes.

(M series) 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.

(T series) When this signal turns to “0”, 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 “1” changes the spindle mode from Cs contour control back to speed control.

Spindle contour control change completion signal FSCSL <F178#1>

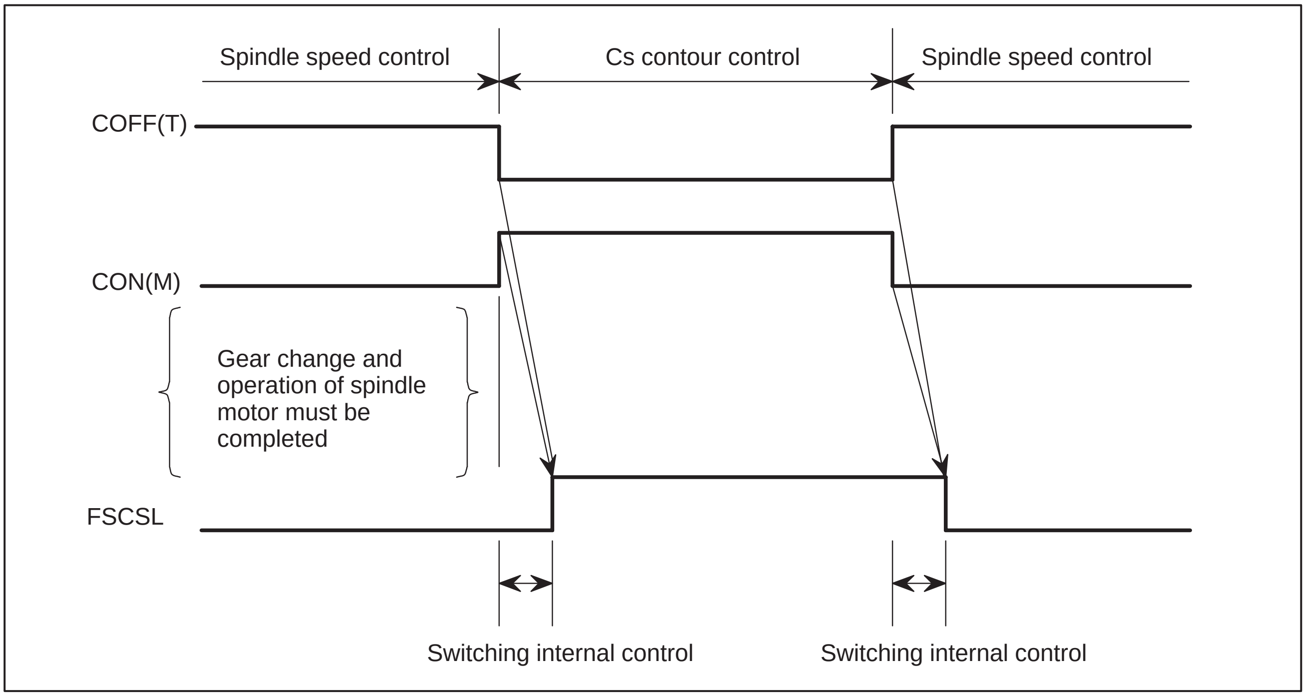
[Classification] Output signal

[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 (M) or COFF(T) 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 <G229#5, #4> or other required signals must be appropriately processed on the machine side).

Other signals

Gear select signal (Input)
GR1, GR2, <118#2, #3> (T)
<G123#2, #3> (M)
Gear select signal (Output)
GR3O, GR2O, GR1O
<F152#2, #1, #0>
(M series)

Refer to 9.3 “Spindle speed Control”.

Clutch/Gear signal (Serial spindle)
CTH1A, CTH2A
<G229#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

Analog spindle							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
When the M series does not include the constant surface speed control option, GR1 and GR2 do not need to be input. Input CTH1A and CTH2A when gears are changed using GR10, GR2O and GR3O.

Cs contour control axis reference position return completion signal
ZP3 <F148#2> (T series)
ZP4 <F148#3> (M series)

[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
- Manual handle feed axis select signal

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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G118					GR2	GR1			(T series)
G123								COFF	(T series)
G123	CON				GR2	GR1			(M series)
G229	MRDYA		SFRA	SRVA	CTH1A	CTH2A			

	#7	#6	#5	#4	#3	#2	#1	#0
F148					ZP4	ZP3		
F152						GR3O	GR2O	GR1O
F178							FSCSL	

Parameter

The following describes major parameters.

In addition, parameters such as axis feedrate, acceleration/deceleration, and display can be used. Also, digital servo parameters (Nos. 8300 or later (T)/Nos. 8400 or later (M)) for the Cs contour axis are not required to be set.

	#7	#6	#5	#4	#3	#2	#1	#0	
0008				ADW2	ADW1	ADW0			(M series)

[Data type] Bit

ADW0, ADW1, ADW2 Sets the name of the fourth axis as indicated below:

#4 ADW2	#3 ADW1	#2 ADW0	Name
0	0	0	A
0	0	1	B
0	1	0	C
0	1	1	U
1	0	0	V
1	0	1	W
1	1	0	A
1	1	1	A

NOTE
1 Multiple axes must not have identical axis names.
2 When an optional second miscellaneous function is provided, address B cannot be used. To use address C with the T series, set the ADRC bit (bit 4 of parameter 0029) to 0.

Any axis name can be assigned to an axis for Cs contour control.

	#7	#6	#5	#4	#3	#2	#1	#0
0010	APRS							

[Data type] Bit

APRS Automatic setting of a coordinate system when the manual reference position return is performed
0 : Not set automatically
1 : Set automatically

	#7	#6	#5	#4	#3	#2	#1	#0	
0011						ADLN			(M series)

[Data type] Bit

ADLN Type of fourth axis
0 : Rotation axis
1 : Linear axis

NOTE
Inch/metric conversion cannot be made to the rotation axis. The machine coordinate values are rounded in 0° to 360°. 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 the rotation axis for Cs contour control axis

	#7	#6	#5	#4	#3	#2	#1	#0	
0030								ADW30	(T series)

[Data type] Bit

ADW30 Names the third axis:
0 : B.
1 : C.

	#7	#6	#5	#4	#3	#2	#1	#0
0031	CNRST							

[Data type] Bit

CNRST When reference position return is completed, relative coordinate value of Cs contour control axis is
0 : Not cleared
1 : Cleared

	#7	#6	#5	#4	#3	#2	#1	#0
0032	ROT10					LIN3		

LIN3 Specifies whether the third axis is a linear ro rotation axis, as follows.
0 : Rotation axis
1 : Linear axis

Set the rotation axis for the Cs contour control axis.

ROT10 Specifies the measurement unit of parameters for the upper limit to the cutting feedrate rapid traverse F0 rate and the FL rate for a reference position return during inch output, as follows:
0 : 0.1 degrees/min
1 : 1 degrees/min

	#7	#6	#5	#4	#3	#2	#1	#0
0065							CZRN	

[Data type] Bit

CZRN 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

0102	Command multiply for third axis (CMR)	(T series)
0103	Command multiply for fourth axis (CMR)	(M series)

[Data type] Byte

- 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]

0271	Number of the servo axis for third axis	(T series)
0272	Number of the servo axis for fourth axis	(M series)

[Data type] Byte

Set the servo axis for third or fourth control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

Set 255 as the number of servo axis to the Cs contour control axis.

0279	Setting of third axis in the basic coordinate system	(T series)
0280	Setting of fourth axis in the basic coordinate system	(M series)

[Data type] Byte

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.

0502	In–position width for third axis	(T series)
0503	In–position width for fourth axis	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for third or fourth axis.

0506	Positioning deviation limit for third axis in movement	(T series)
0507	Positioning deviation limit for fourth axis in movement	(M series)

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 999999999

Set the positioning deviation limit in movement for third or fourth axis.

0520	Rapid traverse rate for third axis	(T series)
0521	Rapid traverse rate for fourth axis	(M series)

[Data type] Word

[Unit of data] 1 deg/min

[Valid data range] 30 to 24000 (IS–A, IS–B)
30 to 12000 (IS–C)

Set the rapid traverse rate when the rapid traverse override is 100% for third or fourth axis.

0524	Time constant used in linear acceleration/deceleration in rapid traverse for third axis	(T series)
0525	Time constant used in linear acceleration/deceleration in rapid traverse for fourth axis	(M series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used in linear acceleration/deceleration in rapid traverse for fourth or third axis.

0537	Backlash compensation value used for rapid traverse for third axis	(T series)
0538	Backlash compensation value used for rapid traverse for fourth axis	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensation value for third or fourth axis.

0595	Positioning deviation limit for third axis in the stopped state	(T series)
0596	Positioning deviation limit for fourth axis in the stopped state	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the positioning deviation limit in the stopped state for third or fourth axis.

0710	Coordinate value of the reference position used when automatic coordinate system setting is performed	(T series)
0711	Coordinate value of the reference position used when automatic coordinate system setting is performed	(M series)

[Data type] Two–word

Set the coordinate value of the reference position on each axis to be used for setting a coordinate system automatically.

6565	Gear ratio (HIGH)
6566	Gear ratio (MEDIUM HIGH)
6567	Gear ratio (MEDIUM LOW)
6568	Gear ratio (LOW)

[Data type] Word

[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.

6569	Position gain at Cs contour control (High gear)
6570	Position gain at Cs contour control (Medium High gear)
6571	Position gain at Cs contour control (Medium Low gear)
6572	Position gain at Cs contour control (Low gear)

[Data type] Word

[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.

6635	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.

> **NOTE**
> This parameter can substitute for parameter 0510 (T series) or 0511 (M series). The parameter determines the position of reference position return under Cs contour control by the serial spindle.

6780	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (1st gear)
6781	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (2nd gear)
6782	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (3rd gear)
6783	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (4th gear)

[Data type] Word

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

[Valid data range] 0 to 9999

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear.

6784	Loop gain of the X axis that interpolates with Cs contour control axis during interpolation (1st gear)
6785	Loop gain of the X axis that interpolates with Cs contour control axis during interpolation (2nd gear)
6786	Loop gain of the X axis that interpolates with Cs contour control axis during interpolation (3rd gear)
6787	Loop gain of the X axis that interpolates with Cs contour control axis during interpolation (4th gear)

[Data type] Word

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

[Valid data range] 0 to 9999

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear.

6788	Loop gain of the Z axis (T)/Y axis (M) that interpolates with Cs contour control axis during interpolation (1st gear)
6789	Loop gain of the Z axis (T)/Y axis (M) that interpolates with Cs contour control axis during interpolation (2nd gear)
6790	Loop gain of the Z axis (T)/Y axis (M) that interpolates with Cs contour control axis during interpolation (3rd gear)
6791	Loop gain of the Z axis (T)/Y axis (M) that interpolates with Cs contour control axis during interpolation (4th gear)

[Data type] Word

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

[Valid data range] 0 to 9999

Set the servo loop gain of the Z axis (T)/Y axis (M) that interpolates with Cs contour control axis on each spindle gear.

6792	Loop gain of the Z axis (M) that interpolates with Cs contour control axis during interpolation (1st gear)
6793	Loop gain of the Z axis (M) that interpolates with Cs contour control axis during interpolation (2nd gear)
6794	Loop gain of the Z axis (M) that interpolates with Cs contour control axis during interpolation (3rd gear)
6795	Loop gain of the Z axis (M) that interpolates with Cs contour control axis during interpolation (4th gear)

[Data type] Word

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

[Valid data range] 0 to 9999

Set the servo loop gain of the Z axis (M) that interpolates with Cs contour control axis on each spindle gear.
For T series, set the same values that set to the parameter Nos.6569 to 6572.

6796	Loop gain of the fourth axis (T) that interpolates with Cs contour control axis during interpolation (1st gear)
6797	Loop gain of the fourth axis (T) that interpolates with Cs contour control axis during interpolation (2nd gear)
6798	Loop gain of the fourth axis (T) that interpolates with Cs contour control axis during interpolation (3rd gear)
6799	Loop gain of the fourth axis (T) that interpolates with Cs contour control axis during interpolation (4th gear)

[Data type] Word

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

[Valid data range] 0 to 9999

Set the servo loop gain of the fourth axis (T) that interpolates with Cs contour control axis on each spindle gear.
For T series, set the same values that set to the parameter Nos. 6569 to 6572.

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.
195	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.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs contour control axis when the signal CON (G123#7) for M series was off or the signal COFF (G123#0) for T series was on. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
409	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle control 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.

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 CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.6	Cs Contour Control Function
FANUC AC SPINDLE MOTOR series (Serial interface) DESCRIPTIONS (B–65042E)	Appendix 3.2	Cs Contour Control Function Start–up Procedure

9.10 MULTI-SPINDLE CONTROL (T SERIES)

General

In addition to the conventional (first) spindle, two other (second and third) spindles can be controlled. These additional spindles allow two-stage gear changes. An S code is used for a command to any of these spindles; which spindle is selected is determined by a signal from the PMC. The second and third spindle can change gears in 2 stages.

When the second spindle is used, one position coder interface channel is added. Which position coder is selected is determined by a PMC signal. (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.)

Selection between 1st position coder and 2nd position coder is made by a signal from PMC.

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 three 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 <G145#0-#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 <G145#3-#5>) to stop its rotation; an unused spindle can still be stopped.

There is a spindle enable signal to control each spindle; ENB <F149#4> controls the first spindle, while ENB2 and ENB3 <F164#2, #3> control the second and third spindles, respectively.

The PMC signal PC2SLC <G146#7> is used to select between the first and second position coders.

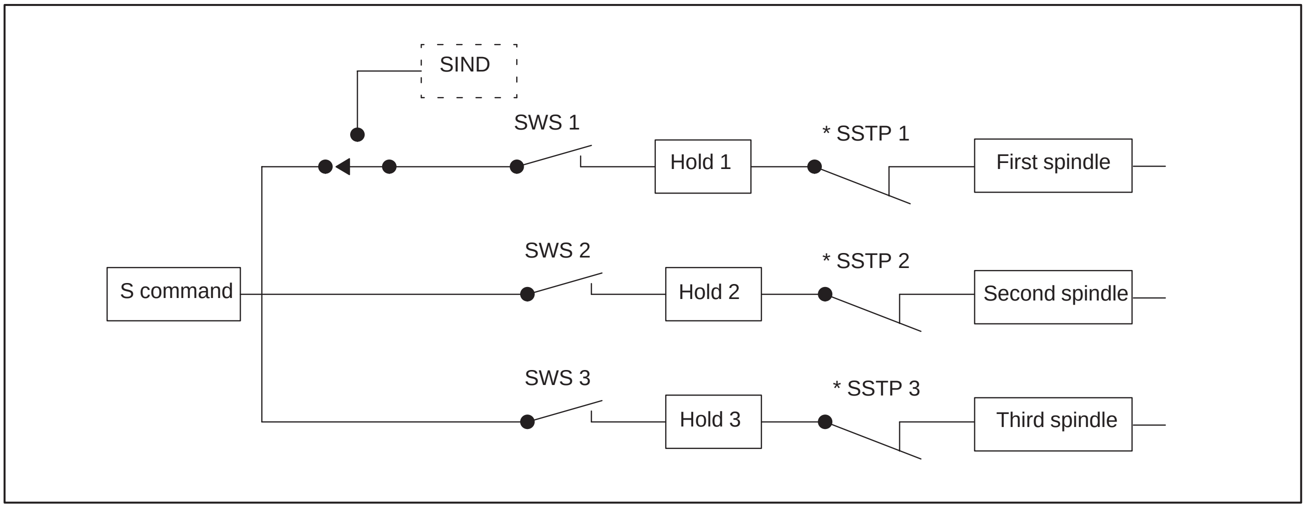
Multi-spindle control (TYPE-A)

When parameter MSPDB (No. 0070#2)=0, TYPE-A is used.

When the first spindle is selected with the SWS1 signal, the SIND signal <G125#7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I <G125#3 to G124#0> are used to set that spindle's analog voltage. These signals do not affect the second and third spindles.

The PMC-based polarity (rotation direction) control signals SGN and SSIN <G125#5,#6> will function for any spindle selected by SWS1 to SWS3.

The concept of Type A multi-spindle control is outlined below.



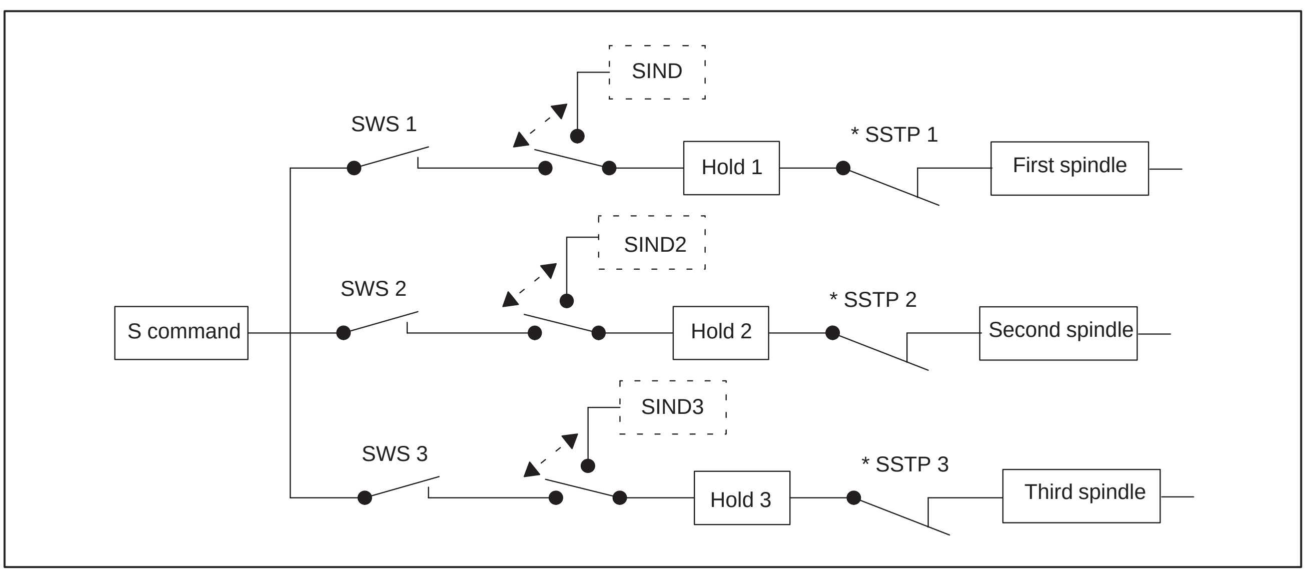
Multi-spindle control (TYPE-B)

Select Type B control by setting parameter MSPDB no. 0070#3 to "1".

Each spindle has its own SIND, SSIN and SGN signals.
Each signal 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 (rotation direction) control signals SSIN, SGN will function.

The concept of Type B multi-spindle control outlined below.



Spindles to be controlled

The multi–spindle control function can control the following combinations of spindles:

1) Three analog spindles
2) One serial spindle (used as the first spindle) and two analog spindles
3) Two serial spindles (used as the first and second spindles) and one analog spindle (used as the third spindle)

In any of the above configurations, the second or third spindle may be left unconnected.

Connection of the spindles

See the table below:

Spindle configuration for multi–spindle control

	Three analog spindles
Function requirements	Multi–spindle control, spindle speed command with four or five digits (The serial spindle interface should not be used.)
Hardware requirements	Analog interface board (For the NC unit, control unit B must be used to connect the analog interface board.)
Spindle connection	First spindle → M26 or M12 connector (on the memory board) Second spindle → M121 connector V01 (on the analog interface board) Third spindle → M121 connector V02 or M122 (on the analog interface board)
Position coder connection	First position coder → M27 connector (on the memory board) Second position coder → M124 connector (on the analog interface board)

	One serial spindle (used as the first spindle) and two analog spindles (used as the second and third spindles)
Function requirements	Multi–spindle control, SRLMSP bit (bit 2 of serial spindle interface parameter 0071) set to 0
Hardware requirements	Analog interface board (For the NC unit, control unit B must be used to connect the analog interface board.)

	One serial spindle (used as the first spindle) and two analog spindles (used as the second and third spindles)
Spindle connection	First spindle → COP5 optical connector (on the memory board) Second spindle → M121 connector V01 (on the analog interface board) Third spindle → M121 connector V02 or M122 (on the analog interface board)
Position coder connection	First position coder When the ISRLPC bit (bit 0 of parameter 0071) is set to: 0 → COP5 optical connector (on the memory board) 1 → M27 connector (on the memory board) Second position coder → M124 connector (on the analog interface board)

	Two serial spindles (used as the first and second spindles) and one analog spindle (used as the third spindle)
Function requirements	Multi–spindle control, SRLMSP bit (bit 2 of serial spindle interface parameter 0071) set to 1
Hardware requirements	Interface for the spindle speed command with four or five digits (when the third spindle is used) (No analog interface board is necessary. Therefore, either control unit A or B can be used for the NC unit.)
Spindle connection	First spindle → COP5 optical connector (on the memory board) Second spindle → CN11B optical connector (on the first spindle control unit) Third spindle → M26 or M12 connector (on the memory board)
Position coder connection	First position coder When the ISRLPC bit (bit 0 of parameter 0071) is set to: 0 → COP5 optical connector (The data output by the position coder connected to the first spindle control unit is captured using the optical cable.) 1 → M27 connector (Both the COP5 optical connector and the M27 connector are provided on the memory board.) Second position coder When the SSMPC bit (bit 3 of parameter 71) is set to: 0 → M27 connector 1 → CN11B optical connector (The data output by the position coder connected to the second spindle control unit is captured using the optical cable.) (The M27 connector is provided on the memory board while the CN11B optical connector is provided on the first spindle control unit.)

In any of the three configurations, control method type A or B can be selected by specifying the MSPDB bit (bit 3 of parameter 70).

NOTE

Connection when a serial spindle interface is used

When a serial spindle interface is used, the NC and the amplifier unit for spindle control are connected using a single optical cable (COP5 on the NC and CN11A on the spindle control unit). The position coder and other detectors are not directly connected to the NC unit but are instead connected to the amplifier unit for spindle control. The information obtained from the position coder or other detectors is supplied from the amplifier unit used for spindle control to the NC along the optical cable.

When two serial spindles are connected, the CN11B connector of the first amplifier unit used for spindle control and the CN11A connector of the second amplifier unit used for spindle control are connected. The position coders and other detectors are connected to the amplifier units for spindle control in the same way as when a single serial spindle is used. The information detected by the second spindle control amplifier unit is supplied from the first spindle control amplifier unit to the NC, using the optical cable.

Relationship with other optional 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 or 2nd spindle). The spindle selection signal (SWS1 to SWS3) for the spindle must stay set at “1” during machining using this function.

- **Spindle speed fluctuation detection**

When the spindle speed fluctuation detection function is combined with multi-spindle control, two position coders can be used. Monitor the states of the second position coder selection signal (PL2SLC) and spindle selection signals (SWS1 to SWS3).

- **Actual spindle speed output**

The actual spindle speed output function conveys speed information obtained from the selected position coder specified by the 2nd position coder selection signal (PC2SLC) to the PMC.

When the parameter HSO (No. 0070#2)=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 functions 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 and third spindles can be controlled as usual.

- **Polygon turning**

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.

- **Spindle synchronization, simplified synchronization control**

During spindle synchronization or simplified synchronization control, the second spindle operates in phase with the first spindle. Multi-spindle control for the first and third spindles can be used during synchronization control, but multi-spindle control for the second spindle is disabled.

- **Rigid tapping**

Using the spindle selection signal (SWS1 to SWS3), rigid tapping can use which the three spindles as the rigid tap spindle. There are certain restrictions:

- Set the SWS1 to SWS3 signals before directing rigid tapping;
- Do not switch the SWS1 to SWS3 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.

In rigid tapping, the first spindle and the first position coder form a position loop, while the second and third spindles and the second position coder form another position loop, irrespective of the status of the position coder select signal (PC2SLC). When the SRLMSPD bit (bit 2 of parameter 0071) is set to 1, that is, when serial spindles are used as the first and second spindles and an analog spindle as the third spindle, the position loops are formed independently of the status of the position coder select signal (PC2SLC) and the setting of the SSMPC bit (bit 3 of parameter 0071) used for selecting the second position coder. One of the position loops is formed by the first spindle and the position coder connected to the first spindle control unit, while the other is formed by the second spindle and the position coder connected to the second spindle control unit. The actual spindle speed display depends on the status of the position coder select signal (PC2SLC) and the setting of the SSMPC bit (bit 3 of parameter 0071), used for selecting the second position coder.

- **0–TTC**

Refer to 9.4 “Spindle Control for 0–TTC”.

Signal

Spindle Selection Signal SWS1, SWS2, SWS3 <G145#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 <G145#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 rpm for output to the first spindle.
0 : Sets 0 rpm for output to the first spindle.

***SSTP2** 1 : Does not set 0 rpm for output to the second spindle.
0 : Sets 0 rpm for output to the second spindle.

***SSTP3** 1 : Does not set 0 rpm for output to the third spindle.
0 : Sets 0 rpm for output to the third spindle.

Gear selection signal GR21 <G145#6> GR31 <G145#7>

[Classification] Input signal

[Function] Gear selection signals for 2nd and 3rd spindle when multi–spindle is equipped (2–stage). Use GR1 and GR2 <G118#2, #3> 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 <G146#7>

[Classification] Input signal

[Function] Position coder selection signal used for control.

PC2SLC 1 : Uses feedback pulses obtained by the second position coder for control.
0 : Uses feedback pulses obtained by the first position coder for control.

When the second position coder is not installed, do not switch this signal and always select the first position coder.

Spindle enable signal ENB2<F164#2> ENB3<F164#3>

[Classification] Output signal

[Function] These signals inform PMC of whether or not to perform output to the second and 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 9.11.)

[Output condition] ENB2 1 : Outputs a value other than 0 to the second spindle control unit.
0 : Outputs 0 to the second spindle control unit.

ENB3 1 : Outputs a value other than 0 to the third spindle control unit.
0 : Outputs 0 to the third spindle control unit.

Spindle control signal by PMC

1st spindle SIND, SSIN, SGN, <G125#7, #6, #5> (Input)
R12I to R01I<G125#3 to G032#0> (Input)

2nd spindle SIND2, SSIN2, SGN2, <G107#7, #6, #5> (Input)
R12I2 to R01I2<G107#3 to G106#0> (Input)

3rd spindle SIND3, SSIN3, SGN3, <G109#7, #6, #5> (Input)
R12I3 to R01I3<G109#3 to G108#0> (Input)

[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 a 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 and third spindles.

When multi–spindle control is being used, and TYPE–A is selected (bit 3 (MSPDB) of parameter No. 0070 is set to 0), the signals for the second and third spindles cannot be used.

For details of each signal, see Section 15.4.

NOTE
The addresses of the SIND signals for spindles (G106 to G109) may also be used for the external key input signals. When the multi–spindle function is used, the external key input function cannot be used.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G106	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G107	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G108	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G109	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
G120		*SSTP						
G124	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G146	PC2SLC					GR2	GR1	
F164					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 this function.

	#7	#6	#5	#4	#3	#2	#1	#0
0064	P2SG2	P2SG1						

[Data type] Bit

P2SG2, P2SG1 Gear ratio of spindle to second position coder

Magnification	P2SG2	P2SG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

	#7	#6	#5	#4	#3	#2	#1	#0
0070						MSPDB		

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).

	#7	#6	#5	#4	#3	#2	#1	#0
0071				SRL2SP	SSMPC	SRLMSP		ISRLPC

[Data type] Bit

ISRLPC When a serial spindle is used:

1: The position coder signal connected to the M27 connector is captured.

0: The position coder signal being fed along the optical cable is captured.

SRLMSP When a serial spindle is used, the spindle configuration for multi–spindle control is set as follows:

1: The first and second spindles are serial, while the third spindle is analog.

0: The first spindle is serial, while the second and third spindles are analog and are connected to the analog interface board.

SSMPC When a serial spindle is used with the multi–spindle function and when the SRLMSP bit (bit 2 of parameter 71) is set to 1 (see above):

1: The position coder signal connected to the second spindle control unit, which is set as the second position coder, is captured.

SRL2SP The number of serial spindles to be connected is:

1: 2.

0: 1.

0556	Maximum spindle speed

[Data type] Word

[Unit of data] rpm

[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.

3 When the multi–spindle control option is selected (T series), set the maximum speed for each spindle in the following parameters:

NOTE

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.

0613	PSANG2

PSANG2 Set the gain adjustment data for the second spindle. The setting method is the same as that for the first spindle (parameter 516).

0614	SPDLC2

SPDLC2 Set the spindle speed offset compensation value for the second spindle. The setting method is the same as that for the first spindle (parameter 539).

0615	Maximum spindle speed for gear 1 of the second spindle
0616	Maximum spindle speed for gear 2 of the second spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the second spindle.

> **NOTE**
> These parameters are used for the multi–spindle control.

0617	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 for controlling the multi–spindles.

0618	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

[Method of adjustment] 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. 0618.

$$\text{Setting value} = \frac{-8191 \quad \times \text{ offset voltage (V)}}{12.5}$$

5) After the parameter has been set, command a spindle speed whose analog output becomes 0 and confirm the voltage becomes 0V.

0619	Maximum spindle speed for gear 1 of the third spindle
0620	Maximum spindle speed for gear 2 of the third spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed for each gear of the third spindle.

> **NOTE**
> These parameters are used for 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 <G146#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 A, SIND functions only when SWS1 is set to "1". In Type B, SIND functions whether SWS1 is "1" or "0"; each spindle is selected by either of its respective SWS 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 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, and third 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 spindle to perform orientation rotation.

9.11 RIGID TAPPING

9.11.1 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 9.11.2)
- Addition of a sequence to the PMC (described in 9.11.6 and 9.11.7)
- Setting of related parameters (described in 9.11.8)

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
- M series: Up to three stages T series: Up to four stages
  (Shared by the second spindle. Up to two stages for the second spindle.)

CAUTION
1 The description given in this section covers up to the fourth axis.
2 In the case of the M series, when both the CT3G (bit 1 of parameter 0077) and the VALT (bit 6 of parameter 0037) are set to 1, the maximum spindle speed for the third gear is set in parameter 0617.

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, and Z, as well as axes parallel to the basic axes, by setting the corresponding parameters accordingly (bit 6 (FXY) of parameter No. 0057).

The spindle operations of G84 and G74 are reversed with respect to each other.

The first spindle allows rigid tapping, even if the spindle is an analog or serial spindle.

- **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.

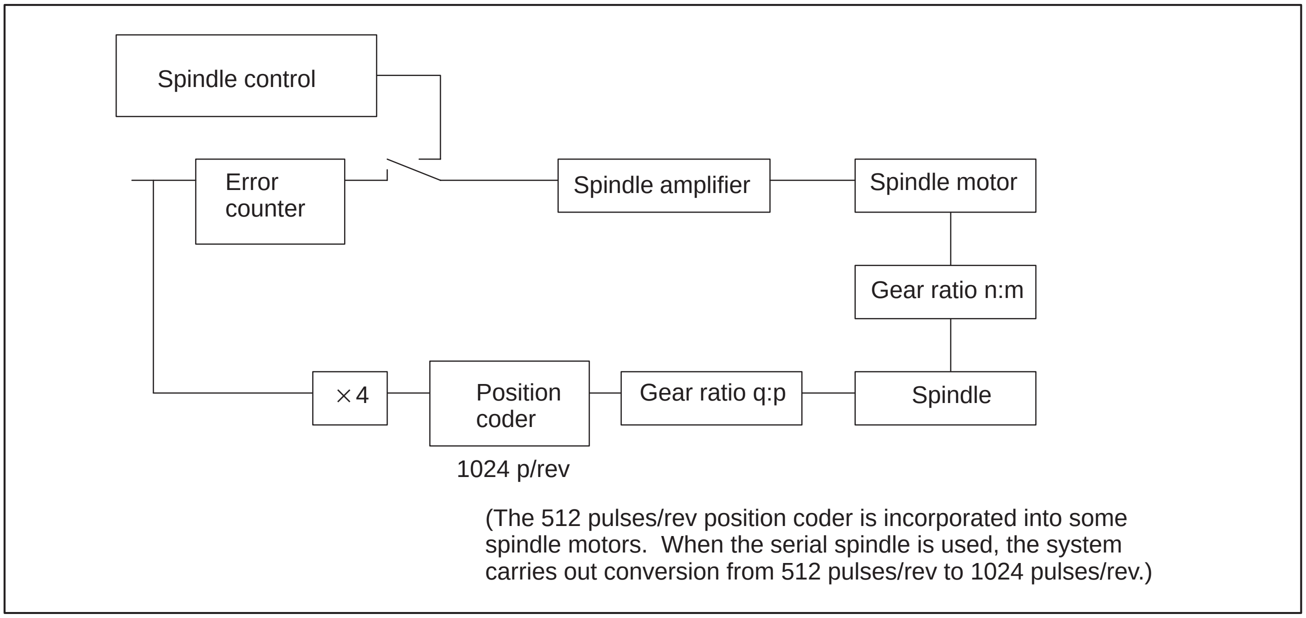
In addition to rigid tapping using the first spindle (either analog or serial), rigid tapping using the second spindle (serial) and third spindle (analog) is also possible when multi–spindle control is being used.

For 0–TTC, rigid tapping can be performed using a combination of the spindle and tapping axis selected in each path.

Rigid tapping using a mixture of paths is not allowed.

9.11.2 Connection Among Spindle, Spindle Motor, and Position Coder

As shown in the figure below a gear 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 (M series) or four gear stages (1st spindle of T series), two gear stages (2nd or 3rd spindle of T series) can be provided between the spindle and the spindle motor. The gear ratio is arbitrary. The spindle move distance per spindle motor rotation is different, based on the gear ratio. The speed command to the spindle motor must 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 Nos. 0663 to 0668 (M)/Nos.0427 to 0434 (T), or parameter set PSG1 and PSG2 No. 0028#6, 7(M)/No.0003#6, #7(T), parameter P2SG1 and P2SG2 No.0064#6, 7 for 2nd spindle of T series. Which parameter set to use is specified by parameter VALPC No.0063#3 (M)/No.0063#6(T).

· **Arbitrary gear ratio (VALPC=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 VALPC to 1 and set the gear ratio using parameter Nos.0663 to 0668 (M)/Nos.0427 to 0434(T).

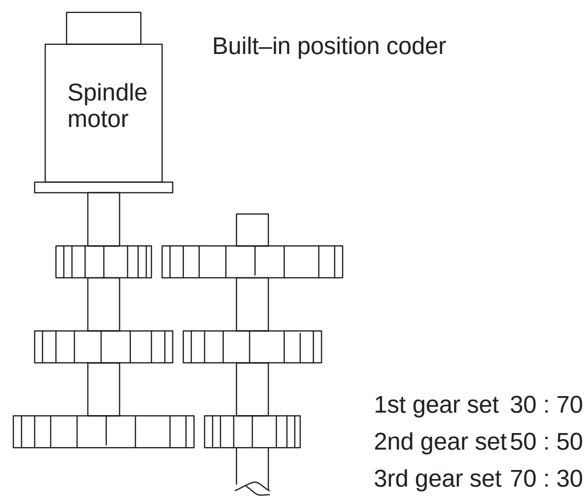
When position coder is mounted on a spindle, the gear ratio for the spindle motor and position coder cannot be changed by shifting the spindle motor and spindle gears. Parameters Nos. 0663 to 0665(M)/Nos.0427 to 0430(T) must all specify the same value for the teeth of the individual spindle gears. Parameters Nos.0666 to 0668(M)/Nos. 0431 to 0434(T) 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.)

The M series allows up to three stages, regardless of which gear selection method has been selected. (Parameter 0663 to 0668.)

The T series supports up to four stages for the first spindle and up to two stages for the second or third spindle (parameters 0427 to 0434).

Example)



Parameter No.		Set value		Meaning
M series	**T series**	**512p/rev Position coder**	**1024p/rev Position coder**	
0663	0427	70		Number of teeth of the 1st gear for the spindle side
0664	0428	50		Number of teeth of the 2nd gear for the spindle side
0665	0429	30		Number of teeth of the 3rd gear for the spindle side
0666	0431	60 Note)	30	Number of teeth of the 1st gear for the position coder side
0667	0432	100 Note)	50	Number of teeth of the 2nd gear for the position coder side
0668	0433	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 PSG1 and PSG2 (No.0027#6, #7(M)/No.0003#6, #7(T)). 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 of T series, set it to parameters P2SG1 and P2SG2 (No.0064#6, #7).

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.

Built–in position coder 512p/rev	Gear ratio		Parameter		Gear ratio of spindle to position coder	Detection unit (deg)
	Spindle motor	Spindle	PSG2	PSG1		
	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 (F150#2) and gear selection signals GR1O, GR2O, and GR3O (F152#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 S11 to S58 (F185 to F187) to the PMC. Using the PMC, determine whether gears need changing, and make the change if needed. Input gear selection signals GR1 and GR2 <G123#2, #3(M)/G118#2, #3(T)>, GR21 <G145#6> for 2nd spindle of T series, or GR31 <G145#7> for 3rd spindle of T series 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 (G229#3,#2 for the first spindle, and G223#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 = $\frac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed = $\frac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{Medium speed gear ratio}}$
High–speed gear	Maximum medium–speed gear speed +1 revolution	Maximum high–speed gear speed = $\frac{\text{Maximum spindle motor speed} \times \text{L\%}}{\text{High–speed gear ratio}}$

NOTE
The table shows sample spindle speed ranges for a spindle having three gears. In the table, L% is a spindle motor protection constant (up to 100), provided to caluculate the maximum speed. (Different L values can be specified for individual gears if gear change method B is selected for M type gear selection (when the LGCM (bit 6 of parameter 0035) is set to 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. "+ a" 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 6 (G84S) of parameter No. 0012 = 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. 0540).

The table above shows the maximum medium-speed gear speed during rigid tapping for medium-/high-speed gear change position E (parameter No. 0556).

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. 0615(M)/No.0406(T) and specify loop gains for each gear for parameter Nos. 0669 to 0671(M)/ Nos.0407 to 0410(T). Unless "0" is specified for parameter No.0615(M)/No.0406(T), the loop gains for each gear are disabled, and the loop gain common to all gears, the value of parameter No.0615 (M)/No.0406(T), is enabled.

Specify the time constant and the maximum spindle speed for each gear. Use parameters Nos.0692, 0693, and 0613(M)/Nos.0415 to 0418 (T) to specify the time constant.
Use parameters Nos.0694, 0695 and 0617 (M)/Nos.0423 to 0426(T) to specify the maximum spindle speed.

For M type gear selection method, set the maximum spindle speed to parameter No. 0617, irrespective of the number of gear stages used.

Setting bit 5 (RGCTO) of parameter No.0035(M)/bit3(RGCTO) of parameter No.0029(T) to "1" enables setting of the extraction time constant for each gear set. Specify the extraction time constant for each gear in parameter Nos.0400 to 0402(M)/Nos.0419 to 0422(T).

If bit 3 (VSLRC) of parameter No.0063 (M)/bit 6 (VSLRC) of parameter No.0063(T) 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 M series, four gear sets with T series, or two gear sets with 2nd position coder of T series. However 1:8 to 8:1 is the recommended value.

9.11.3 Rigid Tapping Specification

- **Feed rate**

In rigid tapping mode, the tapping axis is fed at a rate specified by F; the spindle speed is S × 360(deg/min). Override is invalid for both of them. An override of up to 200% can be applied to withdrawal operations by setting bit 4 (RGDOV) of parameter No. 0063 to "1", and setting an appropriate override value in parameter No.0258(M)/No.0254(T). The time constant for withdrawal operations can be modified by bit 5 (RGCTO) of parameter No.0035(M)/bit3(RGCTO) of parameter No.0029(T); when it is set to "1", the values in parameter Nos.0400 to 0402 (M)/Nos.0419 to 0422 (T) are used as the time constant for withdrawal.

- **Acceleration and deceleration**

Linear acceleration/deceleration is valid for rigid tapping.

- **Override**

Override is invalid for rigid tapping. Fixed override can be applied to withdrawal operations by setting bit 4 (RGDOV) of parameters No. 0063 or parameter No.0258 (M)/No.0254(T).

- **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, interlock, and single block**

The feed hold, interlock, and single block functions are effective for G84 (G74).
The feed hold and single block functions in rigid tapping mode can be nullified by setting bit 2 (RGMFH) of parameter No.0388 (M)/bit 3 (RGMFH) of parameter No. 0397 (T) 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.0255 (M)/Nos. 0214 to 0217 (T). The backlash compensation is normally made for the tapping axis.

9.11.4 Display Data on the Diagnosis

For rigid tapping adjustment, the parameter screen displays information related to rigid tapping.

The following information items are displayed, when rigid tapping is executed:

· Spindle position deviation → Diagnosis No.627(M)/No.0435(T)

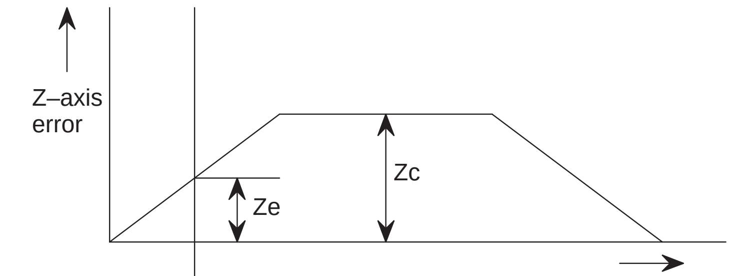
· Number of command pulses distributed to the spindle (momentary value) → Diagnosis No.628(M)/No.0436(T)

· Cumulative number of command pulses distributed to the spindle → Diagnosis No.799(M/T)

· Momentary error difference between the spindle and tapping axis → Diagnosis No.696(M)/No.0437(T)

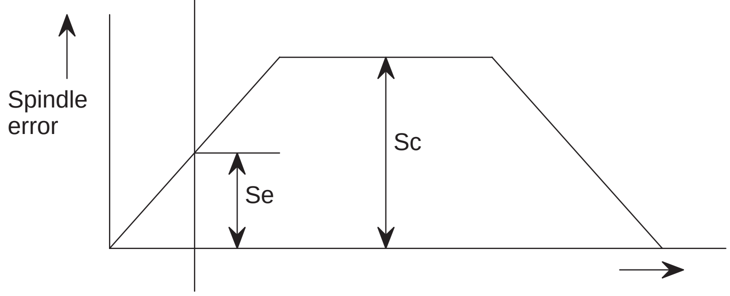
· Maximum error difference between the spindle and tapping axis → Diagnosis No.697(M)/No.0438(T)

Diagnosis No.696 (M)/No.0437(T) is cleared to “0” when rigid tapping mode is set or canceled, and diagnosis No.697 (M)/0438(T) is cleared to “0” in the positioning of the rigid tapping cycle.
The following figure shows the tapping axis as the Z axis.



$$Zc = \frac{\text{Speed}}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \quad \text{(Theoretical value)}$$

Ze = Z–axis error counts (measured value)



$$Sc = \frac{\text{Speed} \times 360}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \quad \text{(Theoretical value)}$$

Se = Spindle error counts (measured value)

Speed : mm/min or inch/min
Gain : 0.01 s^{-1}
Detection unit :mm, inch, or deg
Speed : rpm

$$\Delta Z = \frac{Ze}{Zc} \times 100 \ [\%]$$

$$\Delta S = \frac{Se}{Sc} \times 100 \ [\%]$$

The error difference between the spindle and Z axis can be obtained by $\Delta S - \Delta Z$. This value is sampled at intervals of 64 ms.

Diagnosis screen

- **Spindle position deviation**

0435	SPINDLE MOTION ERROR	(T series)
0627	SPINDLE MOTION ERROR	(M series)

Spindle position deviation during rigid tapping

[Data type] Word

[Unit] Pulse

- **Number of pulses distributed to the spindle**

0436	SPINDLE MOTION PULSE	(T series)
0628	SPINDLE MOTION PULSE	(M series)

Number of pulses distributed to the spindle during rigid tapping

[Data type] Word

[Unit] Pulse

- **Error difference between the spindle and tapping axis (momentary value)**

0437	RIGID ERROR	(T series)
0696	RIGID ERROR	(M series)

Momentary error difference between the spindle and tapping axis during rigid tapping (signed)

[Data type] Word

[Unit] %

- **Error difference between the spindle and tapping axis (maximum)**

0438	RIGID ERROR (MAX)	(T series)
0697	RIGID ERROR(MAX)	(M series)

Maximum error difference between the spindle and tapping axis during rigid tapping (absolute value)

[Data type] Word

[Unit] %

- **Cumulative number of pulses distributed to the spindle during rigid tapping**

0799	SPINDLE PULSE(SUM)

Cumulative number of pulses distributed to the spindle during rigid tapping

[Data type] Two–word

[Unit] Pulse

9.11.5
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.4.2 of the “Operator’s Manual for Machining Center (B–61404E) ”.

G□□ X(Z)__ C__ Z(X)__ R__ P__ F__ K__ (M□□) ;

- G□□: G84 — Front tapping cycle; G88 — Side tapping cycle
- X(Z)__ C__: Specifies a tapping position.
- Z(X)__: Specifies the position of the bottom face of a tapped hole
- R__: Specifies the position of an R point
- P__: Dwell time at the bottom face of a tapped hole and upon R point return
- F__: Cutting feedrate
- K__: Number of repeats
- (M□□): M code for C–axis clamping

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 can be canceled by G80;. Note, however, that a G code for another canned cycle, or a group 01 G code can 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 3 (G84RGD) of parameter No. 0076 to 1)

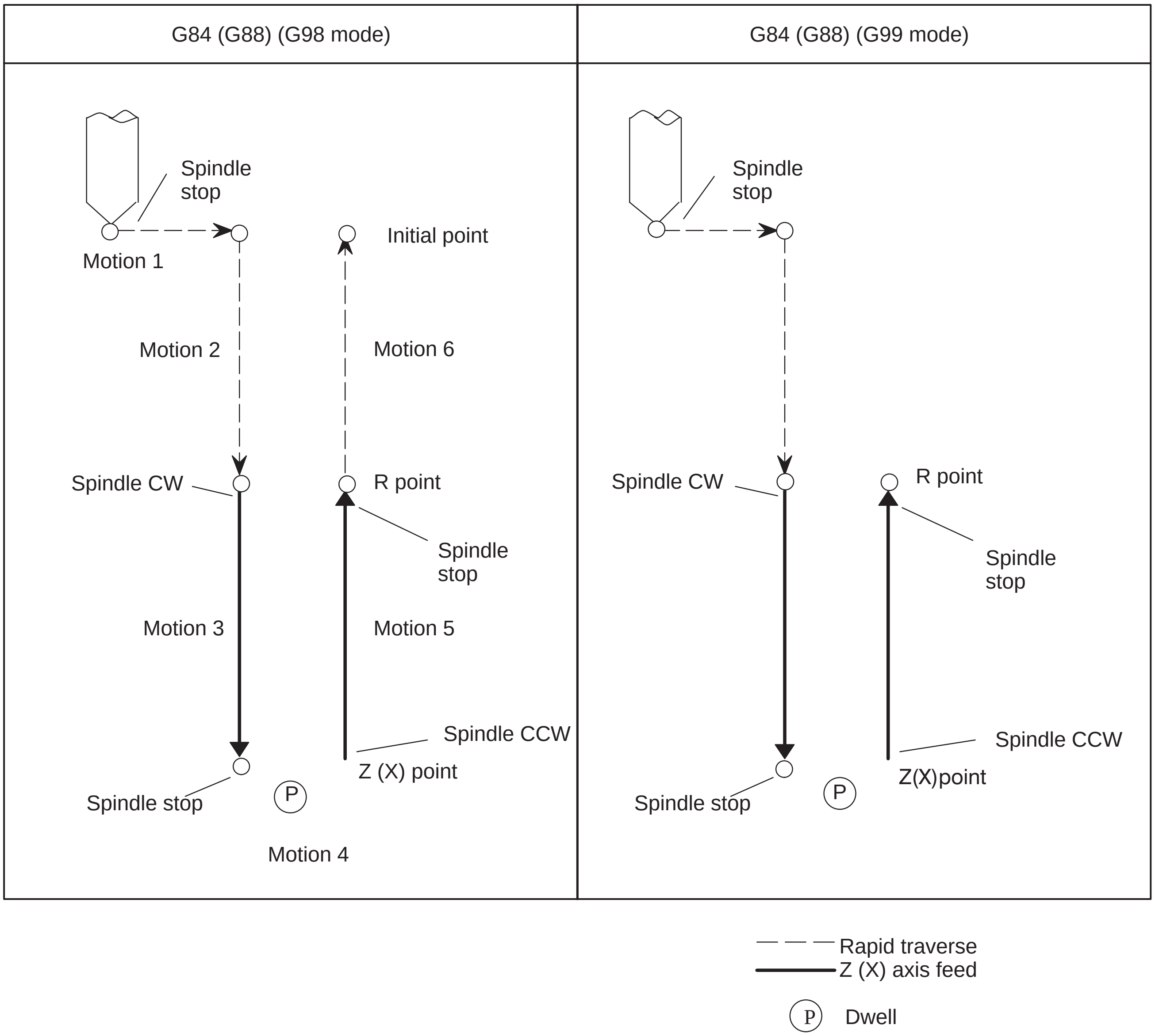
```
G□□X(Z)__C__Z(X)__R_P_F_K_S**** (M□□);   ┐
X(Z)__C__;                               │ Rigid
X(Z)__C__;                               │ tapping
  .                                      │ mode
  .                                      │
G80;                                     ┘
```

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. 0423 to 0426) 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·G88 (Tapping cycle)



CAUTION
During cutting feed along the Z–axis (X–axis), the feedrate override is assumed to be 100%. The spindle speed override is also assumed to be 100%. For a retract motion (motion 5), a fixed override of up to 200% can be applied by specifying bit 4 (RGDOV) of parameter No. 0063 and parameter No. 0258 (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 rpm.

```
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.

9.11.6 Signal

9.11.6.1 Signals for the rigid tapping function

Rigid tapping signal RGTPN <G123#1>

[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 <F165#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”.

9.11.6.2 Signals related to S code output

Spindle enable signal ENB <F149#4>

Second and third spindle enable signals ENB2 <F164#2> ENB3 <F164#3> (T series)

[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 (BCD output) S11 to S58 <F185#0 to F187#3>

Spindle–speed function strobe signal SF <F150#2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in BCD format, to the PMC.

[Output condition] When an S code is specified, the specified value is output, in binary format, with the signals. Upon the completion of output, 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 7 (SFOUT) of parameter No. 0020.

[2] T–type gear selection method

SF output depends on the setting of bit 4 (SFOB) of parameter No. 0029.

T series: The following parameter needs to be set to output S codes and SF: Bit 0 (EVSF) of parameter No. 0049 = 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**
> The timing charts, given later, give examples of gear switching by setting the parameters as follows:
> M series: SFOUT = 0, SFOB = 0
> T series: EVSF = 1

9.11.6.3 Signals related to gear switching

Gear selection signals (output) GR3O, GR2O, GR1O <F152#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).

As 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 signals.

Reference information: The table below indicates the relationship between the output signals and gear selection.

	GR3O	GR2O	GR1O
1st (low) speed gear	0	0	1
2nd (medium) speed gear	0	1	0
3rd (high) speed gear	1	0	0

Gear selection signals (input) GR2, GR1 <G123#3, #2> (M series) <G118#3, #2> (T series)

[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	0	0
2nd (medium) speed gear	0	1
3rd (high) speed gear	1	0
4th (high) speed gear	1	1

← 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 <G145#6> GR31 <G145#7> (T series)

[Classification] Input signal

[Operation] When rigid tapping with the second or third spindle is being performed (for the T series only), the signal is used in a PMC sequence.

The signal notifies the CNC of spindle gear information when the second or third spindle has been selected.

The input signal is related to gear selection as described below.

GR21 1 : The second stage is currently selected as the second or third spindle gear.

0 : The first stage is currently selected as the second or third spindle gear.

	GR21 GR31
1st speed gear	0
2nd speed gear	1

When a serial spindle is used, the serial spindle clutch/gear selection signals (G229#3, #2 for the first spindle, and G233#3, #2 for the second spindle) must be set in addition to the setting of the gear selection signal described above.

9.11.6.4 Signals related to second and third spindle rigid tapping (T series)

Gear selection signal (input)
GR21 <G145#6>
GR31 <G145#7>
(T series)

See the description of the signals related to gear switching, given above.

Signals related to multi–spindle control (T series)

Spindle selection signals SWS1, SWS2, SWS3 <G145#0, #1, #2>

[Classification] Input signal

[Operation] SWS1, SWS2, and SWS3 are used to transfer spindle commands when the multi–spindle control option is used. In rigid tapping, the signals can be shared to select a spindle to be used for rigid tapping.

To select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state		
	SWS3	SWS2	SWS1
P/S 205	0	0	0
First spindle	0	0	1
2	0	1	0
1	0	1	1
3	1	0	0
1	1	0	1
2	1	1	0
1	1	1	1

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, *SSTP3 <G145#3, #4, #5>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi–spindle control option is used. In a PMC sequence for rigid tapping, the ENB, ENB2, and ENB3 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 does not specify 0 rpm.
0 : The output to the first spindle specifies 0 rpm.

***SSTP2** 1 : The output to the second spindle does not specify 0 rpm.
0 : The output to the second spindle specifies 0 rpm.

***SSTP3** 1 : The output to the third spindle does not specify 0 rpm.
0 : The output to the third spindle specifies 0 rpm.

Second position coder selection signal PC2SLC <G146#7>

[Classification] Input signal

[Operation] This signal is used to select the second position coder when the multi–spindle control option is being used. Note, however, that it cannot be used with a spindle selected to perform rigid tapping.

1 : Control is exercised using a feedback pulse signal from the second position coder.
0 : Control is exercised using a feedback pulse signal from the first position coder.

For rigid tapping, this signal is not used. Instead, a position loop is constructed by combining the first spindle with the first position coder, or by combining the second or third spindle with the second position coder.

However, the display of the actual speed is switched by this signal, even during rigid tapping.

9.11.6.5 Signal addresses

	#7	#6	#5	#4	#3	#2	#1	#0	
G118					GR2	GR1			(T series)
G123					GR2	GR1			(M series)
G123							PGTPN		
G145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1	(T series)
G146	PC2SLC								(T series)

	#7	#6	#5	#4	#3	#2	#1	#0	
F149				ENB					
F150						SF			
F152						GR3O	GR2O	GR1O	(M series)
F164					ENB3	ENB2			(T series)
F165							PGSPM	RGSPP	(M series)
F185	S28	S24	S22	S21	S18	S14	S12	S11	
F186	S48	S44	S42	S41	S38	S34	S32	S31	
F187					S58	S54	S52	S51	

9.11.6.6 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, ENB3)

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”.

In rigid tapping using the second or third spindle with the T series, the ENB2 or ENB3 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 or third spindle output must be disabled in rigid tapping mode by setting SIND2 or SIND3 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 (S11 to S58) must be output to the PMC. The required parameter settings are described below.

· M series: Set bit 4 (SFOB) of parameter No. 0029 to 0 to output SF.

· T series: Set bit 0 (EVSF) of parameter No. 0049 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 RGTPN <G123#1> 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 S**** is outside the previously selected gear range, the spindle–speed function strobe signal SF <F150#2> and gear selection signals (output) GR3O, GR2O, GR1O <F152#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 <F150#2> and spindle–speed function code signals S11 to S58 <F185#0 to F187#3> are output to the PMC. (However, parameter setting is required to enable output of the S codes and SF signal. See the description of bit 0(EVSF) of parameter No. 0049.)

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 <G123#3, #2> (M)/<G118#3, #2> (T) 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 stage is selected, the system assumes that the third gear has been 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 or third spindle is notified to the CNC by the signal GR21 <G145#6> or GR31 <G154#7>.)

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.)

9.11.7 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. 9.11.7.1 (a) to Fig. 9.11.7.3 (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. 9.11.7.1 (a)	Fig. 9.11.7.2 (a)	Fig. 9.11.7.3 (a)
	Performed	Fig. 9.11.7.1 (b)	Fig. 9.11.7.2 (b)	Fig. 9.11.7.3 (b)
T–type	Not performed	Fig. 9.11.7.1 (c)	Fig. 9.11.7.2 (c)	Fig. 9.11.7.3 (c)
	Performed	Fig. 9.11.7.1 (d)	Fig. 9.11.7.2 (d)	Fig. 9.11.7.3 (d)

NOTE

For more information about the M/T type gear selection method, see Section 9.3 SPINDLE CONTROL. Note the following:

T series:	<u>T–type</u>	only
M series:	<u>M–type</u>	when constant surface speed control is not being used.
	<u>T–type</u>	when constant surface speed control is being used.

9.11.7.1 When M29 is specified before G84 (G74)

M type gear selection method

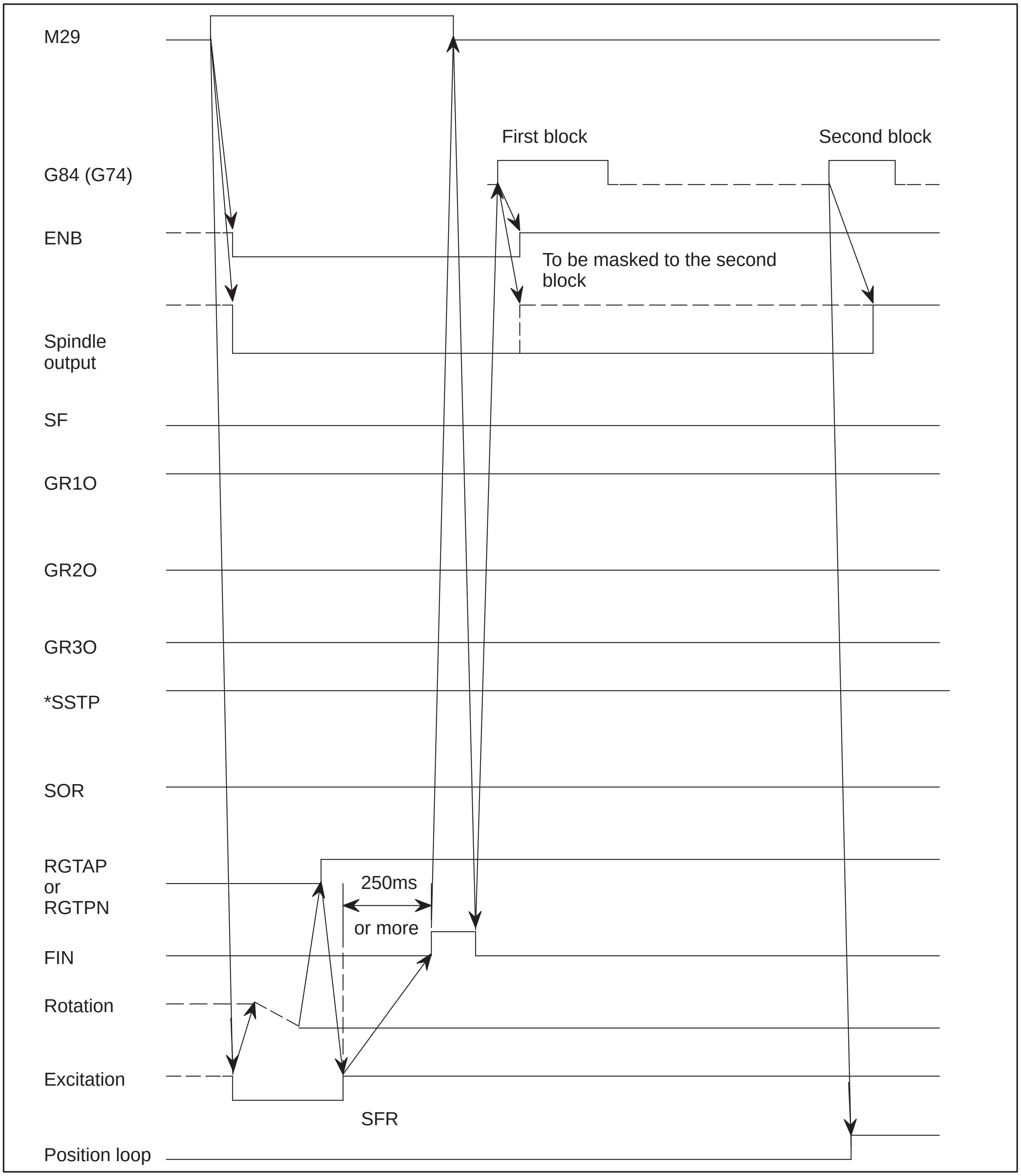
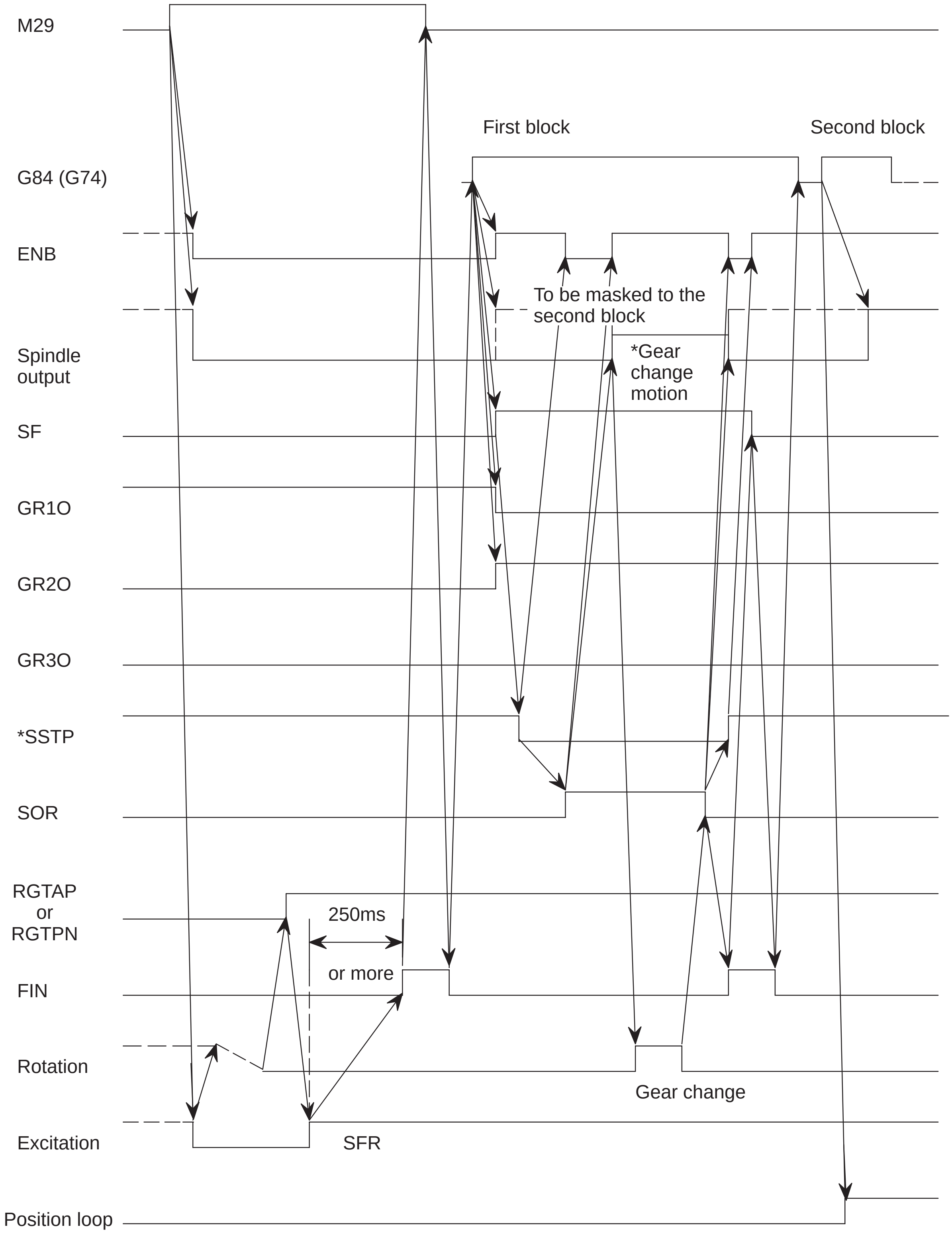


Fig. 9.11.7.1 (a) Gear is not changed



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. 9.11.7.1 (b) When gear change is performed (From low to middle gear)

T type gear selection method

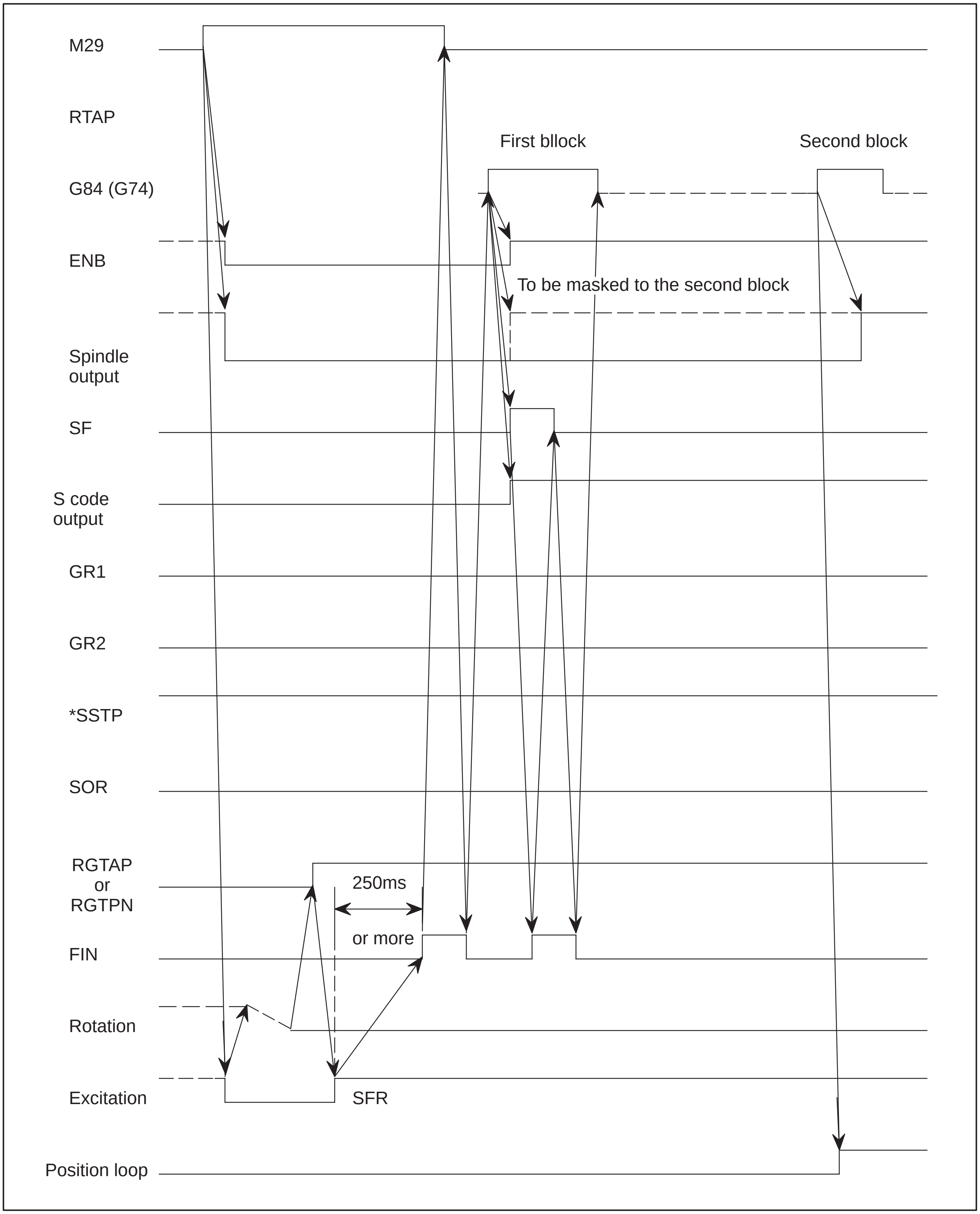
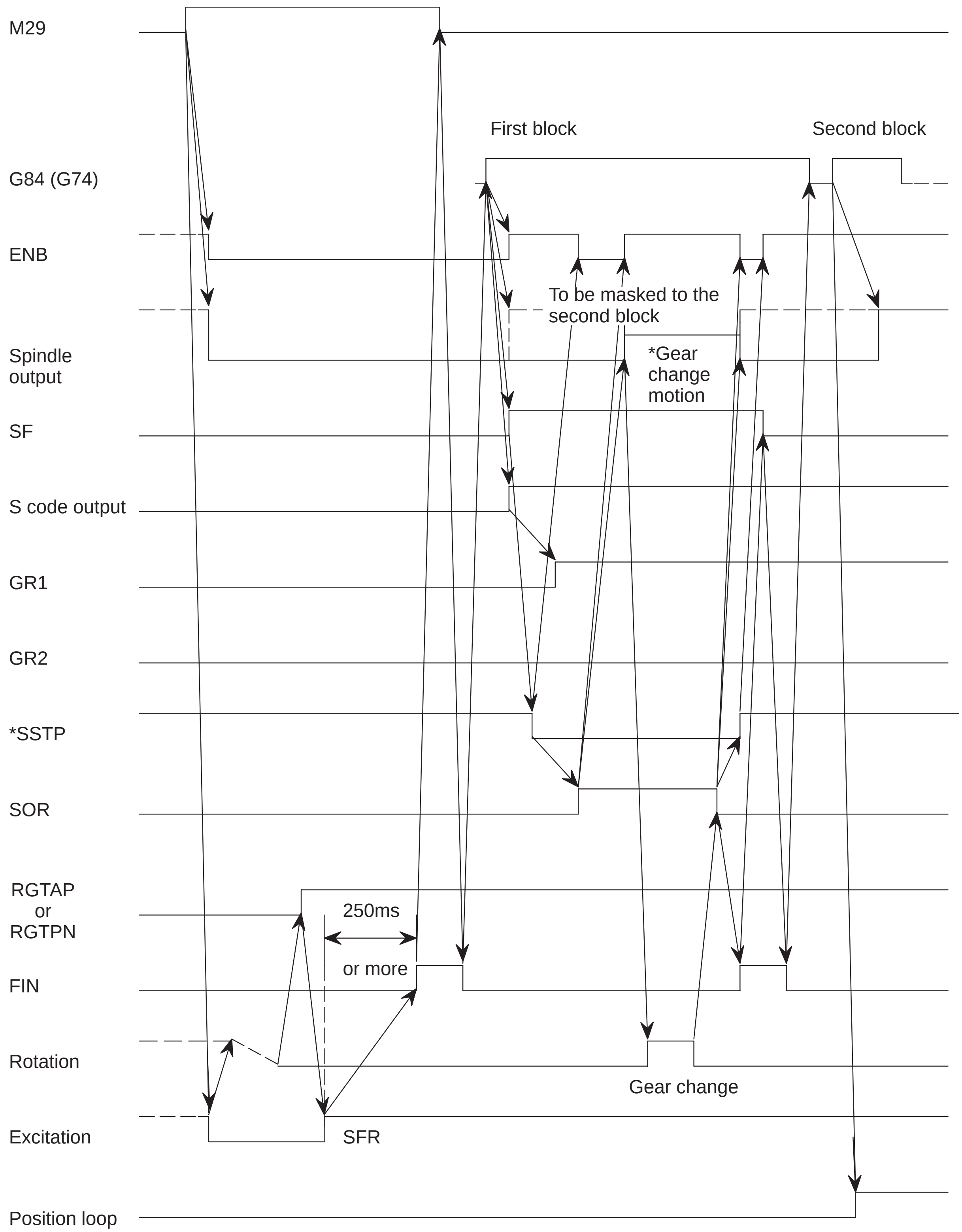


Fig. 9.11.7.1 (c) Gear change is not performed



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. 9.11.7.1 (d) When gear–change is performed (low to middle gear)

9.11.7.2
M29 and G84 (G74) are specified in the same block

M type gear selection

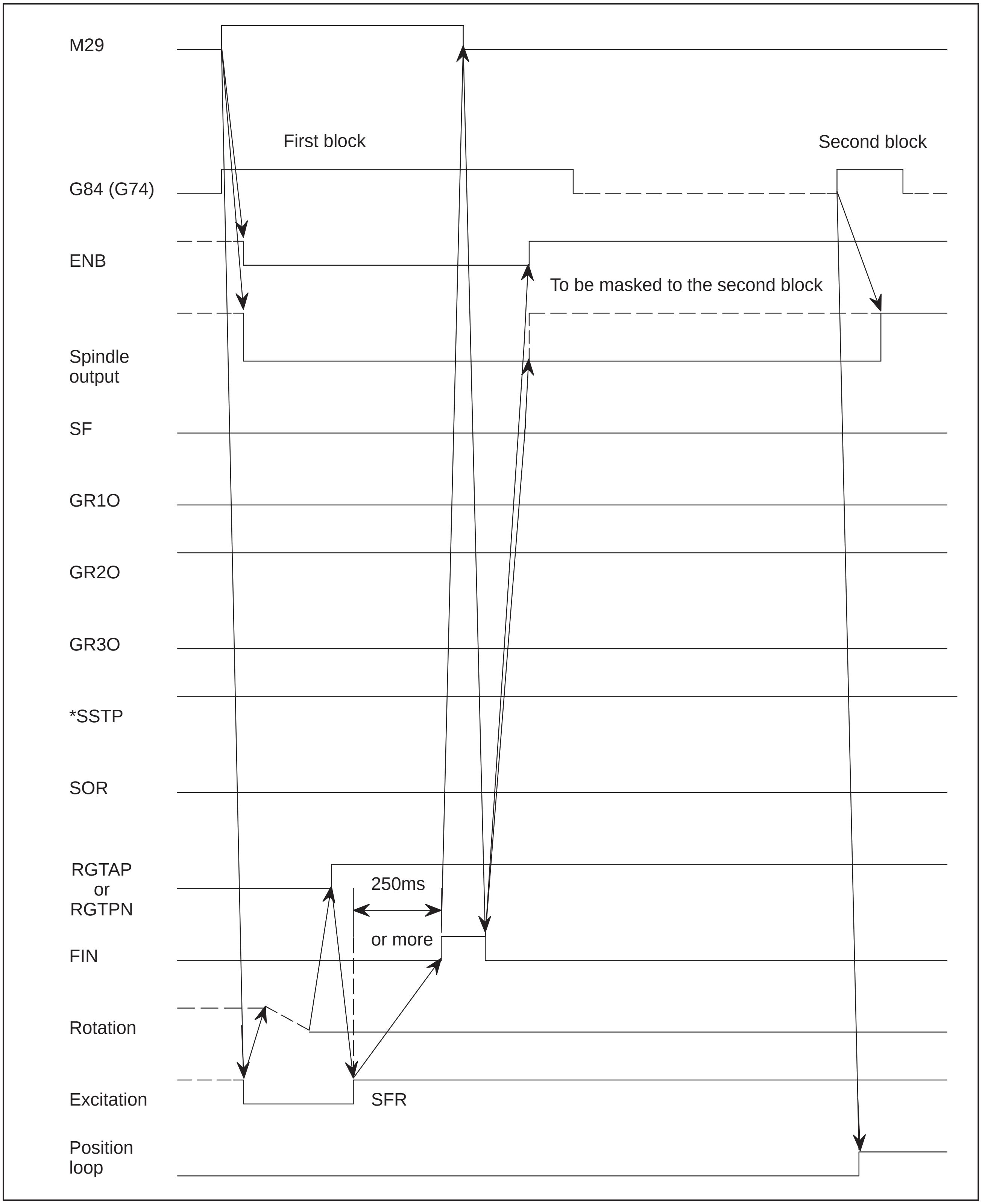
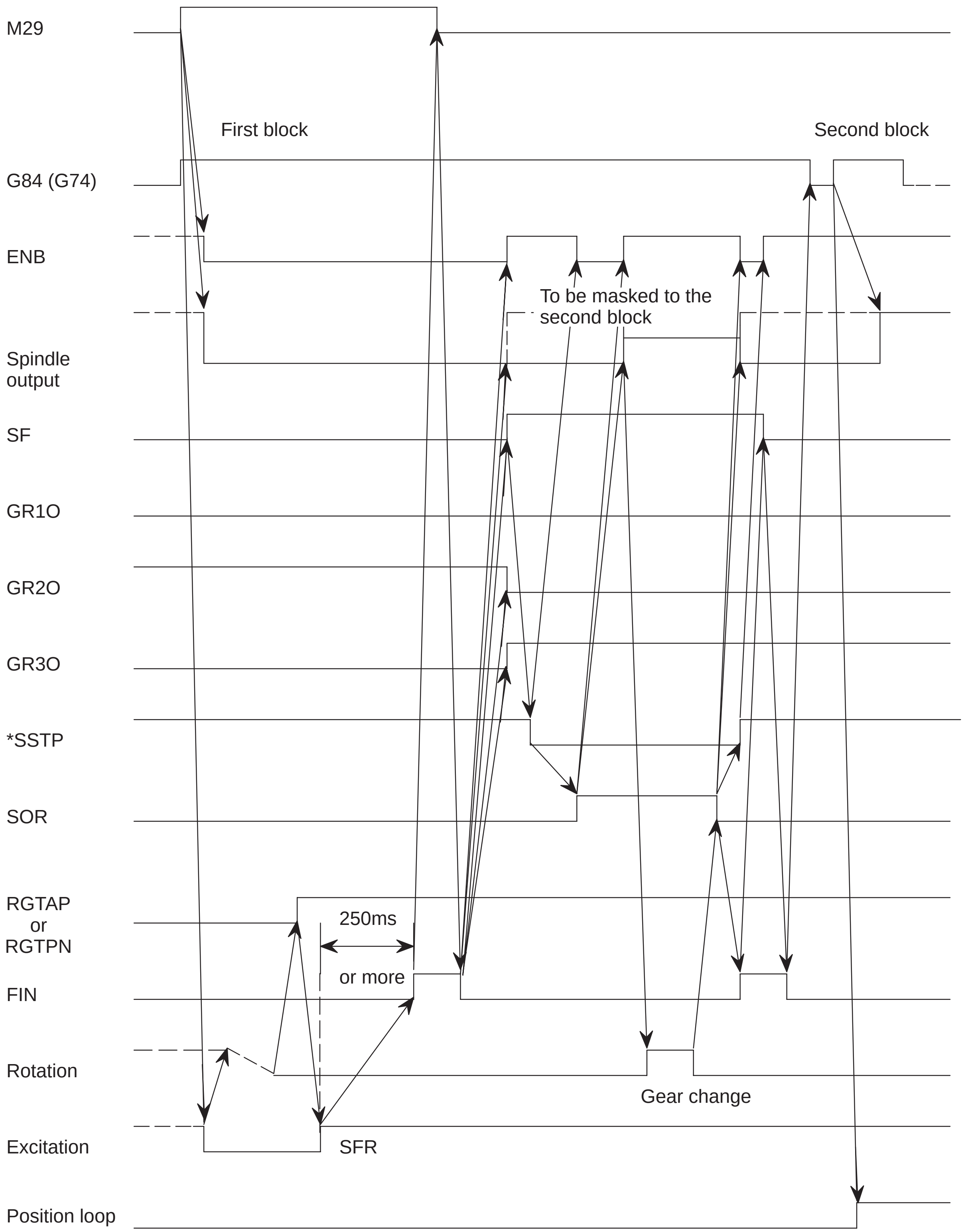


Fig. 9.11.7.2 (a) When gear–change is not performed



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. 9.11.7.2 (b) When gear–change is performed (middle to high)

T type gear selection method

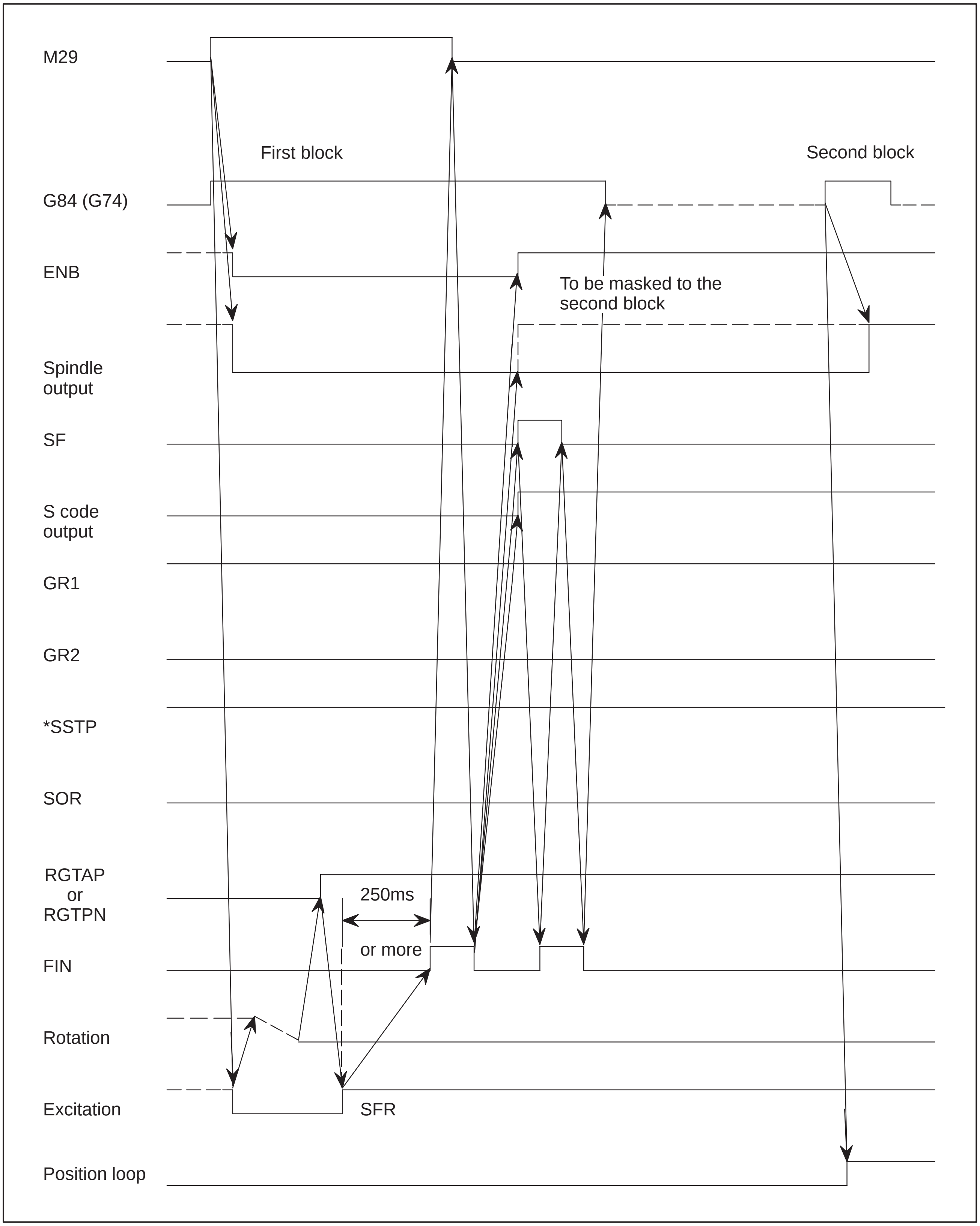
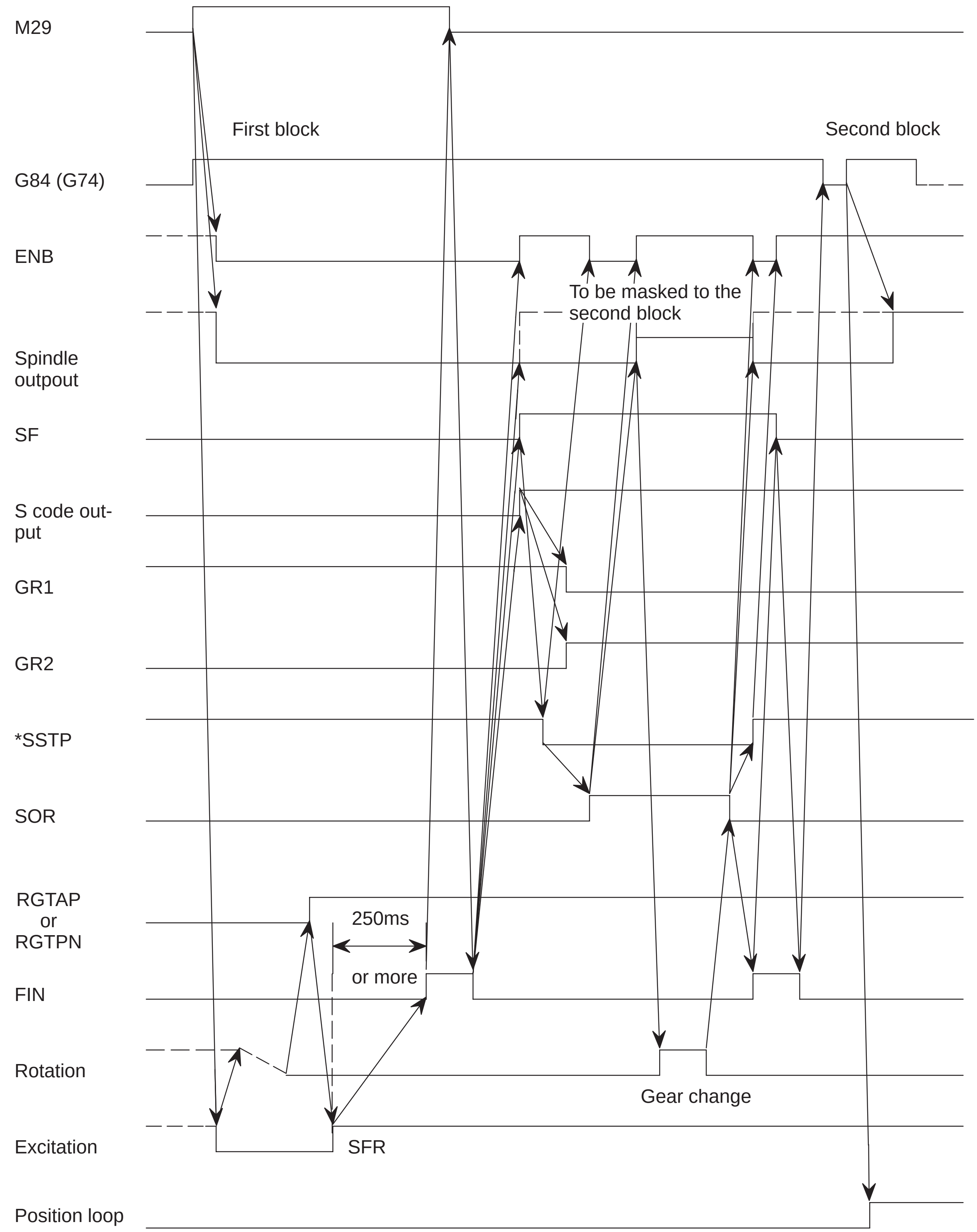


Fig. 9.11.7.2 (c) When gear change is not performed



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 9.11.7.2 (d) When gear–change is performed (middle to high gear)

9.11.7.3 Specifying G84 (G74) for rigid tapping by parameters

M type gear selection

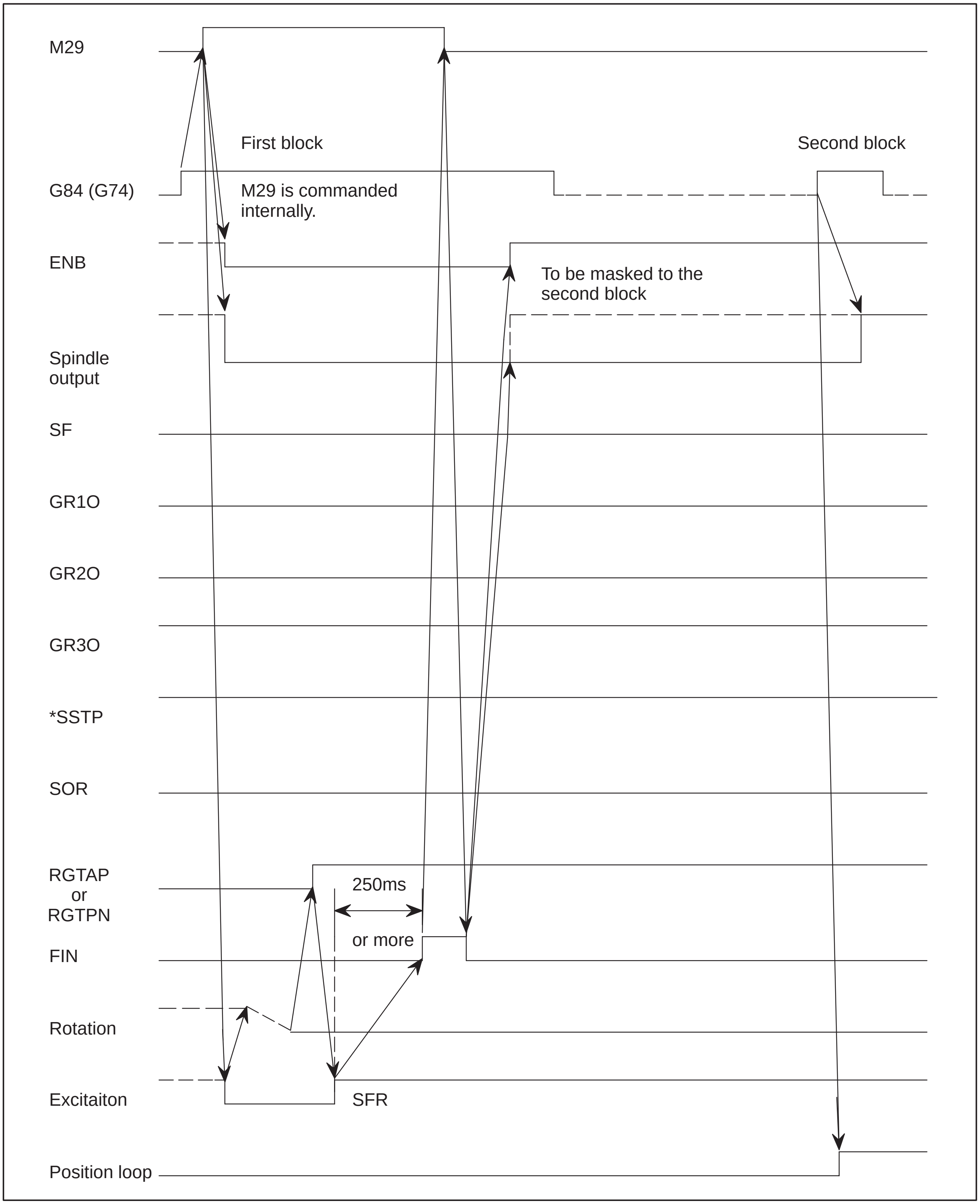
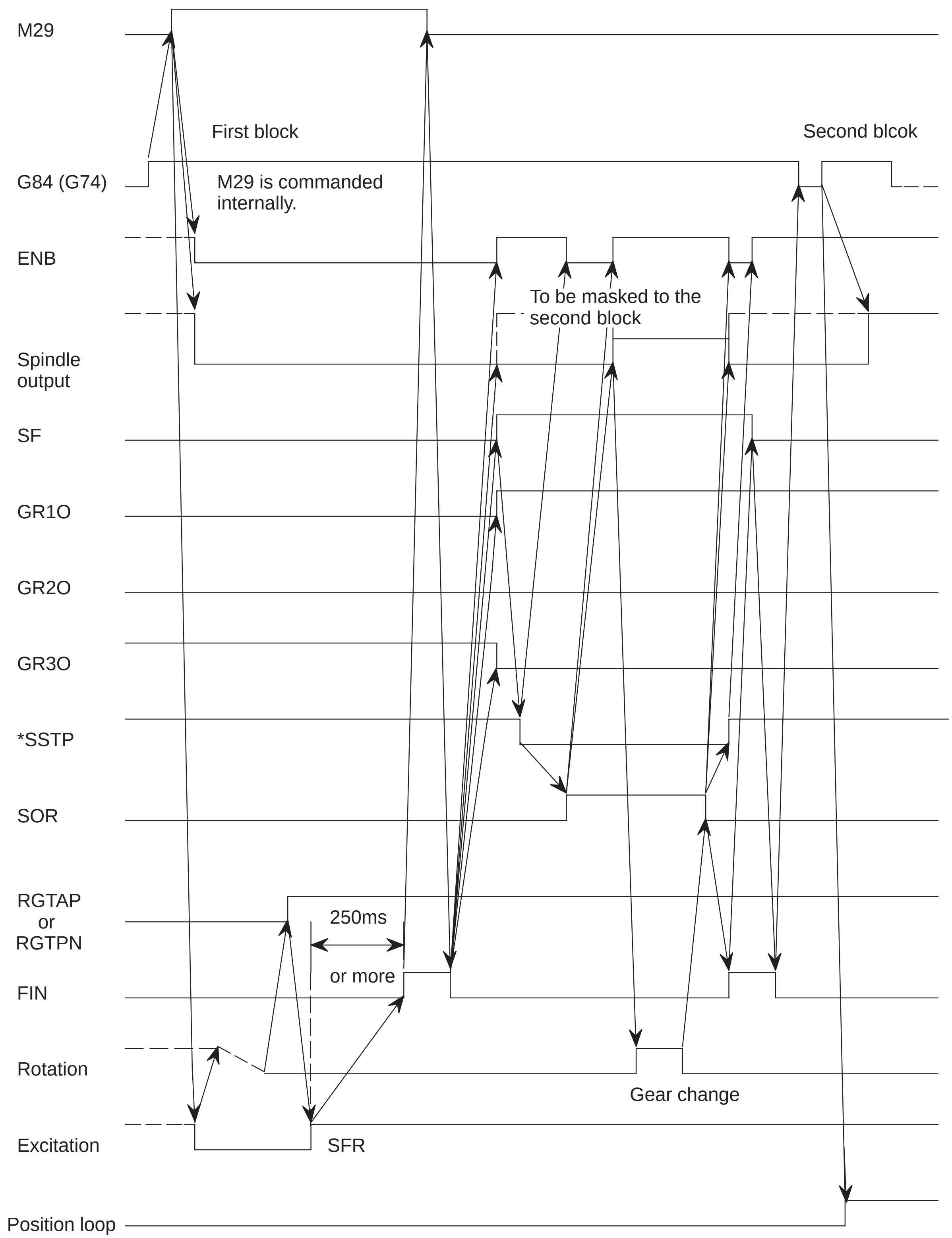


Fig. 9.11.7.3 (a) When gear–change is not performed



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. 9.11.7.3 (b) When gear change is performed (high to low gear)

T type gear selection method

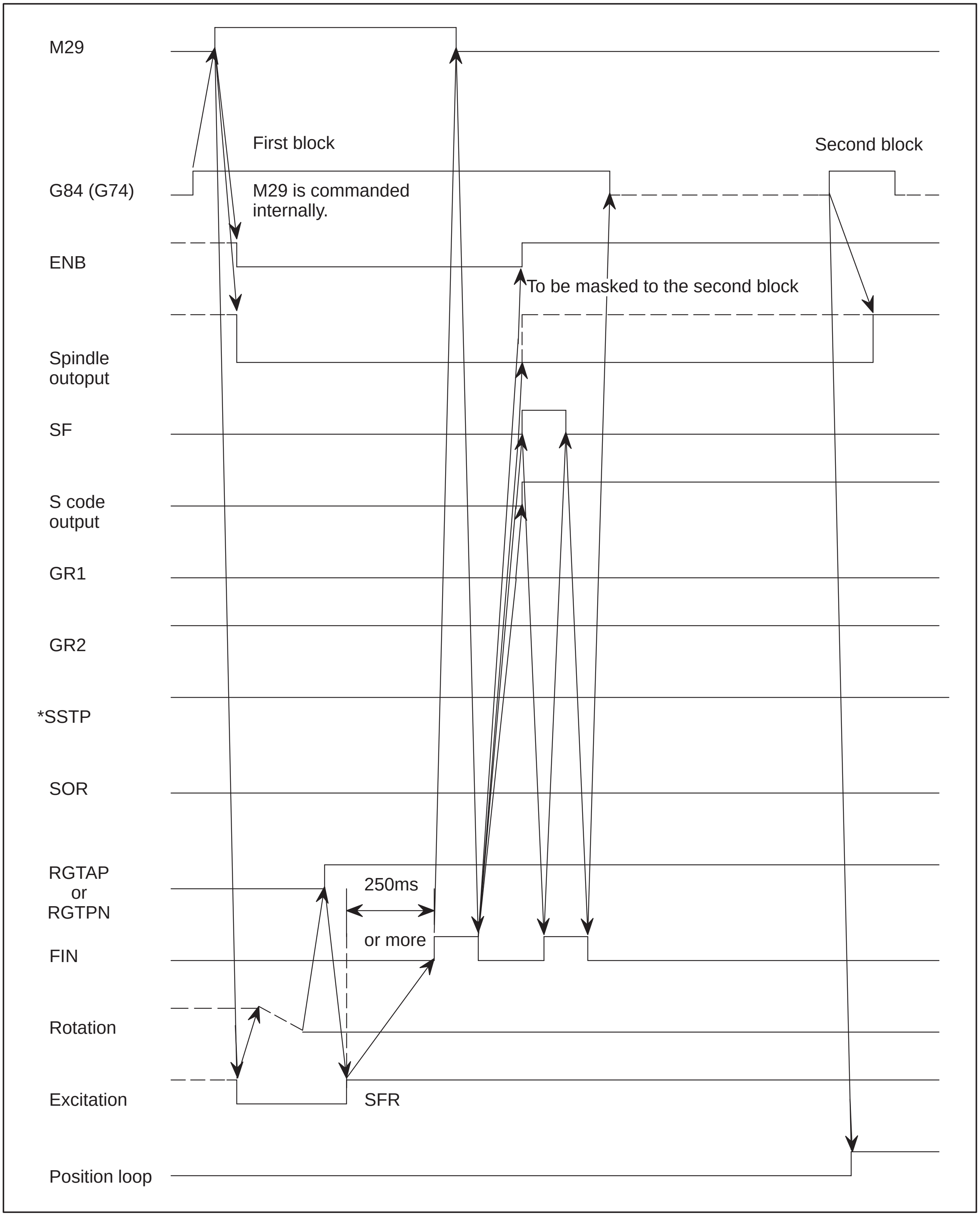
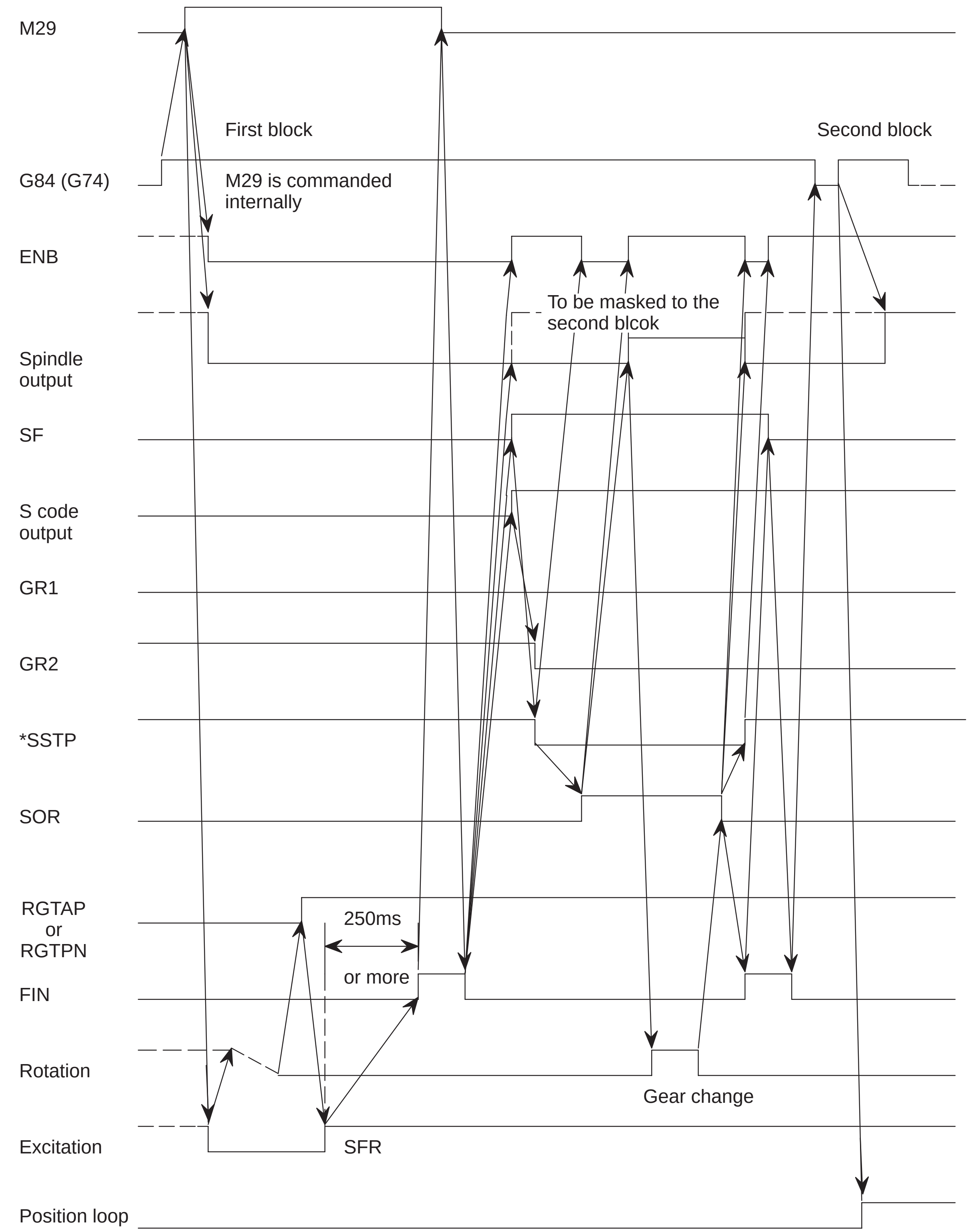


Fig. 9.11.7.3 (c) When gear change is not performed



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.

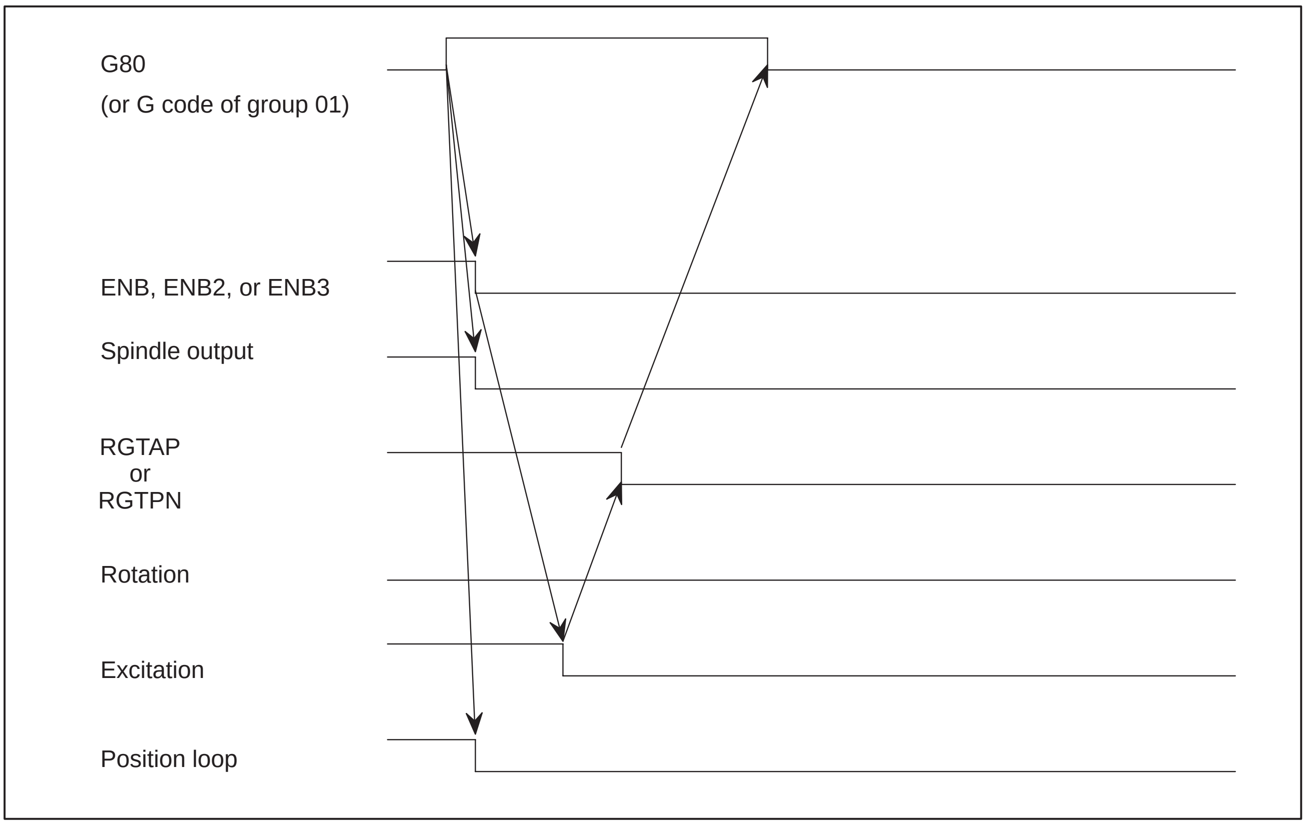
Fig. 9.11.7.3 (d) When gear–change is performed (high to low gear)

9.11.7.4 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 spindle output is produced in the same way as executing S0. Cancel the PMC rigid tapping mode at the falling edge of the ENB signal (ENB2 or ENB3 signal for 2nd or 3rd spindle of T series) 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 RGTPE (parameter no. 0040#2(M)/0041#0(T)) 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 a block contains an M code for controlling the spindle, an error may occur during processing in the PMC.

2 When RGTPE (Parameter No.0040#2(M)/0041#0(T)) 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.

9.11.8 Parameter

		#7	#6	#5	#4	#3	#2	#1	#0	
0003		PSG2	PSG1							(T series)
0028		PSG2	PSG1							(M series)

[Data type] Bit

PSG2, PSG1 Gear ratio of spindle to position coder

Magnification	PSG2	PSG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

		#7	#6	#5	#4	#3	#2	#1	#0	
0012			G84S							(M series)

[Data type] Bit

G84S 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 0540 and 0556)

		#7	#6	#5	#4	#3	#2	#1	#0	
0020		SFOUT								(M series)

[Data type] Bit

SFOUT: 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	
0029					SFOB					(M series)

[Data type] Bit

SFOB: When an S code command is issued in constant surface–speed control,
0 : SF is output.
1 : SF is not output:

	#7	#6	#5	#4	#3	#2	#1	#0	
0029					RGCTO				(T series)
0035			RGCTO						(M series)

[Data type] Bit

RGCTO Cutting time constant in rigid tapping

0 : Uses a same parameter during cutting and extraction (Parameter Nos. 0613, 0692, and 0693 (M)/Nos. 0415 to 0418(T))

1 : Not use a same parameter during cutting and extraction

Parameter Nos. 0613, 0692, and 0693(M)/Nos.0415 to 0418(T): Time constant during cutting

Parameter Nos. 0400 to 0402(M)/Nos.0419 to 0422(T): Time constant during extraction

	#7	#6	#5	#4	#3	#2	#1	#0	
0041								RGTPE	(T series)
0040							RGTPE		(M series)

[Data type] Bit

RGTPE Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.)

0 : Canceled after rigid mode signal RGTPN <G123#1> is set to 0.

1 : Canceled before rigid mode signal RGTPN <G123#1> is set to 0.

	#7	#6	#5	#4	#3	#2	#1	#0	
0049								EVSF	(T series)

[Data type] Bit

EVSF 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.

	#7	#6	#5	#4	#3	#2	#1	#0	
0057		FXY							(M series)

[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 to be performed by using the basic axis (X, Y, or Z) perpendicular to a plane specified in the program, or any axis parallel to that basic axis, as the tapping axis.

	#7	#6	#5	#4	#3	#2	#1	#0	
0063		VSLPC		RGDOV					(T series)
				RGDOV	VSLPC				(M series)

[Data type] Bit

VSLPC Any gear ratio between spindle and position coder in rigid tapping
0 : Not used (The gear ratio is set in parameter Nos.0028#6, #7(M)/0003#6, #7(T).)
1 : Used (The gear ratio is set by parameter Nos. 0663 to 0668(M)/0427 to 0434(T).)

RGDOV Override during extraction in rigid tapping
0 : Invalidated
1 : Validated (The override value is set in parameter No. 0258(M)/0254(T).)

	#7	#6	#5	#4	#3	#2	#1	#0
0076					G84RGD			

[Data type] Bit

G84RGD 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. 0256(M)/0253(T)).
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.)

0214	Spindle backlash in rigid tapping (First gear)	(T series)
0215	Spindle backlash in rigid tapping (Second gear)	(T series)
0216	Spindle backlash in rigid tapping (Third gear)	(T series)
0217	Spindle backlash in rigid tapping (Fourth gear)	(T series)
0255	Spindle backlash in rigid tapping	(M series)

[Data type] Byte

[Unit of data] Detection unit

[Valid data range] 0 to 127

These parameters set the spindle backlash in rigid tapping.

0253	M code that specifies a rigid tapping mode	(T series)
0256	M code that specifies a rigid tapping mode	(M series)

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the M code that specifies the rigid tapping mode.

> **NOTE**
> If 0 is set, 29 (M29) is assumed.

0254	Override value during rigid tapping extraction	(T series)
0258	Override value during rigid tapping extraction	(M series)

[Data type] Byte

[Unit of data] 10 %

[Valid data range] 0 to 20

The parameter sets the override value during rigid tapping extraction.

> **NOTE**
> The override value is valid when bit 4 (RGDOV) of parameter No. 0063 is “1”.

	#7	#6	#5	#4	#3	#2	#1	#0	
0388					RGORT			PCTPH	(M series)

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

[Data type] Bit

PCTPH Rigid tapping
0 : Used as a high–speed peck tapping cycle
1 : Not used as a high–speed peck tapping cycle

RGORT 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.
> The spindle orientation is a zero return in the serial spindle servo mode.

	#7	#6	#5	#4	#3	#2	#1	#0	
0397					RGMFH				(T series)
0388						RGMFH			(M series)

[Data type] Bit

RGMFH Feed hold and single block in rigid tapping
0 : Invalidated
1 : Validated

		#7	#6	#5	#4	#3	#2	#1	#0	
0399				SIG						(T series)
0388				SIG						(M series)

[Data type] Bit

SIG When gears are changed for rigid tapping, the use of SIND <G124 and G125> is
0 : Not permitted.
1 : Permitted.

0400	In–position width of tapping axis in rigid tapping	(T series)
0618	In–position width of tapping axis in rigid tapping	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the in–position width of a tapping axis in rigid tapping.

0401	In–position width of spindle in rigid tapping	(T series)
0619	In–position width of spindle in rigid tapping	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the in–position width of a spindle in rigid tapping.

CAUTION
The broad in–position width deteriorates the screw precision.

0402	Limit value of position deviation during movement along the tapping axis for rigid tapping	(T series)
0620		(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 23767

This parameter sets the limit of positional deviation during movement along the tapping axis for rigid tapping.

CAUTION
The units are multiplied by 10 when a high–resolution detector is used.

0403	Return or clearance in peck tapping cycle	(M series)

[Data type] Word

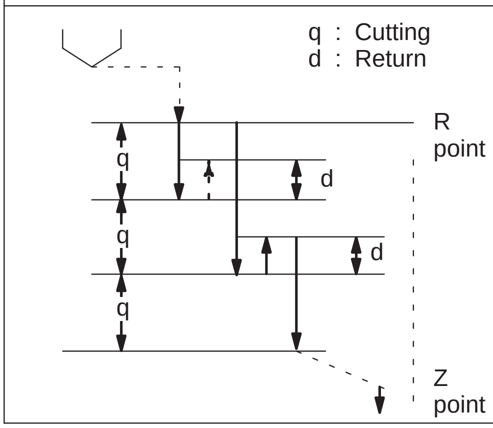
[Unit of data]

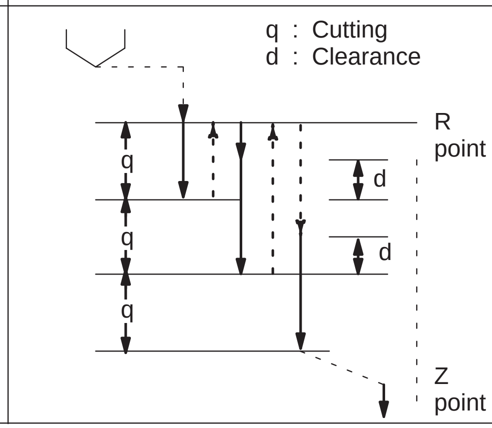
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the return or clearance in the peck tapping cycle.

When bit 0(PCTPH) in parameter No. 0388 is 0 (High–speed peck tapping cycle)	When bit 0(PCTPH) in parameter No. 0388 is 1 (Peck tapping cycle)





0403	Limit value of spindle positioning deviation during movement in rigid tapping	(T series)
0621	Limit value of spindle positioning deviation during movement in rigid tapping	(M series)

[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 rigid tapping.

$$\text{Limit value} = \frac{S \times 360 \times 100 \times 1.5}{60 \times G \times \alpha}$$

where

S: Maximum spindle speed in rigid tapping

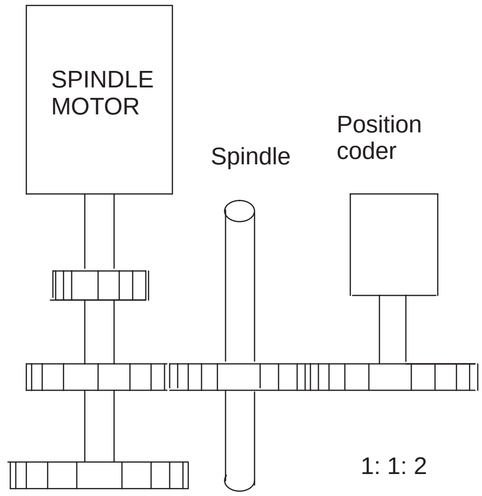
(Setting value of parameter Nos. 0617, 0694, and 0695 (M)/Nos. 0423 to 0426(T))

G: Loop gain of rigid tapping axis

(Setting value of parameter Nos. 0615, and 0669 to 0671 (M)/ Nos.0406 to 0410(T))

α: Detection unit

Example)



When the spindle motor, spindle, and position coder are connected as shown left, let the variables be as follows:

S = 3600
G = 3000
L = 360°
(One spindle rotation per spindle motor rotaion)
α = La / 4096
= 720°/4096
= 0.17578°
La= 720°
(One position coder rotation requires two spindle rotations =360° × 2)
4096=Detection pulse per positioncoder rotation

$$\text{Setting value} = \frac{3600 \times 360 \times 100 \times 1.5}{60 \times 3000 \times 0.17578}$$

= 6144

NOTE
The detection unit is a=La/2048 when the position coder built–in spindle motor uses a position coder of 512 pulses per revolution.

0404	Limit value of tapping axis positioning deviation during stop in rigid tapping	(T series)
0622	Limit value of tapping axis positioning deviation during stop in rigid tapping	(M series)

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a tapping axis positioning deviation during stop in rigid tapping.

0405	Limit value of spindle positioning deviation during stop in rigid tapping	(T series)
0623	Limit value of spindle positioning deviation during stop in rigid tapping	(M series)

[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 stop in rigid tapping.

0406	Position control loop gain of spindle and tapping axis in rigid tapping (Common in each gear)	(T series)
0615		(M series)

[Data type] Word

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

[Valid data range] 1 to 9999

This parameter sets the position control loop gain of a spindle and tapping axis in rigid tapping.

The loop gain setting significantly influences the screw precision. Perform a cutting test to adjust the loop gain and its multiplier to the optimum values.

> **NOTE**
> To change the loop gain for every gear, set this parameter value to “0” and set the loop gain for every gear to parameter Nos. 0669 to 0671 (M)/No.0407 to 0410 (T). If this parameter value is not “0”, the loop gain for every gear is invalidated. This parameter then becomes a loop gain that is used in common for all gears.

0407	Position control loop gain of spindle and tapping axis in rigid tapping (First gear)	(T series)
0669		(M series)
0408	Position control loop gain of spindle and tapping axis in rigid tapping (Second gear)	(T series)
0670		(M series)
0409	Position control loop gain of spindle and tapping axis in rigid tapping (Third gear)	(T series)
0671		(M series)
0410	Position control loop gain of spindle an tapping axis in rigid tapping (Fourth gear)	(T series)

[Data type] Word

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

[Valid data range] 1 to 9999

These parameters set the position control loop gain of a spindle and tapping axis for every gear in rigid tapping.

> **NOTE**
> To set the loop gain for every gear, set parameter No. 0615(M)/0406(T) to “0”.

0411	Spindle loop gain multiplier in the rigid tapping mode (for gear 1)	(T series)
0616	Spindle loop gain multiplier in the rigid tapping mode (for gear 1)	(M series)
0412	Spindle loop gain multiplier in the rigid tapping mode (for gear 2)	(T series)
0624	Spindle loop gain multiplier in the rigid tapping mode (for gear 2)	(M series)
0413	Spindle loop gain multiplier in the rigid tapping mode (for gear 3)	(T series)
0625	Spindle loop gain multiplier in the rigid tapping mode (for gear 3)	(M series)
0414	Spindle loop gain multioplier in the rigid tapping mode (for gear4)	(T series)

[Data type] Word

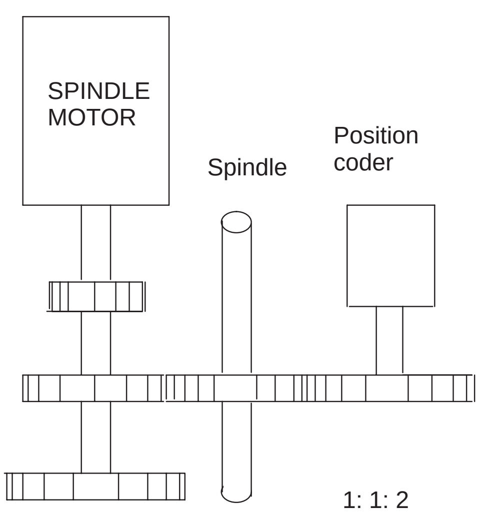
[Unit of data]

[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 and 100P gain by conducting the cutting test.

> **NOTE**
> These parameters are used for analog spindles.
> These parameters are used for analog spindles.
> Loop gain multiplier = $2048 \times \frac{E}{L} \times \alpha \times 1000$
> where;
> E: Voltage in the velocity command at 1000 rpm
> L: Rotation angle of the spindle per one rotation of the spindle motor
> α: Unit used for the detection

Example)



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 rpm 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°
(One rotation of the position coder corresponds to two rotations of the spindle = 360° × 2.)
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

Thus, Loop gain multiplier
$= 2048 \times 1.667/360 \times 0.17578 \times 1000 = 1667$

> **NOTE**
> When the position coder which is built in a spindle motor sends 512 pulses per rotation, the unit used for the detection, a, is La/2048.

0415 / 0692	Acceleration/deceleration time constant for every gear in rigid tapping (First gear)	(T series) / (M series)
0416 / 0693	Acceleration/deceleration time constant for every gear in rigid tapping (Second gear)	(T series) / (M series)
0417 / 0613	Acceleration/deceleration time constant for every gear in rigid tapping (Third gear)	(T series) / (M series)
0418	Acceleration/deceleration time constant for every gear in rigid tapping (Fourth gear)	(T series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

These parameters set the spindle and tapping axis's time constant for every gear during linear acceleration/deceleration in rigid tapping.

Set the time required until a spindle speed reaches the maximum spindle speed (parameter Nos. 0617, 0694, and 0695 (M)/Nos. 0423 to 0426(T)). The actual time constant is a proportional value between the maximum spindle speed and the specified spindle speed.

0419 / 0400	Acceleration/deceleration time constant during extraction in rigid tapping (First gear)	(T series) / (M series)
0420 / 0401	Acceleration/deceleration time constant during extraction in rigid tapping (Second gear)	(T series) / (M series)
0421 / 0402	Acceleration/deceleration time constant during extraction in rigid tapping (Third gear)	(T series) / (M series)
0422	Acceleration/deceleration time constant during extraction in rigid tapping (Fourth gear)	(T series)

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

These parameters set the linear acceleration/deceleration time constant of a spindle and tapping axis for every gear during extraction in rigid tapping.

> **NOTE**
> The time constant is valid when parameter RGTO (No.0035#5 (M)/0029#3(T)) is "1".

0423	Maximum spindle speed in rigid tapping (First gear)	(T series)
0694	Maximum spindle speed in rigid tapping (First gear)	(M series)
0424	Maximum spindle speed in rigid tapping (Second gear)	(T series)
0695	Maximum spindle speed in rigid tapping (Second gear)	(M series)
0425	Maximum spindle speed in rigid tapping (Third gear)	(T series)
0696	Maximum spindle speed in rigid tapping (Third gear)	(M series)
0426	Maximum spindle speed in rigid tapping (Fourth gear)	(T series)

[Data type] Two–word

[Unit of data] rpm

[Valid data range] Spindle and position coder gear ratio

1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

These parameters set the maximum spindle speed for every gear in rigid tapping.

> **NOTE**
> In a system having one–stage gear, set the same value as parameter No. 0694 to parameter No. 0617. In a system having two–stage gear, set the same value as parameter No. 0695 to parameter No. 0617. If it is not set as such, P/S alarm no. 200 will be informed.
> These are applicable to M series.

0427	Number of gear teeth on the spindle side in rigid tapping (First gear)	(T series)
0663	Number of gear teeth on the spindle side in rigid tapping (First gear)	(M series)
0428	Number of gear teeth on the spindle side in rigid tapping (Second gear)	(T series)
0664	Number of gear teeth on the spindle side in rigid tapping (Second gear)	(M series)
0429	Number of gear teeth on the spindle side in rigid tapping (Third gear)	(T series)
0665	Number of gear teeth on the spindle side in rigid tapping (Third gear)	(M series)
0430	Number of gear teeth on the spindle side in rigid tapping (Fourth gear)	(T series)

[Data type] Word

[Valid data range] 1 to 32767

These parameters set the number of gear teeth on the spindle side for every gear when any gear ratio is set in rigid tapping.

NOTE
This parameter is valid when parameter VALPC (No.0063#3(M)/0063#6(T)), is "1".
Set the same value to parameters when the spindle has a position coder.

0431	Number of gear teeth on the position coder side in rigid tapping (First gear)	(T series)
0666	Number of gear teeth on the position coder side in rigid tapping (First gear)	(M series)
0432	Number of gear teeth on the position coder side in rigid tapping (Second gear)	(T series)
0667	Number of gear teeth on the position coder side in rigid tapping (Second gear)	(M series)
0433	Number of gear teeth on the position coder side in rigid tapping (Third gear)	(T series)
0668	Number of gear teeth on the position coder side in rigid tapping (Third gear)	(M series)
0434	Number of gear teeth on the position coder side in rigid tapping (Fourth gear)	(T series)

[Data type] Word

[Valid data range] 1 to 32767

These parameters set the number of gear teeth on the position coder side for every gear when any gear ratio is set in rigid tapping.

NOTE
This parameter is valid when parameter VALPC (No.0063#3(M)/0063#6(T)), is "1".
Set the same value to above parameters when the spindle has a position coder.
A spindle motor incorporating the position coder uses a position coder with 2048 pulses per revolution. In this case, set the value that is two times as many as the actual number of gear teeth (because of conversion to 4096 pulses per revolution).

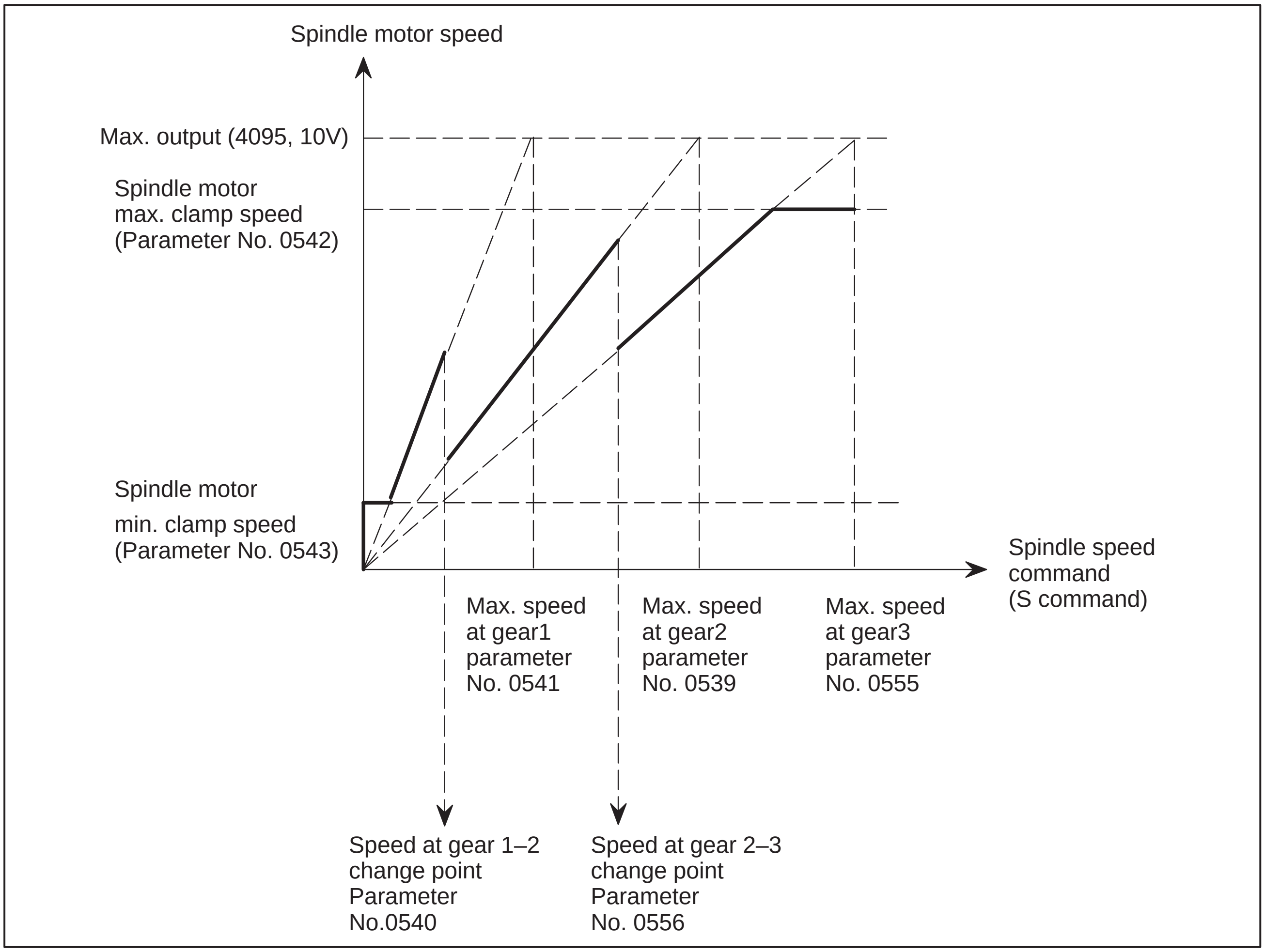
0540	Spindle speed when switching from gear 1 to gear 2 during tapping	(M series)
0556	Spindle speed when switching from gear 2 to gear 3 during tapping	(M series)

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (G84S,#6 of parameter 0012, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



9.11.9 Alarm and Message

Number	Message	Description
200	ILLEGAL S CODE COMMAND	In the 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. Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tapping, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tapping, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tapping, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the 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	Rigid tapping signal (DGNG061#0) is not 1 when G84 or G74 block for M series (G84 or G88 block for T series) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the signal is not turned on. Modify the program.
206	CAN NOT CHANGE PLANE (RIGID TAP) (M series)	Plane changeover was instructed in the rigid mode. Correct the program.
4n0	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) stops is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n1	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) moves is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n3	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–8 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.

9.11.10 Caution and Notes

Caution and notes on spindles

CAUTION

1 When using an analog spindle, set the spindle speed offset value parameter (No. 0577(M)/0539(T)) 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.0577(M)/0539(T). 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 parameter VLPC (No.0063#3(M)/0063#6(T)) 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 setting parameter (No.0627(M)/0436(T)).)

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.

Cautions 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 or third 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 or third spindle.

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 contorl 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.

CAUTION

(3) Although the system can change to rigid tapping mode directly from Cs contouring contorl mode, positions designated in Cs contouring control mode are not preserved if riged 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, nullify jogging and manual handling feed of the Cs contouring axis in the PMC during rigid tapping.

(5) When the multi–spindle control is also available and the rigid tapping is performed on the second or third spindle, the rigid tapping can be specified to the second or third spindle during the Cs contouring control of the first 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.0615, and 0669 to 0671(M)/Nos.0406 to 0410(T). 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 series (Serial Interface) DESCRIPTIONS (B–65042E) or FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E).

6544	Proportional gain of the velocity loop in servo mode (gear 1, gear 2)
6545	Proportional gain of the velocity loop in servo mode (gear 3, gear 4)

[Data type] Word

[Valid data range] 0 to 32767

Set a proportional gain for the velocity loop in a servo mode (such as rigid tapping mode).

6552	Integral gain of the velocity loop in the servo mode (gear 1, gear 2)
6553	Integral gain of the velocity loop in the servo mode(gear 3, gear 4)

[Data type] Word

[Valid data range] 0 to 32767

Set an integral gain of the velocity loop in a servo mode (such as rigid tapping mode).

6565	Position gain in the servo mode (HIGH) (CFPGH)
6566	Position gain in the servo mode (MEDIUM HIGH) (CFPGMH)
6567	Position gain in the servo mode (MEDIUM LOW) (CFPGML)
6568	Position gain in the servo mode (LOW) (CFPGL)

[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

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. 0615 and 0669 to 0671(M)/Nos.0406 to 0410(T) (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 (G229#3, #2 for the first spindle, and G233#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.

CTH1	CTH2	Gear selected	Parameter No. to be used		
0	0	HIGH	6565	6544	6552
0	1	MEDIUM HIGH	6566		
1	0	MEDIUM LOW	6567	6545	6553
1	1	LOW	6568		

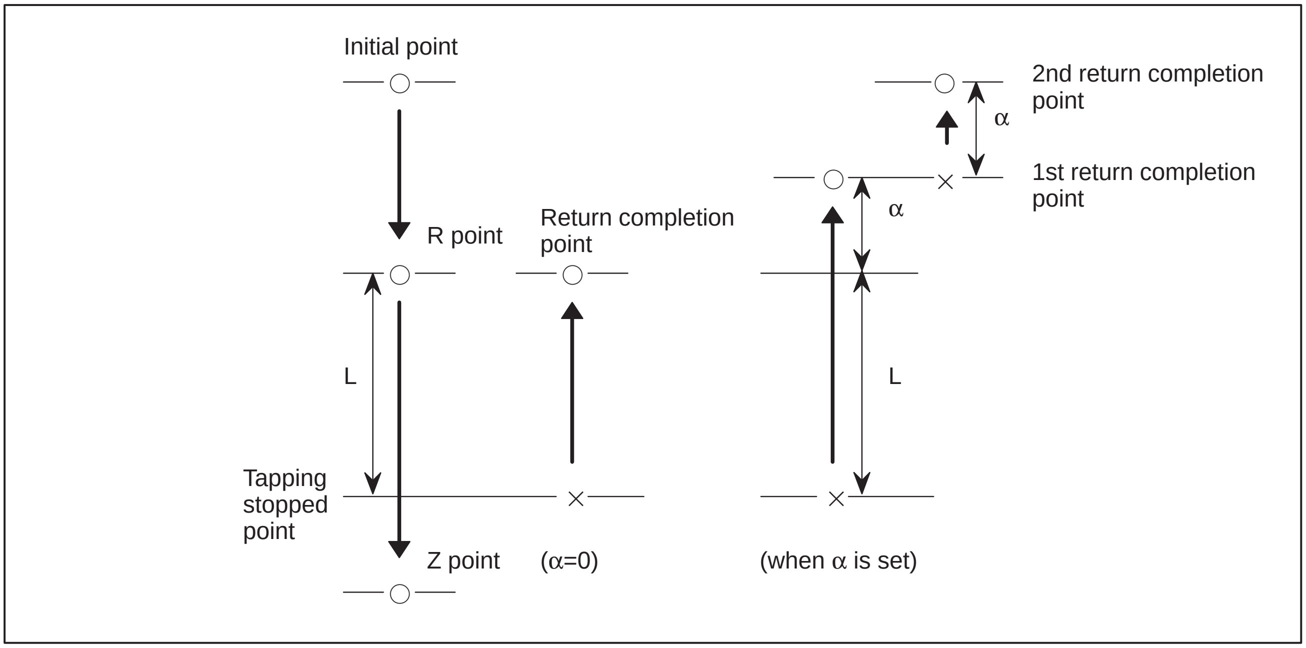
9.11.11
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.13.2	RIGID TAPPING
CONNECTION MANUAL (This manual)	9.3	SPINDLE SPEED CONTROL
CONNECTION MANUAL (This manual)	9.10	MULTI–SPINDLE
FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	Appendix 3.5	SERIAL SPINDLE RIGID TAPPING TECHNICAL MANUAL

9.12 RIGID TAPPING RETURN (M SERIES)

Overview

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 signal. This function automatically stores information relating to the tapping that was executed most recently. When a tap return signal is input, only the rigid tapping cycle return operation is executed, based on the stored information. The tap is pulled towards the R point. When a return value of α is set in a parameter, the amount by which the tap is pulled back can be increased by α.



Procedure

(1) Start
Reset the CNC, then select MDI mode. Turning the tapping return start signal (RTNT) on starts rigid tapping return.

(2) Completion
Upon the completion of rigid tapping return, the tapping return completion signal (RTPT) is turned on, causing the CNC to automatically enter the reset state. Turning the tapping return start signal off also turns the tapping return completion signal off.

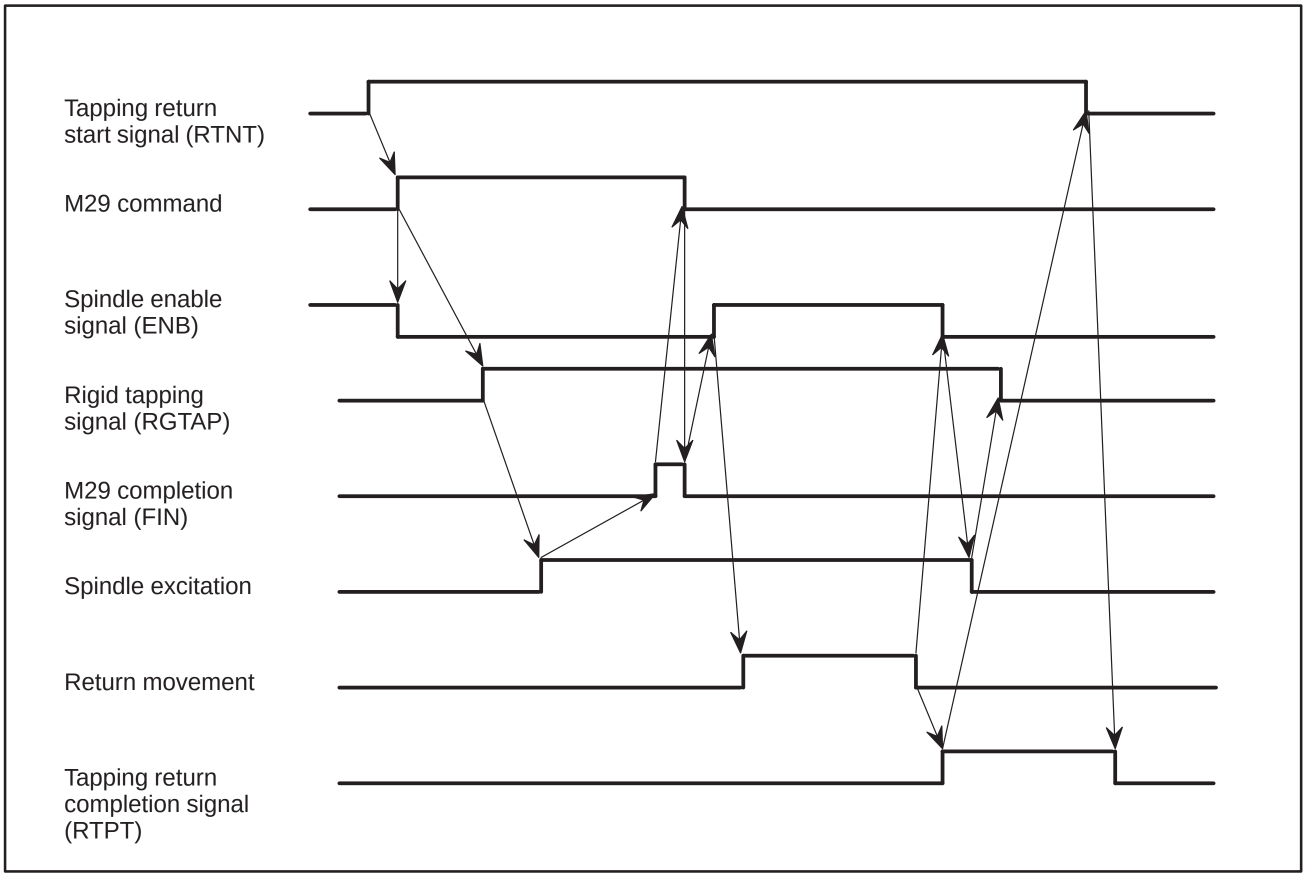
(3) Stop
During tapping return, turning the tapping return start signal off stops tapping return, thus placing the CNC in the reset state. To resume rigid tapping return, restart rigid tapping return. Rigid tapping return can also be stopped by means of a reset or feed hold.

(4) Resume

Once rigid tapping return has been stopped, it can be resumed by performing the same operation as that used for starting rigid tapping return. If rigid tapping return has been completed, however, the start operation does not restart tapping return. If return value α is set, however, the start operation performs tapping return using α only.

Time chart

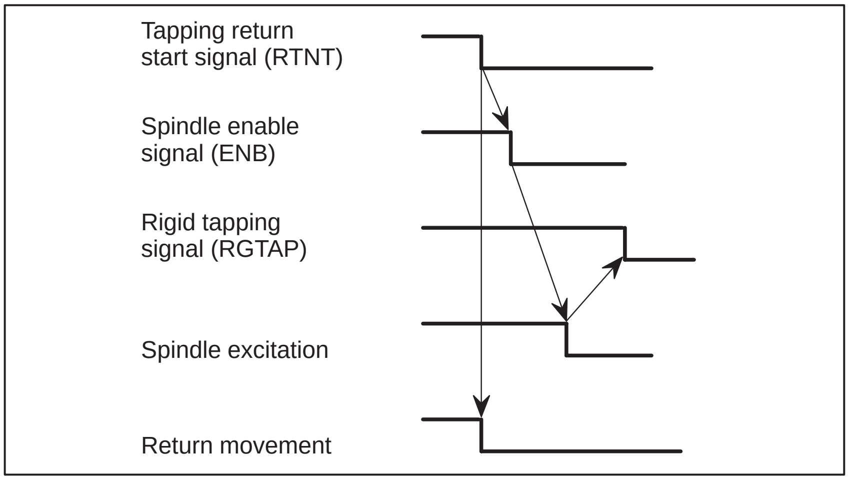
(1) Start and completion



In the reset state, turning the tapping return start signal (RTNT) on in MDI mode causes the rigid tapping M command to be output. For tapping return, specify neither gear switching nor orientation. Signal SF is also output if no S command has been specified after power–on.

Upon the completion of tapping return, the spindle enable signal (ENB) is turned off, in the same way as at the end of ordinary rigid tapping. Therefore, perform the same sequence as that used for canceling rigid tapping. Once tapping return has been completed, the tapping return completion signal (RTPT) is turned on and the CNC enters the reset state.

(2) Stopping tapping return



When tapping return is stopped, the spindle enable signal is turned off, in the same way as for ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. Also, the CNC enters the reset state automatically when tapping return is stopped.

Signal

Tapping return start signal RTNT <G123#6>

[Classification] Input signal

[Function] Draws out the tap, which has cut into the workpiece as a result of rigid tapping being stopped, to the R point.

[Operation] Setting this signal to 1 enables rigid tapping return. Rigid tapping return is not, however, started merely by setting this signal to 1. See the time chart for details of the actual rigid tapping return procedure. If this signal is set to 0 before the tapping return completion signal (RTPT) is set to 1, rigid tapping return is stopped. Rigid tapping return is terminated normally if this signal is set to 0 after the tapping return completion signal (RTPT) is set to 1. Tapping return can be resumed after being stopped.

Tapping return completion signal RTPT <F192#6>

[Classification] Output signal

[Function] Used to post notification of the completion of rigid tapping return.

[Output condition] This signal is set to 1 upon the completion of rigid tapping return. This signal is set to 0 when rigid tapping return has not been completed or when it has not been started.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G123		RTNT							(M series)
F192		RTPT							(M series)

Parameter

0378	Override for rigid tapping return

Sets an override value to be applied to rigid tapping return. No override is applied if 0 is set.

[Valid data range] 0 to 20

[Unit of data] 10%

> **NOTE**
> This parameter is enabled only when the parameter used to enable tool extraction (RGDOV: bit 4 of No. 0063) is set to 1.

0960	Return value α for rigid tapping return

Sets an extra return value for rigid tapping return. The tool will be pulled beyond the R point by α. If rigid tapping return has already been completed, the tool is pulled by α only.

[Valid data range] 0 to 99999999

[Unit of data] Input increments

Caution

> **CAUTION**
> An override can be applied to rigid tapping return, provided it is enabled with the corresponding parameter (bit 4 (RGDOV) of No. 0063 = 1).

Note

NOTE

1 Turning the tapping return start signal (RTNT) on starts tapping return only when the CNC is placed in both the reset state and MDI mode.
2 The machining data for tapping return is maintained until a rigid tapping command is subsequently specified, even if the power is turned off. Tapping return can, therefore, be specified even if the power has been turned off after rigid tapping.
3 Rigid tapping return 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 return was stored.
4 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.
5 Rigid tapping return 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 return may fail to draw the tapping tool completely out of the workpiece. In such a case, set return value α.
6 During tapping return, switching the mode to manual mode stops tapping return.
7 For tapping return, the CNC internally activates a return program. Tapping return may, therefore, cause some G codes or M/F/S codes to be overwritten (G80/G84/G74, G94/G95, G30).

9.13 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
0–TTC	First serial spindle at tool post 1	First serial spindle at tool post 2

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 0303 of the NC function.
  The positions of spindle phase synchronization can be specified in spindle parameter 6534 on each of tool post 1 and tool post2 with 0–TTC.

  When the two spindles are subject to spindle–phase synchronous–control (until the spindle–phase synchronous–control completion signal, FSPPH <F178#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 <F178#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 0576 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 6520).

(Example) Maximum speed of the spindle motor of tool post 1: 6000 rpm
Maximum speed of the spindle motor of tool post 2: 4500 rpm

In the example above, a maximum spindle speed of 6,000 rpm is specified for a spindle of tool post 1, although a spindle–speed command can specify up to 12 bits, 4096. If 6,000 rpm 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 rpm 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 0540 to 0543. 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. 0080 #6 (for the master spindle) and #7 (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 0577(M)/0539(T)) 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

Spindle synchronous control signal SPSYC<G146#2>

[Classification] Input signal

[Function] Selects spindle synchronous control mode for serial interface spindles.

[Operation] Setting this signal to 1 selects spindle synchronous control mode.
Setting this signal to 0 cancels spindle synchronous control mode.

Spindle phase synchronous control signal SPPHS<G146#3>

[Classification] Input signal

[Function] Selects spindle phase synchronous control mode for serial interface spindles. Input this signal after the system has entered the spindle speed synchronous state (FSPSY <<F178#2> = 1). Once the system has entered the spindle phase synchronous state (FSPPH <<F178#3> = 1), setting this signal to 0 does not cause the spindle phase to shift, because the CNC monitors the rising edge of this signal. Phase alignment is performed even if this signal is set to 1 in the spindle phase synchronous state.

[Operation] Setting this signal to 1 selects spindle phase synchronous control mode.
Setting this signal to 0 selects spindle phase synchronous control mode.

Spindle synchronous speed control completion signal FSPSY<G178#2>

[Classification] Output signal

[Function] Indicates that spindle synchronous control (rotational speed) has been completed for serial interface spindles.

[Output condition] This signal is set to 1 in the following case:

1. In spindle synchronous control mode, when the two spindles have reached the rotational speeds specified with the spindle synchronous speed command signal and the difference between the speeds of these spindles does not exceed the value specified with parameter No. 6533

This signal is set to 0 in the following cases:

1. In spindle synchronous control mode, when the rotation speeds of the two spindles are less than those specified with the spindle synchronous speed command signal
2. In spindle synchronous control mode, when the difference between the speeds of these spindles exceeds the value specified with parameter No. 6533
3. When spindle synchronous control mode is not selected

Spindle phase synchronous control completion signal FSPPH<G178#3>

[Classification] Output signal

[Function] Indicates that spindle phase synchronous control (phase control) has been completed for serial interface spindles.

[Output condition] This signal is set to 1 in the following case:

1. In spindle synchronous control mode, when phase alignment using the spindle phase synchronous control signal (SPPHS <<G146#3>) has been completed after the two spindles have reached the rotational speeds specified with the spindle synchronous speed command signal, and the difference between the numbers of error pulses for these spindles does not exceed the value specified with parameter No. 0303

This signal is set to 0 in the following cases:

1. In spindle synchronous control mode, when phase alignment between the two spindles has not been completed
2. In spindle synchronous control mode, when the difference between the numbers of error pulses for the two spindles exceeds the value specified with parameter No. 0303
3. When spindle synchronous control mode is not selected

Spindle synchronous control alarm signal SYCAL<G178#4>

[Classification] Output signal

[Function] Indicates that the difference between the numbers of error pulses for the two spindles exceeds the value specified with parameter No. 0576, in spindle synchronous control mode for serial interface spindles.

[Output condition] This signal is set to 1 in the following case:

1. In spindle synchronous control mode, when the difference between the numbers of error pulses for the two spindles exceeds the value specified with parameter No. 0576 after spindle synchronous control has been completed

This signal is set to 0 in the following cases:

1. In spindle synchronous control mode, when the difference between the numbers of error pulses for the two spindles does not exceed the value specified with parameter No. 0576
2. When spindle synchronous control mode is not selected

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G146					SPPHS	SPSYC		
F178				SYCAL	FSPPH	FSPSY		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0080	SP2ENG	SP1ENG						

[Data type] Bit

SP1ENG 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

SP2ENG 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

0303	Error pulse between two spindles when synchronizing phases in the serial spindle synchronization control mode

[Data type] Byte

[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 <F178#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.

0576	Allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode

[Data type] Word

[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 <F178#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 <F178#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 series (Serial interface) DESCRIPTIONS (B–65042E)	Appendix 3.3	Start–up procedure for spindle synchronization control
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.7	Spindle synchronization control

9.14 CONTROLLING THE SPINDLE SPEED RATIO FOR SERIAL INTERFACE SPINDLES (0–TTC)

General

For spindle synchronous control for serial interface spindles of the 0–TTC, this function can apply control such that the ratio between the spindle speeds on tool posts 1 and 2 becomes 1:n (n = 1, 2, 3, ...).

This function requires the specification of the spindle synchronous control option for serial interface spindles. See also Section 9.13, ”Spindle Synchronous Control” as well as this section. Setting signal SBRT to 1 in serial interface spindle synchronous mode causes the spindle on tool post 2 to rotate at the speed obtained by multiplying the speed of the spindle at tool post 1 by the magnification set in parameter SBRATIO.

Setting signal SBRT to 0 restores normal spindle synchronous mode.

If the speed of the spindle on tool post 2 exceeds the value set in parameter RTSMAX, signal RSMAX is set to 1 and the speed of the spindle on tool post 2 is clamped to the value set in parameter RTSMAX. At this time, the speed ratio between the two spindles is maintained.

To check the synchronization between the spindles on the two tool posts during spindle speed ratio control, the signals FSPPH and SYCAL used for spindle synchronous control for serial spindles can also be used.

Signal

Spindle synchronous polygon code signal SBRT <G146#6>

[Function] This signal places the CNC in spindle speed ratio control mode.

Serial spindle synchronization polygon signal RSMAX <F189#7>

[Function] This signal is used to post notification of the speed of the spindle on tool post 2 being clamped to RTSMAX during spindle speed ratio control.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G146		SBRT						

	#7	#6	#5	#4	#3	#2	#1	#0
F189	RSMAX							

Parameter

0249	SBRAITO

[Data type] Byte

This parameter sets the magnification to be applied to the speed of the spindle on tool post 2 during spindle speed ratio control.
(Valid data range: 1 to 9)

0877	RTSMAX

[Data type] Two–word

This parameter sets the maximum value for the speed of the spindle on tool post 2 during spindle speed ratio control.
(Valid data range: 0 to 19999 rpm)

9.15 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 mechanically stopped by using stoppers.
- 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 stopper, rotate the spindle at a constant low speed and drive a stopper or pin into the spindle. The spindle can be rotated at a 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 <G110, G111>

SHB00 to SHB11 for the second spindle <G112, G113>

[Classification] Input signal

[Function] This command is used for specifying a stop position with an absolute position within one rotation in the following equation:

$$= \frac{360}{4096} \times \sum_{i=0}^{\#n} (2_i \, Pi)$$

where
Pi = 0 when SHAi = 0
Pi = 1 when SHAi = 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G110	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G111					SHA11	SHA10	SHA09	SHA08
G112	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G113					SHB11	SHB10	SHB09	SHB08

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0080					MORCM2	MORCM1		

[Data type] Bit

MORCM1: Whether the stop–position external–setting type orientation function is used by the first spindle motor
0 : Not used
1 : Used

MORCM2 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 (Nos. 6531 and 6071) are invalid.

Note

NOTE
Spindle orientation with the spindle positioning function differs from that described in this section. For details, see Section 9.8, “Spindle Positioning.”

Reference item

FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	VIII.	POSITION CODER METHOD SPINDLE ORIENTATION
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.1 12.1	POSITION CODER METHOD SPINDLE ORIENTATION MAGNETIC SENSOR METHOD SPINDLE ORIENTATION

9.16 SPINDLE OUTPUT SWITCHING

General

Spindle output switching switches between the two 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, see also 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.

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 shall not rotate with no drive 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 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 <F152 #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. 0541 = Maximum spindle speed – 1, value of No. 0539 = Maximum spindle speed)

  When parameter No. 0541 is equal to parameter No. 0539, the CNC judges that one gear is used, and does not output the GR2O signal.

  The parameters related to gear switching points, G845, bit 6 of parameter No. 0012, and LQCM, bit 6 of parameter No. 0036, parameter Nos. 0540 and 0585 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. 0541, 0589, and 0555.)

  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 <G118 #2, #3>, are used (for lathes or machining centers in which constant surface speed control is provided).

  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. 0540 and 0541 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 9.11	SPINDLE CONTROL RIGID TAPPING
FANUC AC SPINDLE MOTOR series (Serial Interface) DESCRIPTIONS (B–65042E)	XIII	OUTPUT SWITCHING CONTROL
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.9	OUTPUT SWITCHING CONTROL

9.17 POSITION CODER FEEDBACK CONTROL FUNCTION (0–TC)

General

This function is used to prevent inaccurate threading due to varying feedback pulses while the spindle is rotating at low speed in feed–per–rotation mode. When the direction of any pulse returned in a certain step is opposite to the direction of rotation as predetermined by the PMC, the number of those pulses is stored. The difference between the number of stored pulses and the feedback pulses returned in the next step is used for determining the feedrate (Fig. 9.17).

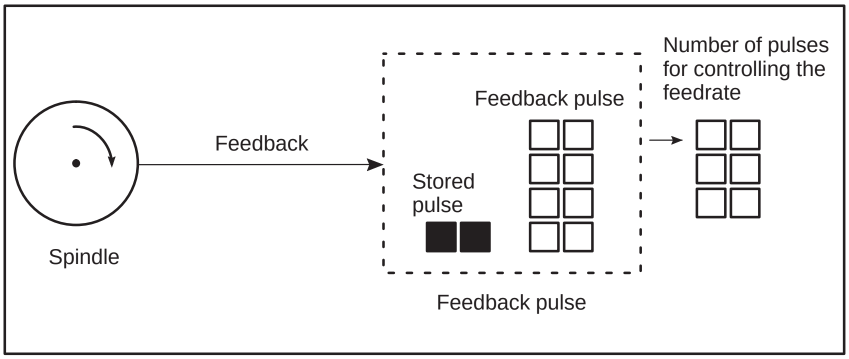


Fig.9.17

1 This function is enabled when bit 7 of parameter 0074 is set accordingly.
2 This function is turned on or off by a signal (G105#7).
3 When the direction of any spindle feedback pulse returned in a certain step is opposite to the direction of rotation specified by the PMC, the feedback pulses in that step are stored. The system internally assumes the feedback of that step to be 0. (Signal G105#6)
4 If the feedback pulses of the next step have a predetermined direction of rotation, the difference between the number of these pulses and those stored in 3, above, is used as the number of feedback pulses for this step.

Operating procedure

1 To enable the function, set the PLCREV bit (bit 7 of parameter 0074) to 1.
2 Set a direction of rotation of the spindle for this function from the PMC by using the KILPLUS signal (G105#6). Set this signal before setting the function ON/OFF signal PLCRVO (G105#7) to 1.
If the direction of a returned feedback pulse is opposite to the direction of rotation set by KILPLUS, the feedback pulse is disabled.
3 When the function ON/OFF signal PLCRVO (G105#7) is set to 1, this function is activated.

Signal

Position coder feedback control signal PLCRVON<G105#7>

[Function] This signal disables spindle feedback pulses of a given direction. (The signal is valid when the PLCREV bit (bit 7 of parameter 0074) is set to 1.)

[Operation] A feedback pulse having a direction specified by KILPLUS is disabled.

Position coder feedback direction selection signal KILPLUS<G105#6>

[Function] When KILPLUS is set to 1, positive direction feedback pulses are disabled. When KILPLUS is set to 0, negative direction feedback pulses are disabled. (This signal is valid when both the PLCREV bit (bit 7 of parameter 0074) and the PLCRVON signal (G105#7) are set to 1.)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G105	PLCRVON	KILPLUS						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0074	PLCREV							

PLCREV
1 : A spindle feedback pulse having a direction opposite to that specified by the PMC is invalid.
0 : All spindle feedback pulses are valid, irrespective of their direction.

Caution

CAUTION
Set the rotation direction selection signal (G105#6) before setting the signal of G105#7.

NOTE
1 When the number of feedback pulses having a direction opposite to that specified by the PMC exceeds 3275, the number of stored pulses is reset to 0. Then, the remaining pulses are counted.
2 When there is a feedback pulse having a direction opposite to that specified by the PMC, SACT also goes to 0.

9.18 SERIAL SPINDLE SIMPLE SYNCHRONOUS CONTROL

General

In serial spindle simple synchronous control mode, the second spindle is controlled based on the spindle control commands for the first spindle. This function enables the second spindle to use Cs contour axis control, rigid tapping, and Cs axis control (all of which are optional functions), which were previously supported only for the first spindle. The first and second spindles cannot, however, be controlled independently. The second spindle is controlled using the same commands as those for the first spindle. In simple synchronous control mode, synchronization between the first and second spindles may not be guaranteed, for example, in spindle control mode or during reference position return in Cs contour axis control mode.

This function requires a software option for serial interfaces. The PMC is required for control. As hardware, two spindle systems, both of which support the connection of two spindles, are required.

Simple synchronous control mode is selected by inputting the ESRSYC signal from the PMC.

In spindle mode, the second spindle rotates based on the same commands as those specified for the first spindle. When the first and second spindle units have the same configuration (such as the maximum motor speed and gear ratio between the motor and spindle), the first and second spindles rotate at the same speed. (Synchronization between the first and second spindles is not guaranteed, however. This function merely assures that the motor speeds are the same.)

When this function is used in combination with the Cs contour axis control option, contouring control can also be applied to the second spindle. This means that the second spindle can perform the same operation as that of the first spindle, not that the two spindles can be operated independently. Only a conventional position display is provided (which displays only the data for the first spindle). Reference position return is also performed in the conventional way. Check for return to the reference position is, however, performed for both axes, so that the reference position return completion signal (T series: ZP3, M series: ZP4) is not turned on until reference position return has been completed for both axes. Spindle mode and contouring mode can be switched while simple synchronous control is applied.

When this function is used in combination with the rigid tapping option, rigid tapping can also be performed for the second spindle. This means that the second spindle can perform the same operation as that being performed by the first spindle, not that the two spindles can be operated independently. Data is displayed in the conventional way, including the spindle positional deviation, specified with parameter No. 0627.

When this function is used in combination with the Cs axis control option, Cs axis control can also be applied to the second spindle. This means that the second spindle can perform the same operation as that being performed by the first spindle, not that the two spindles can be operated independently. Only the conventional position display is provided, in the same way as for Cs contour axis control. Orientation is also performed conventionally. The completion of orientation is, however, checked for both spindles. The spindle indexing sequence also progresses after checking the completion of indexing for both spindles.

In simple synchronous control mode, commands can be specified in the same way as ordinary commands for the main spindle, regardless of whether spindle mode, Cs contour axis control mode, rigid tapping mode, or Cs axis control mode is selected. Simple synchronous control does not require modification of any conventional PMC signal address or sequence. Simple synchronous control can be enabled or disabled merely by turning the ESRSYC signal on or off.

Even during simple synchronous control, a function that is directly specified from the PMC (such as spindle orientation) controls the two spindles independently. Specifically, PMC control for the following areas is always performed independently for the first and second spindles regardless of whether simple synchronous control mode is selected: DGN DI G229 to G232 and DO F281 to F284 for the first spindle and DGN DI G233 to G236 and DO F285 to F288 for the second spindle.

While simple synchronous control is applied, synchronization between the two spindles is not guaranteed in spindle mode. In Cs contour axis control mode, rigid tapping mode, or Cs axis control mode, however, synchronization between the two spindles can be guaranteed by matching the position gains for the two spindles, except in the following cases: During reference position return in Cs contour axis control mode or orientation in Cs axis control mode, or when commands for the spindles are directly specified from the PMC (such as spindle orientation).

While simple synchronous control is applied, errors are monitored for the control applied to the first spindle and that applied to the second spindle. This includes in–position check and the monitoring of excessive errors during movement, or upon the stop, of the tool. The same parameters as those for the first spindle are, however, also used for the second spindle, and alarms issued upon the occurrence of an error are displayed without discriminating between the first and second spindles.

While simple synchronous control is applied, the positional deviation and other data for the first spindle are displayed in the conventional positions. For the second spindle, only the error (positional deviation) is displayed at DGN No. 755.

PMC signal SYCAL is provided to monitor mutual errors for the two spindles while simple synchronous control is applied. The error between the two spindles is constantly monitored. The SYCAL signal is turned on if the error exceeds the value set in parameter No. 576 (set using the absolute value of the number of error pulses; detection units vary depending on whether Cs contour axis control, rigid tapping, or Cs axis control mode is selected). The SYCAL signal is turned off once the error falls below the set value. This signal is invalid in spindle mode.

Supplementary description

If this function is used in combination with the serial spindle synchronous control option, the ESRSYC signal can be turned on only while synchronous control is not applied. The synchronous control signal (SRSYC) also cannot be turned on while simple synchronous control is applied. If the synchronous control signal and simple synchronous control signal are turned on simultaneously, the mode selected first is maintained and the CNC issues a PS alarm (No. 194). This alarm remains active until the most–recently input signal (SRSYC or ESRSYC) is turned off and the CNC is reset.

In other than synchronous control mode, no restriction is imposed on the time at which the ESRSYC signal is turned on. When the first spindle is used in spindle mode or Cs contour axis control mode, turning on the ESRSYC signal immediately initializes the second spindle to the same mode.

In rigid tapping mode or Cs axis control mode, however, turning on the ESRSYC signal does not place the spindles in simple synchronous control mode, even though no alarm is issued. Simple synchronous control mode is set upon the initialization of the first spindle to spindle or Cs contour axis control mode after rigid tapping or Cs axis control mode has been canceled. Once simple synchronous control mode has been set, a rigid tapping or Cs axis control command causes the second spindle to be initialized to rigid tapping or Cs axis control mode, in the same way as for the first spindle.

No restriction is imposed on the time at which the ESRSYC signal is turned off. Turning off the ESRSYC signal immediately initializes the second spindle to spindle mode, regardless of the previous mode. The first spindle maintains the previous mode.

The following table summarizes the state transition of the first and second spindles according to the ESRSYC and synchronous control (SRSYC) signals:

Change of simple synchronous control signal	**ESRSYC 0→1**			**ESRSYC 1→0**		
State transition of first spindle	SP→SP (main- tained)	CT→CT (main- tained) *1	SV→SV (main- tained) *2	SP→SP (main- tained)	CT→CT (main- tained)	SV→SV (main- tained)
State transition of second spindle	SP→SP *3	SP→CT *1’	SP→SP *2’	SP→SP *4	CT→SP *4	SV→SP *4
When used in combination with the synchronous control option	SRSYC = 1 (during synchro- nous control) →PS194 alarm is issued. SRSYC = 0 (not during syn- chronous control) →The state changes as listed above.			SRSYC = 1 (during synchro- nous control) →PS194 alarm is released by resetting the CNC. SRSYC = 0 (not during syn- chronous control) →The state changes as listed above.		

In the above table:
SP indicates spindle mode.
CT indicates Cs contour axis control mode (contouring mode).
SV indicates rigid tapping mode (M series) or Cs axis control mode (T series).

*1, *1’: The second spindle is initialized to contouring mode. At this time, however, the position of the second spindle is undefined, so that reference position return must be performed for both the first and second spindles. A conventional reference position return command can be used. (Refer to the specifications of reference position return for serial spindle Cs contour axis control.)

*2, *2’: When the first spindle is used in rigid tapping mode or Cs axis control mode, the input of the ESRSYC signal is internally masked in the CNC, so that simple synchronous control mode is not set. Once the first spindle has been initialized to spindle or contouring mode after the cancellation of rigid tapping or Cs axis control mode, the second spindle enters simple s ynchronous control mode, being initialized in the same way as the first spindle. Once simple synchronous control mode has been set, initialization to rigid tapping or Cs axis control mode, command execution for that mode, and release from the mode are performed for both the first and second spindles.

*3 The mode is not changed, but the second spindle is initialized to spindle mode. After initialization to spindle mode, the second spindle rotates at the speed specified for the second spindle, so that the speed may vary when the signal state changes.

*4 The second spindle is initialized to spindle mode, regardless of the previous mode. The first spindle maintains the previous mode.

Parking function

This function stops the first and second spindles in simple synchronous control mode, regardless of whether spindle, Cs axis, Cs contour axis, or rigid tapping mode is selected.

In spindle mode, this function has the same effect as the S0 command. In other modes, this function reduces the pulse distribution to 0. This function also disables spindle orientation for the Cs axis and reference position return for the Cs contour axis.

Because this function stops spindle rotation only, it can be used to continue machining by stopping only that spindle which becomes unnecessary during two–spindle control, or to apply parking to the first spindle so that Cs contour control, Cs axis control, or rigid tapping is apparently performed with the second spindle.

CAUTION

1 While the tool is moving in a mode which involves a position loop, such as Cs contour control, Cs axis control, and rigid tapping, applying parking causes the actual machine position to shift from the coordinate system. This does not cause an excessive error alarm because it is caused by parking. Take particular care, therefore, when using the parking signal.

2 The parking signal takes effect immediately in simple synchronous control mode. If the parking signal is input during reference position return in Cs contour control mode or spindle orientation in Cs axis control mode, however, operation is continued until reference position return or spindle orientation has been completed. Only then is parking applied.

3 While both the first and second spindles are in the parking state, do not specify reference position return in Cs contour control mode or spindle orientation in Cs axis control mode. The specified positioning operation cannot be completed while both axes are placed in the parking state.

4 Even when the first spindle has completed reference position return and reference position return completion signal ZP3 (ZP4 for the M series) has been turned on, turning on the simple synchronous control signal causes ZP3 (ZP4 for the M series) to be turned off (because the second spindle is not positioned to the reference position). The parking signal input (on/off) is not, however, related to the state of the reference position return completion signal. For example, performing reference position return with the first axis placed in the parking state does not result in the first spindle being manipulated, but ZP3 (ZP4 for the M series) is turned on upon the completion of reference position return for the second spindle.

NOTE

If parking is applied to a spindle in a mode which involves a position loop, such as Cs contour control, Cs axis control, or rigid tapping, the spindle is stopped at the current position, and an excessive error alarm is issued if it is moved from that position by any external force. (In simple synchronous control mode, errors are constantly monitored for both spindles, regardless of the parking state.)

Signal

Spindle simple synchronous control signal ESRSYC <G104#4>

[Classification] Input signal

[Function] This signal selects serial spindle simple synchronous control mode.

Parking signals PKESS2, PKESS1 <G138#7, #6>

[Classification] Input signal

[Function] These signals apply parking for the corresponding spindle while serial spindle simple synchronous control is applied.

[Operation] The spindle specified with PKESS1 or PKESS2 is placed in the stop state (parking state).

Synchronization error detection signal SYCAL <F178#4>

[Function] This signal is used to post notification of the synchronization error between the two spindles exceeding the set value (parameter No. 576).

[Output condition] This signal is output when the synchronization error between the two spindles exceeds the value set in parameter No. 576.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G104				ESRSYC				
G138	PKESS2	PKESS1						
F178				SYCAL				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0071				SRL2SP				

[Data type] Bit

SRL2SP 1: Two serial spindles are connected.
0 : One serial spindle is connected.

> **NOTE**
> If the option for serial spindle synchronous control is used, this parameter is automatically set upon power–on. Once this parameter has been set, the following operations become possible:
> 1) Checking the connection of the second serial spindle amplifier and communicating with it.
> 2) Controlling the second spindle in asynchronous control mode (SIND2).
>
> Serial spindle synchronous control cannot, however, be used simultaneously with the external key input function, because the addresses used for SIND2 are shared with those for external key input.

When using the simple synchronous control function, this parameter must be set to 1 because two serial spindles are required.

0576	SYCEALM

[Data type] Word

SYCEALM Allowable number of error pulses between the two spindles in serial spindle synchronous control mode, or allowable number of error pulses between the two spindles in serial spindle simple synchronous control mode
Valid data range: 0 to 32767

> **NOTE**
> This parameter is used to output the signal (SYCAL) used to detect the phase error between the two spindles in serial spindle synchronous control mode. The signal is turned on if the phase error detected between the two spindles exceeds the value set in this parameter.
> When using this parameter to detect error pulses in simple synchronous control mode, particular care regarding the mode in which the spindle is being used is necessary. (This parameter is invalid in spindle mode. In Cs contour axis control mode, rigid tapping mode, or Cs axis control mode, this parameter is valid but the detection units corresponding to one pulse vary with the mode.)

Display of positional deviation in spindle synchronous control mode or simple synchronous control mode (DGNOS 0754 to 0756)

0754	SRLERRS1

SRLERRS1 Motion error for the main spindle in synchronous control mode or simple synchronous control mode

0755	SRLERRS2

SRLERRS2 Motion error for the second spindle in synchronous control mode or simple synchronous control mode

0756	SRLSYCERR

SRLSYCERR Absolute value of the synchronization error between the main spindle and second spindle in synchronous control mode or simple synchronous control mode

The above three DGN values are displayed in pulse units. In synchronous control mode, one pulse is equivalent to an error of 360/4096 degrees.

In simple synchronous control mode, the detection units corresponding to one pulse vary depending on whether Cs contour axis control, rigid tapping, or Cs axis control mode is selected, as well as the detector type and the method used to mount it.

Alarm and message

P/S alarm

Alarm No.	Description
194	Contouring, Cs axis control, or rigid tapping mode, or a simple synchronous control command, has been specified in serial spindle synchronous control mode. (Specify the mode or command after canceling synchronous control mode.) Alternatively, a synchronous control command has been specified in simple synchronous control mode. (Specify the command after canceling simple synchronous control mode.)

System alarm

Alarm No.	Description
946	In serial spindle control, the connection of the second spindle has been specified but the CNC cannot communicate with the second serial spindle amplifier. (Check that the second amplifier is connected and has been started.)

Note

NOTE

1 In simple synchronous control mode, the same rotation or move commands are issued to both the first and second spindles. If the direction of rotation specified for a spindle must be reversed for some reason, the PMC (SFR and SRV signals) or serial spindle parameters can be used, depending on the selected mode.
2 In simple synchronous control mode, both the first and second spindles are monitored. For example, if a simple synchronous control command is specified in Cs contour axis control mode when the second spindle is not excited, a VRADY OFF alarm may be issued. Therefore, also pay careful attention to the PMC used to control the second spindle.
3 SYCAL is not a latched signal; it is turned on and off while constantly monitoring whether the error between the two spindles falls within the allowable range, as specified with a parameter.

10 TOOL FUNCTIONS

10.1 TOOL FUNCTION

General

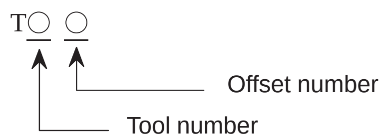
(M series)

Selection of tools can be done by commanding tool numbers with up to an 4-digit numeral after address T.

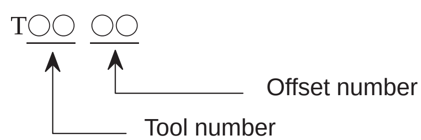
(T series)

Selection of tools and offset amounts can be done by commanding tool numbers and offset numbers with up to an 4-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 T2D (No.0014#0)=1)



When the last two digits are used to specify the offset number:
(Parameter T2D (No.0014#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.

Signal

See Section 8.1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0013							GOFU2		(T series)

[Data type] Bit

GOFU2 Geometry offset number of tool offset (When the option of tool geometry/wear compensation is selected, it is effective.)
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	
0014									T2D	(T series)

[Data type] Bit

T2D Offset number of tool offset (Wear offset number when option of tool geometry/wear compensation is selected)
0 : Specified using the lower two digits of a T code
1 : Specified using the lower one digit of a T code

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 Tcode was out of range.

Caution

CAUTION
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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.10.1	TOOL SELECTION FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.10.1	TOOL SELECTION FUNCTION
CONNECTION MANUAL (This manual)	8	AUXILIARY FUNCTION

10.2 TOOL COMPENSATION VALUE/ NUMBER OF TOOL COMPENSATION/ 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 unified to the tool compensation.

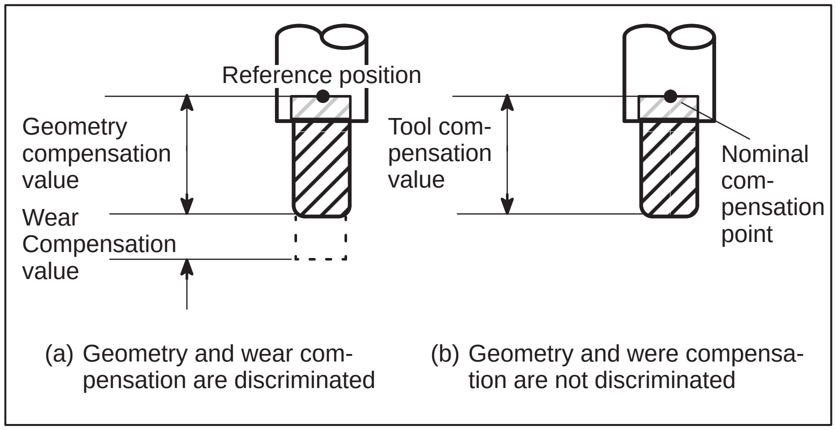


Fig. 10.2(a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the CRT/MDI panel 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

- **Number of tool compensation**

The memory can hold 32, 64, 99, or 200 sets of tool compensation values.

- **Tool compensation memory**

One of the tool compensation memory A/B/C can be selected according to offset amount.

- Tool compensation memory A

  There is no difference between geometry compensation memory and wear compensation memory in this tool compensation memory A. Therefore, amount of geometry offset and wear offset together is set as the offset memory. There is also no differences between cutter compensation (D code) and tool length compensation (H code).

- Tool compensation memory B

  Memory for geometry compensation and wear compensation is prepared separately in tool compensation memory B. Geometry compensation and wear compensation can thus be set separately. There is no difference between cutter compensation (D code) and tool length compensation (H code).

- Tool compensation memory C

  Memory for geometry compensation as well as wear compensation is prepared separately 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).
  No separate memory will be added for each compensation type. Instead, the existing memory is dividend into two, the first for D code and the second for H code. So, the quantity of compensation values of each type is halved, as listed below.

The above description is summarized as follows:

Tool compensation memory	Compensation amount
A	Tool compensation amount (Geometry compensation value + Wear compensation value)
B	Geometry compensation value
	Wear compensation value
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)).

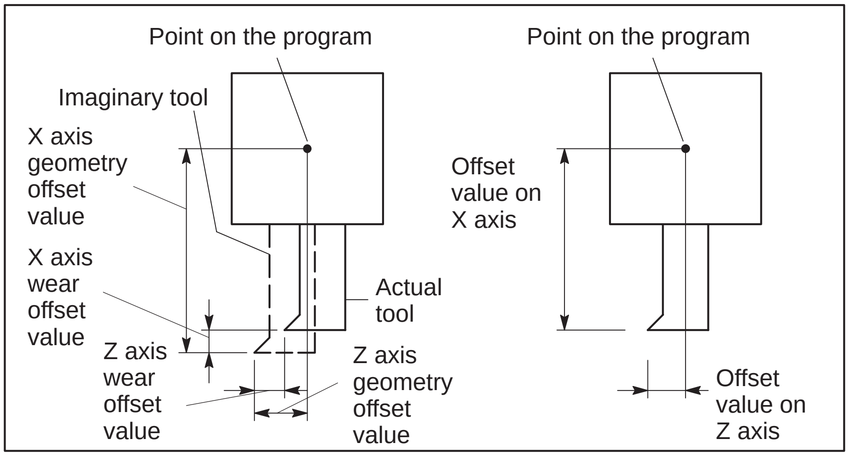


Fig. 10.2 (b) Difference the tool geometry offset from tool wear offset

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 CRT/MDI panel 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 16 or 32 sets of tool compensation values.

- **Tool compensation memory**

There are two types of tool offset amount memory, which can be selected according to offset amount.

- Tool geometry/wear compensation option not specified

  There is no difference between geometry offset memory and wear offset memory. Therefore, amount of geometry offset and wear offset together is set as the offset memory.

- Tool geometry/wear compensation option specified

  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 follws:

Tool compensation memory	Compensation amount
Without geometry/ wear compensation	Tool compensation amount (Geometry compensation value + Wear compensation value)
With geometry/wear compensation	Geometry compensation
	Wear compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0001				ORC					(T series)

[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	
0013							GOFU2		(T series)

[Data type] Bit

GOFU2 Geometry offset number of tool offset (When the option of tool geometry/ wear compensation is selected, it is effective.)
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	
0015							NWCH		(T series)

[Data type] Bit

NWCH 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	
0075					WNPT				(T series)

[Data type] Bit

WNPT Imaginary tool tip direction used for tool nose radius compensation, when the geometry/wear compensation option is equipped, is the direction specified by:
0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
0078							NOINOG	NOINOW

[Data type] Bit

NOINOW Setting the tool wear compensation value by MDI key input is:
0 : Not disabled
1 : Disabled

NOINOG Setting the tool geometry compensation value by MDI key input is:
0 : Not disabled
1 : Disabled

0728	Maximum value of incremental input for tool wear compensation	(T series)

[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 by G10	P/S 32 offset value is out of range by G10

0729	Maximum value of tool wear compensation	(T series)

[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 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.

Note

NOTE
In the 0–TTC, the number of specified tool compensation values equals the number of tool compensations for each tool post.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II 14.7	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II 14.4	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

10.3 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 <F188#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 <G139#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 external device after replacing the worn–out tools that are displayed on the CRT. The data can also be cleared from the MDI.

[Output condition] 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 <F192#0> (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 <G140#4> (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 <G140#0>

[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 external device then turn the tool skip signal TLSKP to “1”. The next T-code command will pass over the first tool in the group for which the skip was designated, and select the second 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. 0039#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 <F188#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.

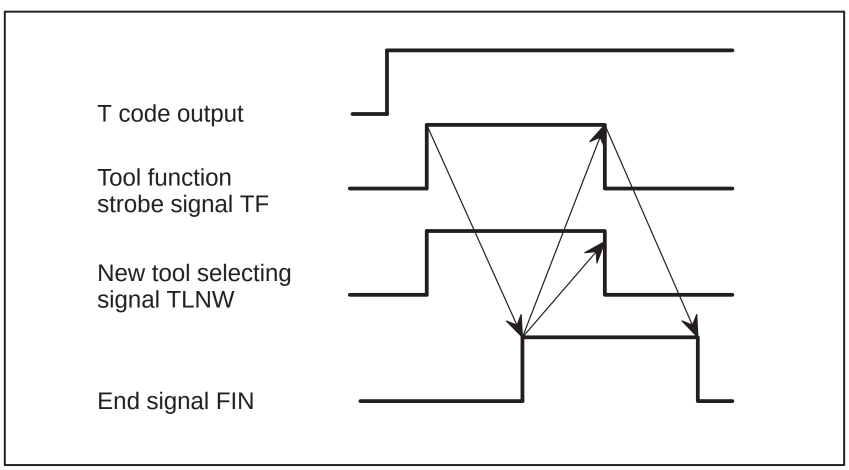
The new tool select signal is issued at the same timing as TF (tool function strobe signal).

[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”.



Remaining tool life expired signal TLCHB <F192#2>

[Classification] Output signal

[Function] Reports that the remaining tool life has reached the set value.

[Operation] The signal is output when the counter value becomes equal to the remaining tool life that is specified in parameter No. 0489 as the time allowed until a new tool must be selected.

All tools' life expired signal TLCHE<F192#1>

[Classification] Output signal

[Function] Reports that a tool group in which the operating life of all tools had expired was selected.

[Operation] The signal is output when a command is issued to a tool group in which the operating life of all tools has expired.

Tool group number select signal TL01 to TL64 <G139#0 to #6>

[Classification] Input signal

[Function] When the TLRST and TLSKP signals are both input, the tool group number must be given in advance, using the tool group number selection signals TL01 to TL64.
Command the following value in binary form:
Tool group number to be specified –1

[Operation] A specified tool group is selected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01	
G140				TLRSTI				TLSKP	(M series)
F192						TLCHB	TLCHE	TLCHI	
F188							TLNW	TLCH	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0039			IGSK	GRST	TLSK	TCTM	TST2	GST1	(T series)
	M6TCD	IGIN	IGSK	GRST	TLSK	TCTM	GST2	GST1	(M series)

[Data type] Bit

GST1, GST2 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.

Lower side ranges in M series column are for the tool life management of 512 pairs.

GST2	GST1	M series		T series	
		Group count	Tool count	Group count	Tool count
0	0	1 to 16 1 to 64	1 to 16 1 to 32	1 to 16	1 to 16
0	1	1 to 32 1 to 128	1 to 8 1 to 16	1 to 32	1 to 8
1	0	1 to 64 1 to 256	1 to 4 1 to 8	1 to 64	1 to 4
1	1	1 to 128 1 to 512	1 to 2 1 to 4	1 to 16	1 to 16

TCTM Tool life
0 : Specified by the number of times
1 : Specified by time

TLSK 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

GRST Tool exchange reset signal
0 : Clears only the execution data of a specified group
1 : Clears the execution data of all entered groups

IGSK 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 TLSK, #3 of parameter No. 0039).
1 : Ignores a tool skip signal

IGIN Tool back number
0 : Not ignored
1 : Ignored

M6TCD 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	
0041	M6TST								(M series)

[Data type] Bit

M6TST 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 M6TCD No. 0039#7.
1 : The tool group life is counted immediately.

0336	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.

0489	Remaining tool life allowed until a new tool is selected

[Data type] Word

[Valid data range] 0 to 9999 times (if the tool life is specified in the number of times that the tool is used)
0 to 4300 minutes (if the tool life is specified in minutes)

The signal TLCHB<F192#2> is issued when the tool life counter reaches a parameter–specified value.

0599	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.

Alarm and message

Number	Message	Description
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. 0039 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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.10.2	TOOL LIFE MANAGEMENT FUNCTION
OPERATOR’S MANUAL (For Machining Center) (B–61394E)	II.10.3	TOOL LIFE MANAGEMENT FUNCTION

10.4 CUTTER COMPENSATION

10.4.1 Cutter Compensation B, 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 (cutter compensation B only), 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.

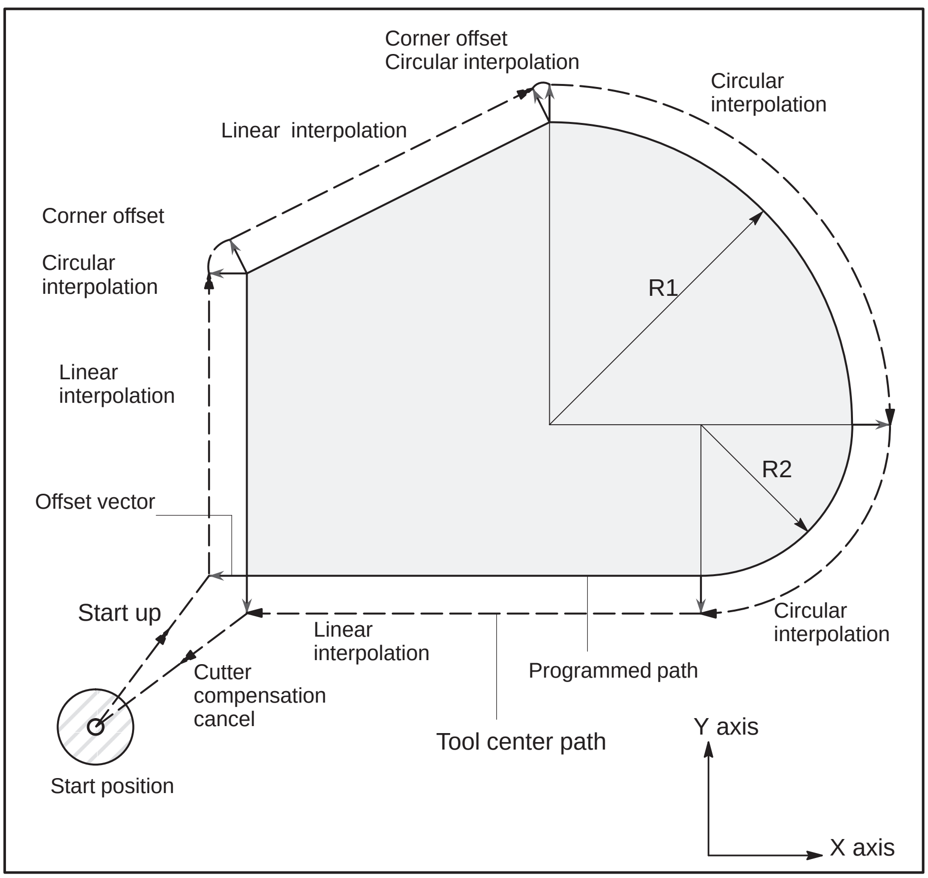


Fig.10.4.1 (a) Outline of Cutter Compensation B

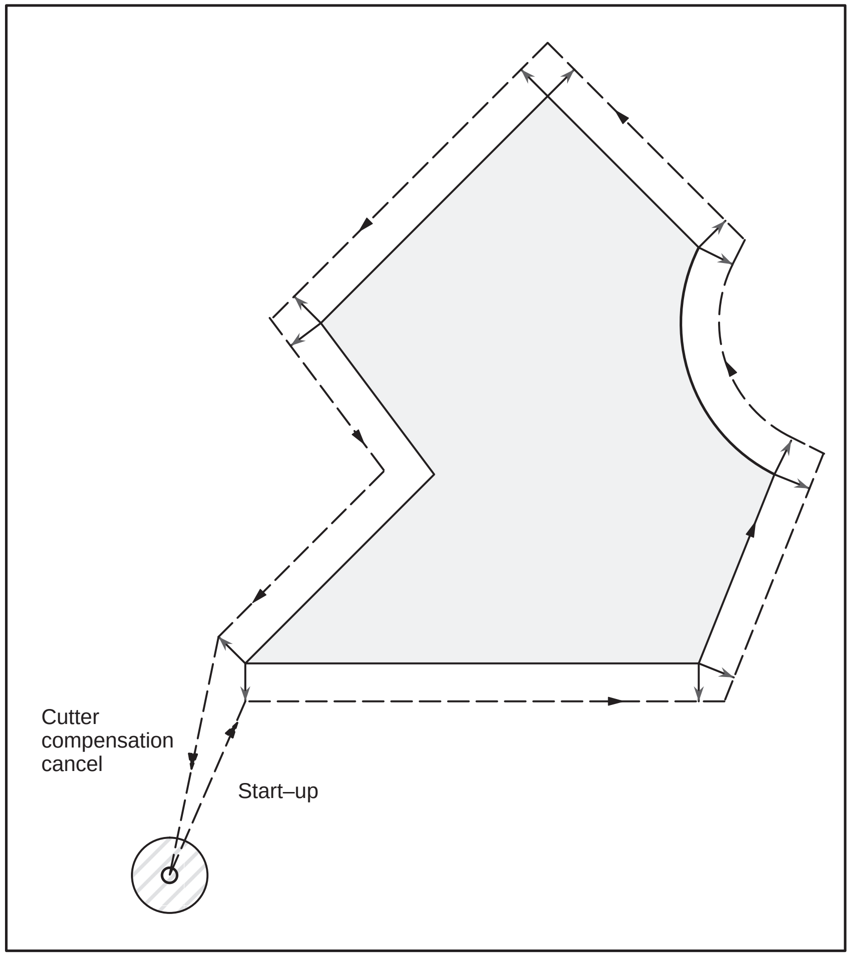


Fig.10.4.1 (b) Outline of Cutter Compensation C

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0016						SUPM			(M series)

[Data type] Bit

SUPM Start up or cancel in cutter compensation C
0 : Type A
1 : Type B

	#7	#6	#5	#4	#3	#2	#1	#0	
0036		OFRD							(M series)

[Data type] Bit

OFRD Offset number of tool length compensation, cutter compensation and tool offset
0 : Specifies the tool length compensation, cutter compensation and tool offset using H codes
1 : Specifies the tool length compensation using an H code, and cutter compensation C using a D code

	#7	#6	#5	#4	#3	#2	#1	#0	
0062						G40V			(M series)

[Data type] Bit

G40V 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.

0557	Limit value that ignores the vector when a tool moves on the outside of a corner during cutter compensation C	(M series)

[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.

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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.14.4	CUTTER COMPENSATION B
	II.14.5, 14.6	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.

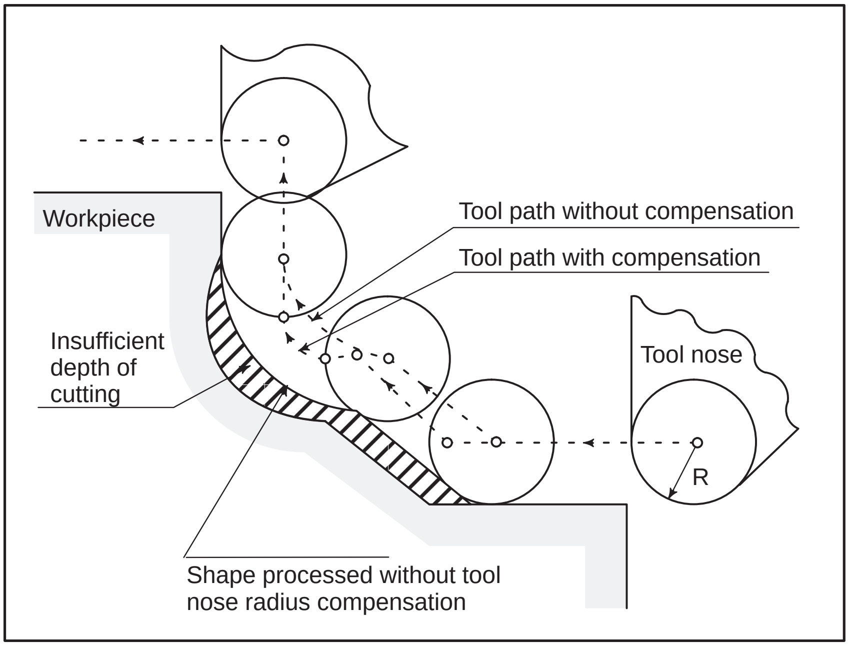


Fig. 10.4.2 Tool path of tool nose radius compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0075					WNPT			

[Data type] Bit

WNPT Imaginary tool tip direction used for tool nose radius compensation, when the geometry/wear compensation option is equipped, is the direction specified by:
0 : Geometry offset number
1 : Wear offset number

0557	Limit value that ignores the vector when a tool moves on the outside of a corner during too nose radius compensation	(T series)

[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.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	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, and F . . . . M series
X, Y, Z, U, V, W, A, B, C, I, J, K, R, and F. . . . . T series

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal notation is used, a value without decimal point is considered to be specified in millimeters. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal notation by using the CPRD (bit 7 of parameter 0015). 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
0015	CPRD							

[Data type] Bit

CPRD 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 s is assumed. (Pocket calculator type decimal point programming)

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 two decimal points were input.) Modify the program.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.8.4	DECIMAL POINT PROGRAMMING
OPERATOR’S MANUAL (For Lathe) (B–61394E)	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 bits 5 (GSPC) and 1 (GSPG) of parameter 0036.

Table 11.2 G code list (1/2)

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/Helical interpolation CW
G03	G03	G03		Circular interpolation/Helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High-speed cycle cutting
G10	G10	G10		Data setting
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 reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable lead thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		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, max. spindle speed setting
G52	G52	G52		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

Table 11.2 G code list (2/2)

G code A	G code B	G code C	Group	Function
G65	G65	G65	00	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 or balance cutting mode
G69	G69	G69		Mirror image for double turrets OFF or balance cutting mode cancel
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 (Other than 0–GCC and 00–GCC)
G74	G74	G76		End face peck drilling (Other than 0–GCC and 00–GCC)
G75	G75	G77		Outer diameter/internal diameter drilling (Other than 0–GCC and 00–GCC)
G76	G76	G78		Multiple threading cycle (Other than 0–GCC and 00–GCC)
G71	G71	G72	01	Traverse grinding cycle (For 0–GCC and 00–GCC)
G72	G72	G73		Traverse direct constant–demension grinding cycle (For 0–GCC and 00–GCC)
G73	G73	G74		Oscilation grinding cycle (For 0–GCC and 00–GCC)
G74	G74	G75		Oscilation direct constant–dimension grinding cycle (For 0–GCC and 00–GCC)
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 revolution feed
—	G90	G90	03	Absolute programming
—	G91	G91		Incremental programming
—	G98	G98	11	Return to initial level (See **Note 6**)
—	G99	G99		Return to R point level (See **Note 6**)
G107	G107	G107	00	Cylindrical interpolation
G112	G112	G112	21	Polar coordinate interpolation mode
G113	G113	G113		Polar coordinate interpolation mode cancel
G250	G250	G250	20	Polygonal turning cancel
G251	G251	G251		Polygonal turning

Note

NOTE

1 If the CNC enters the clear state (see bit 6 (CLER) of parameter 0045) 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) G22 is set when the system is cleared due to power-on. When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.
   (4) Setting bit 6 (G01) of parameter 0011 determines which code, either G00 or G01, is effective.
   (5) Setting bit 7 (G91) of parameter 0030 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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0011		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)

	#7	#6	#5	#4	#3	#2	#1	#0
0030	G91							

[Data type] Bit

G91 When the power is turned on or when the control is cleared
0 : G90 mode (absolute command)
1 : G91 mode (incremental command)

	#7	#6	#5	#4	#3	#2	#1	#0
0036			GSPC				GSP	

[Data type] Bit

GSP Type of the special G code system
1 : Type B
0 : Type A

GSPC Type of the special G code system
1 : Type C
0 : Type A

	#7	#6	#5	#4	#3	#2	#1	#0
0045		CLER						

[Data type] Bit

CLER 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.

For the reset and clear states, refer to APPENDIX E of operator's manual (B–61394E).

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.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	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

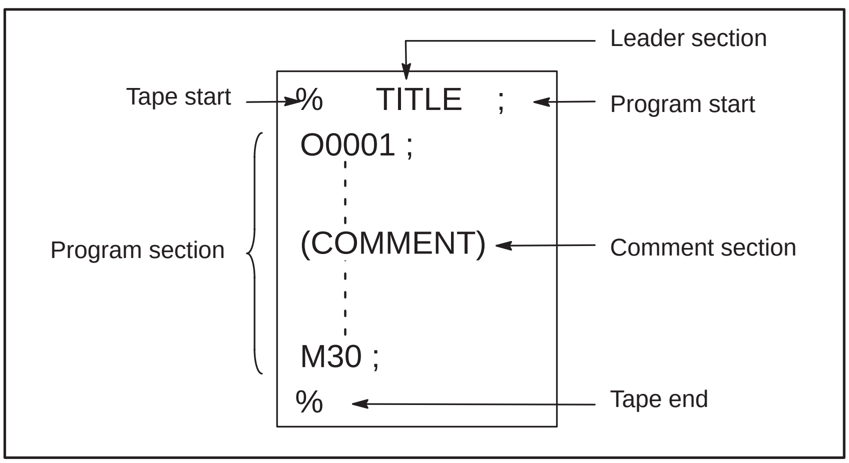


Fig.11.3 Program configuration

Parameter

· Setting entry

TVON When a program is registered to memory, a TV check on it is:
0 : Not performed.
1 : Performed.

	#7	#6	#5	#4	#3	#2	#1	#0
0018		TVC						

TVC A TV check on comments in a program is:
0 : Performed.
1 : Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
0019		NEOP						

[Data type] Bit

NEOP 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
0063						M198P		

[Data type] Bit

M198P Address P of the block including M198 in the subprogram call function
0 : Indicating a file number
1 : Indicating a program number

0248	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

This parameter sets the M code for calling a program in a file stored.

> **NOTE**
> The M code is judged to be M198 when zero is specified as the setting value.

	#7	#6	#5	#4	#3	#2	#1	#0
0396	EORRE							

[Data type] Bit

EORRE When the end-of-record mark (%) is read during program execution:
0 : P/.S alarm No.0008 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.)

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 setting parameter TVON is set to 1).
008	END OF RECORD	The end of record (%) was specified.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.12	PROGRAM CONFIGURATION
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.12	PROGRAM CONFIGURATION

11.4 INCH/METRIC CONVERSION

General

Either inch or metric input can be selected by G code.

Parameter

· Setting entry

INCH Unit of input
0 : In mm
1 : In inches

	#7	#6	#5	#4	#3	#2	#1	#0
0001								SCW

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

[Data type] Bit

SCW 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
0063							PRSTIN	MTDSPI

[Data type] Bit

MTDSPI 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.)

PRSTIN Coordinates at the reference position when a coordinate system is set automatically
0 : Value set in parameter 0708 to 0711 is used.
1 : For input in mm, the value set in parameter 0708 to 0711 is used, or for input in inches, the value set in parameter 0815 to 0818 is used.

	#7	#6	#5	#4	#3	#2	#1	#0	
0077								MICRF	(T series)

[Data type] Bit

MICRF 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.

0708 – 0711	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (Metric input)	0.01	0.001	0.0001	mm
Linear axis (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 value of the reference position on each axis used for setting a coordinate system automatically.

0815 – 0818	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

Set the coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches.

> **NOTE**
> This parameter is valid when PRSTIN in parameter 0063#1 is set to 1.

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.

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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.8.3	INCH/METRIC CONVERSION(G20, G21)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.8.2	INCH/METRIC CONVERSION(G20, G21)

11.5 HIGH SPEED CYCLE CUTTING

General

This function can convert the machining profile to a data group that can be distributed as pulses at high-speed by the macro compiler and macro executor. The function can also call and execute the data group as a machining cycle using the CNC command (G05 command).

• Format

G05 P10○○○ L○○○ ;

P10○○○ is number of the cutting cycle to be called first:
P10001 to P10999
L○○○ is repetition count of the cutting cycle
(L1 applies when this parameter is omitted.) :
L1 to L999

Call and execute the data for the high speed cutting cycle specified by the macro compiler and macro executor using the above command.
Cycle data can be prepared for up to 999 cycles. Select the machining cycle by address P. More than one cycle can be called and executed in series using the cycle connection data in the header.
Specify the repetition count of the called machining cycle by address L. The repetition count in the header can be specified for each cycle.
The connection of cycles and their repetition count are explained below with an example.

Example) Assume the following:
Cycle 1 Cycle connection data 2 Repetition count 1
Cycle 2 Cycle connection data 3 Repetition count 3
Cycle 3 Cycle connection data 0 Repetition count 1
G05 P10001 L2 ;
The following cycles are executed in sequence:
Cycles 1, 2, 2, 2, 3, 1, 2, 2, 2, and3

• Number of control axes

Four axes maximum can be controlled. Four axes can be controlled simultaneously.

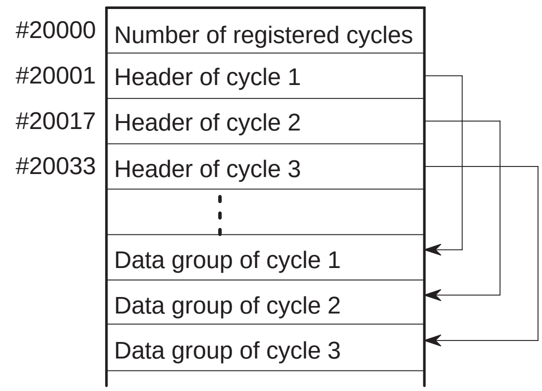
• Pulse distribution

Set the number of pulses per cycle in parameter 0055#4 to #6 as a macro variable (#20000 to #85535) for high speed cycle cutting using the macro compiler and macro executor.
The unit for the number of pulses is the least input increment.

- **Configuration of high–speed cycle cutting data**

Data for the high speed cycle cutting is assigned to variables (#20000 to #85535) for the high–speed cycle cutting by the macro compiler and macro executor.

Configuration of the high speed cycle cutting data



- **Number of Registered Cycles**

Specify the number of cycles (number of headers) of high-speed cycle cutting data. Values from 1 to 999 can be specified.

- **Header**

The header for high-speed cycle cutting data has the following configuration:

Header configuration

#20001/20017/20033..	Cycle repetition count
#20002/20018/20034..	Cycle connection data
#20003/20019/20035..	Number of data items
#20004/20020/20036..	Data type
#20005/20021/20037..	Variable assigned to the 1st axis data
#20006/20022/20038..	Variable assigned to the 2nd axis data
#20007/20023/20039..	Variable assigned to the 3rd axis data
#20008/20024/20040..	Variable assigned to the 4th axis data
#20009/20025/20041..	Variable assigned to the 5th axis data
#20010/20026/20042..	Variable assigned to the 6th axis data
#20011/20027/20043..	Total number of fixed data items for the 1st axis
#20012/20028/20044..	Total number of fixed data items for the 2nd axis
#20013/20029/20045..	Total number of fixed data items for the 3rd axis
#20014/20030/20046..	Total number of fixed data items for the 4th axis
#20015/20031/20047..	Total number of fixed data items for the 5th axis
#20016/20032/20048..	Total number of fixed data items for the 6th axis

Explanations

● Cycle repetition count

Specify the repetition count for the cycle. Values from 0 to 32767 can be specified. When 0 or 1 is specified, the cycle is executed once.

● Cycle connection data

Specify the number (1 to 999) of the cycle to be executed after the cycle. When no connection cycle exists because of the last cycle, specify 0.

● Number of data items

Specify the number of data items per cycle. Valid values are from 1 to 32767.
When a fixed data item is specified, the fixed data is repeated for the specified number of times in one cycle.

● Data type

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
–	–	–	–	r4	r3	r2	r1	–	–	–	–	t4	t3	t2	t1

The bits from t1 to t4, corresponding to the 1st to 6th axes, have the following meanings:
0: Distribution data is always constant.
1: Distribution data is variable or fixed.
When the distribution data is variable or fixed, the bits from r1 to r4, corresponding to the 1st to 4th axes, have the following meanings:
0: Distribution data is read forward.
1: Distribution data is read backwards.
Because the data consists of bits, it is necessary to use a binary-coded decimal value when setting it using the macro compiler and macro executor.
Example)
When constant data is assigned to the 1st and 2nd axes and variable data is assigned to the 3rd and 4th axes, #20004=12; (t4 and t3: 1, t2 and t1: 0)

● Variables assigned to data for the 1st to 4th axes

·**Constant data**
When the corresponding data type bit (t4 to t1) is 0, specify "distribution data value".

·**Variable data**
When the corresponding data type bit (t4 to t1) is 1 and the total number of fixed data items = 0, specify "(Storing start data variable No. of the distribution data)/10".

·**Fixed data**
When the corresponding data type bit (t4 to t1) is 1 and the total number of fixed data items is other than 0, specify "(Storing start data variable No. of the distribution data)/10".
The applicable value for the variable data and fixed data is 2001 to 8553. It is not possible to start storing data in the executable format from a variable No. that is not a multiple of 10.
To read the distribution data backwards, set the variable No. of the data to be distributed last. For example, to read the distribution data in #25000 to #25999 backwards, set 2500 as the data assignment variable.

● Total number of fixed data items for the first to 4th axes

Set the length of the fixed data for the cycle.
The first address of the fixed data must be specified by the data assignment variable. When the total number of fixed data items = 0 and the corresponding data type bit (t4 to t1) is 1, the data is regarded as a variable data.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0055		IT2	IT1	IT0				CSP	(T series)
		IT2	IT1	IT0					(M series)

[Data type] Bit

CSP Cs contouring control function dedicated to a piston lathe is
0: Not used.
1: Used.

IT0, IT1, IT2

IT2	IT1	IT0	
0	0	0	Interpolates the G05 data in 8ms
0	0	1	Interpolates the G05 data in 2ms
0	1	0	Interpolates the G05 data in 4ms
0	1	1	Interpolates the G05 data in 1ms
1	0	0	Interpolates the G05 data in 16ms
1	1	1	Interpolates the G05 data in 0.5ms

0597	Maximum number of simultaneously controlled axes when G05 is specified during high–speed cycle cutting or No. of controlled axes in high–speed remote buffer

[Data type] Word

[Unit of data] 1 to 4

This parameter sets the maximum number of simultaneous control axes when G05 is specified during high-speed cycle cutting or sets the number of control axes in a high-speed remote buffer.

Alarm and Message

Number	Message	Description
115	ILLEGAL VARIABLE NUMBER	The header contents are improper in a high–speed cycle cutting. This alarm is given in the following cases: 1. The header corresponding to the specified cutting cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The storing start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 0597) SETTING ERROR	The number of controlled axes set by the parameter 0597 exceeds the maximum number. Modify the parameter setting value.

Warning

WARNING

Single block stop, dry run, feedrate override, automatic acceleration/deceleration and handle interruption are disabled during high-speed cycle cutting.

Note

NOTE

1 Use of the high–speed cycle cutting function requires expanded RAM and limits the maximum specifiable size of part program memory to within 80 m in terms of tape length.

2 Set the total number of distribution data items for one cycle to a multiple of the following values, according to the distribution cycle. This does not apply when the distribution cycle is 16 ms or 8 ms.
If the total number is not a multiple of one of the following values, movement in the remaining cycle becomes zero.

Distribution cycle 4 ms: Multiple of 2
Distribution cycle 2 ms: Multiple of 4
Distribution cycle 1 ms: Multiple of 8

For example, when all 41 data items (distribution cycle: 2 ms) are specified, movement is zero in the remaining 3 ms.

Reference Item

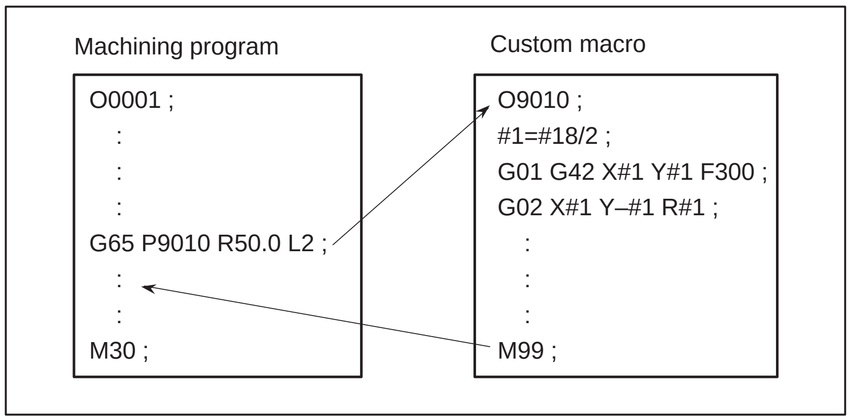
OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.20	HIGH SPEED CYCLE CUTTING
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.20	HIGH SPEED CYCLE CUTTING

11.6 CUSTOM MACRO

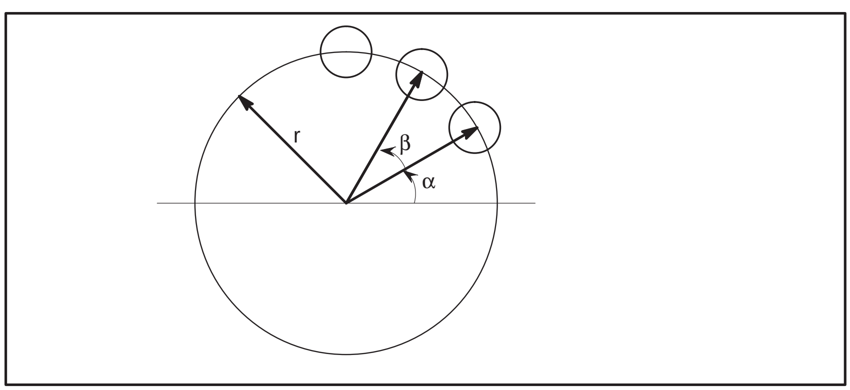
11.6.1 Custom Macro

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
(Specification Custom Macro B)

Signal

Custom Macro Input Signal UI000 to UI015 <G130, G131>

[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 UO000 to UO015 <F162, F163> UO100 to UO131 <F196 to F199>

[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 UO001 UO002 UO003 : UO014 UO015	1 1 1 1 : 1 1	#1100 #1101 #1102 #1103 : #1114 #1115	"0" at "0" and "1" at "1"
UO000 to UO015 UO100 to UO115	16 32	#1132 #1133	16–bit binary code *1 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 0 and Vi=1 when UO1i is 1

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
G130	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G131	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008

	#7	#6	#5	#4	#3	#2	#1	#0
F162	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F163	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Parameter

- **Setting for single block stop**

	#7	#6	#5	#4	#3	#2	#1	#0
0011			SBKM					

[Data type] Bit

SBKM Custom macro statement
0 : Not stop the single block
1 : Stops the single block

- **Other settings**

	#7	#6	#5	#4	#3	#2	#1	#0
0040	LOCC	COMC	TMCR				DPOSUP	

[Data type] Bit

DPOSUP Reading zero when data is output using a DPRNT command
0 : Outputs a space
1 : Outputs no data

TMCR Custom macro (subprogram)
0 : Not called using a T code
1 : Called using a T code

COMC Custom macro B's common variables Nos. 100 through 149
0 : Cleared to "vacant" by reset
1 : Not cleared by reset

LOCC Custom macro B's local variables #1 through #33
0 : Cleared to "vacant" by reset
1 : Not cleared by reset

	#7	#6	#5	#4	#3	#2	#1	#0
0057				CROUT				

[Data type] Bit

CROUT In BPRNT or DPRNT, after the data is outputted in ISO code:
0 : Only the "LF" is outputted.
1 : "LF" and "CR" are outputted.

- **Setting when macro statement is input/output with EIA code**

	#7	#6	#5	#4	#3	#2	#1	#0
0042	*7	*6	*5	*4	*3	*2	*1	*0
0043	=7	=6	=5	=4	=3	=2	=1	=0
0044	#7	#6	#5	#4	#3	#2	#1	#0
0053	[7	[6	[5	[4	[3	[2	[1	[0
0054	]7	]6	]5	]4	]3	]2	]1	]0

[Data type] Bit

These parameters are used to input/output macro statements.

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**

0248	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**

0220	G code that calls the custom macro of program number 9010
0221	G code that calls the custom macro of program number 9011
0222	G code that calls the custom macro of program number 9012
0223	G code that calls the custom macro of program number 9013
0224	G code that calls the custom macro of program number 9014
0225	G code that calls the custom macro of program number 9015
0226	G code that calls the custom macro of program number 9016
0227	G code that calls the custom macro of program number 9017
0228	G code that calls the custom macro of program number 9018
0229	G code that calls the custom macro of program number 9019

[Data type] Word

[Valid data range] 1 to 9999

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**

0240	M code that calls the subprogram of program number 9001
0241	M code that calls the subprogram of program number 9002
0242	M code that calls the subprogram of program number 9003

[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 9003.

> **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**

0230	M code that calls the sub program of program number 9020
0231	M code that calls the sub program of program number 9021
0232	M code that calls the sub program of program number 9022
0233	M code that calls the sub program of program number 9023
0234	M code that calls the sub program of program number 9024
0235	M code that calls the sub program of program number 9025
0236	M code that calls the sub program of program number 9026
0237	M code that calls the sub program of program number 9027
0238	M code that calls the sub program of program number 9028
0239	M code that calls the sub program of program number 9029

[Data type] Two–word

[Valid data range] 1 to 99999999

These parameters set the M codes that call the sub programs of program numbers 9020 through 9029.

NOTE
Setting value 0 is invalid. No custom macro can be called by M00.

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. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 to 999). 3. The number of data in the header is out of the allowable range (0 to 32767). 4. The storing start data variable number of executable format data is out of the allowable range (#20000 to #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.

Number	Message	Description
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.
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 UNDEFINED	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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.15 and 16	CUSTOM MACRO A/B
OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.15 and 16	CUSTOM MACRO A/B

11.6.2 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.

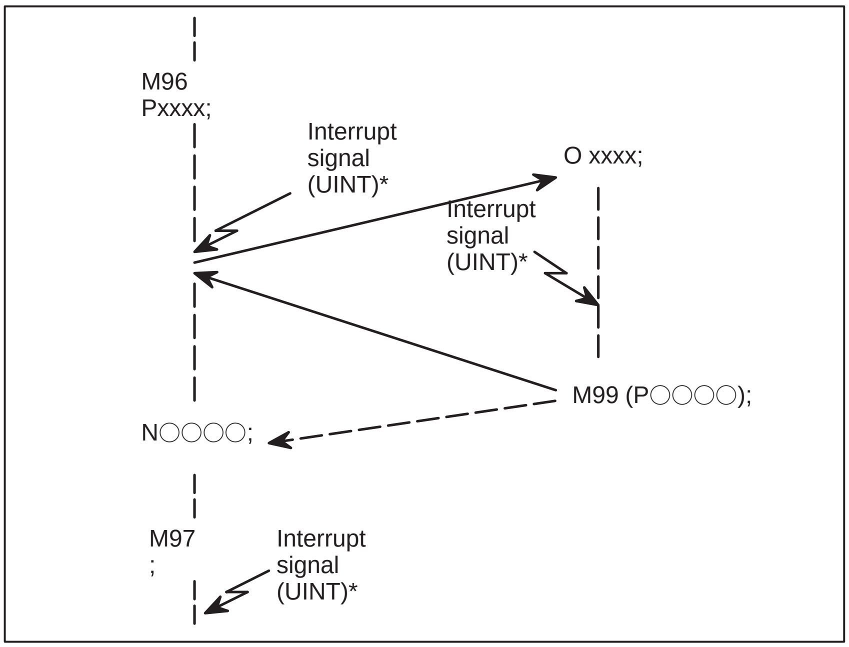


Fig 11.6.2 Interruption type custom macro function

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.

When the interrupt signal (UINT, marked by * in Fig. 11.6.2) is input during execution of the interrupt program or after M97 is specified, it is ignored.

Signal

Interrupt Signal for Custom Macro UINT<G140#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 0056, 0246 and 0247.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G140					UINT			

Parameter

- **Various Setting for Custom Macro**

	#7	#6	#5	#4	#3	#2	#1	#0
0056	MUSR	MCYL	MSUB	MPRM	MTSE	MBLK	MSKT	

[Data type] Bit

MSKT 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 #5061 and later)

MBLK 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)

MTSE Custom macro interrupt signal UINT
0 : Edge trigger method (Rising edge)
1 : Status trigger method

MPRM Custom macro interrupt valid/invalid M code
0 : M96/M97
1 : M code set using parameters (Nos. 0246 and 0247)

MSUB 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)

MCYL Custom macro interrupt
0 : Not performed during cycle operation
1 : Performed during cycle operation

MUSR Interrupt–type custom macro
0 : Not used
1 : Used

• Setting M code that makes interruption effective and ineffective

0246	M code that validates a custom macro interrupt
0247	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. 0056, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPRM is 0, irrespective of the state of this parameter.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.16.11	INTERRUPTION TYPE CUSTOM MACRO
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.16.11	INTERRUPTION TYPE CUSTOM MACRO

11.6.3 Custom Macro Variables Common between Tool Posts (0–TTC)

General

In 0–TTC, common variables are provided separately for each tool post; variable #n used with a tool post 1 is different from variable #n used with tool post 2. By parameter setting (No. 0047#5(VR1), #6(VR5) and No. 0218), however, some or all of common variables #100 to # 149 and #500 to #531 can be made usable commonly by tool post 1 and tool post 2 so that such variables can be written or read for either tool post. Such variables are referred to as custom macro variables common between tool posts.

Parameter

- **Setting the no. of custom macro variables common between tool posts**

	#7	#6	#5	#4	#3	#2	#1	#0
0047		VR5	VR1					

[Data type] Bit

VR1, VR5 Specify the first number for custom macro variables common to both tool posts in the 0–TTC.

VR5	VR1	First number for custom macro variables common to both tool posts
0	0	None (no common custom macro variable is used.)
0	1	No. 100
1	0	No. 500
1	1	Nos. 100 and 500

0218	Number of custom macro variables common between tool posts

[Data type] Byte

[Unit of data] Number of custom macro variables

[Valid data range] 0 to 50

This parameter specifies the number of variables commonly used for both tool posts 1 and 2 (custom macro variables common between tool posts) that are part of custom macro variables 100 to 149 and 500 to 532.
The custom macro variables common to tool posts can be written or read for either of the tool posts.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.24.5	CUSTOM MACRO VARIABLES COMMON TO TOOL POSTS

11.7 CANNED CYCLE (M SERIES)/CANNED CYCLE FOR HOLE MACHINING (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.

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

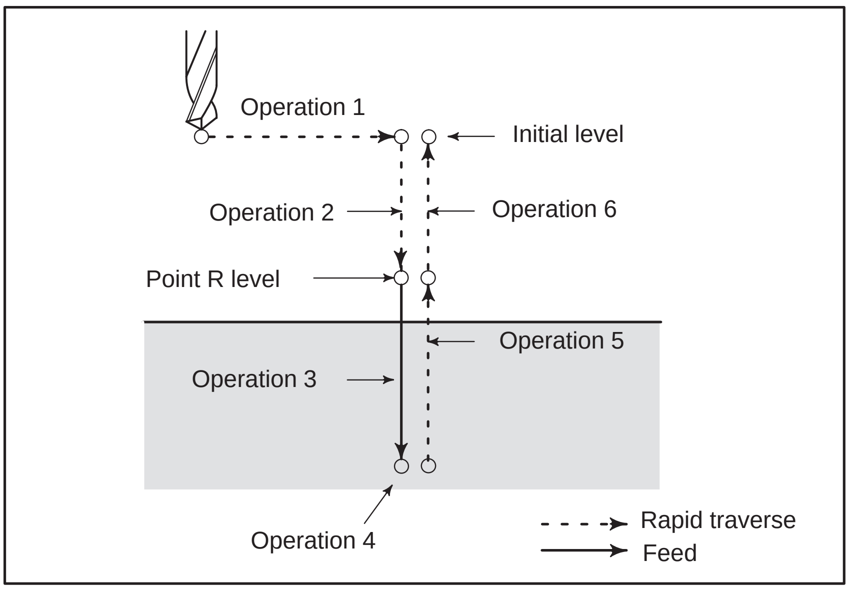


Fig. 11.7 (a) Canned cycle operation sequence

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:

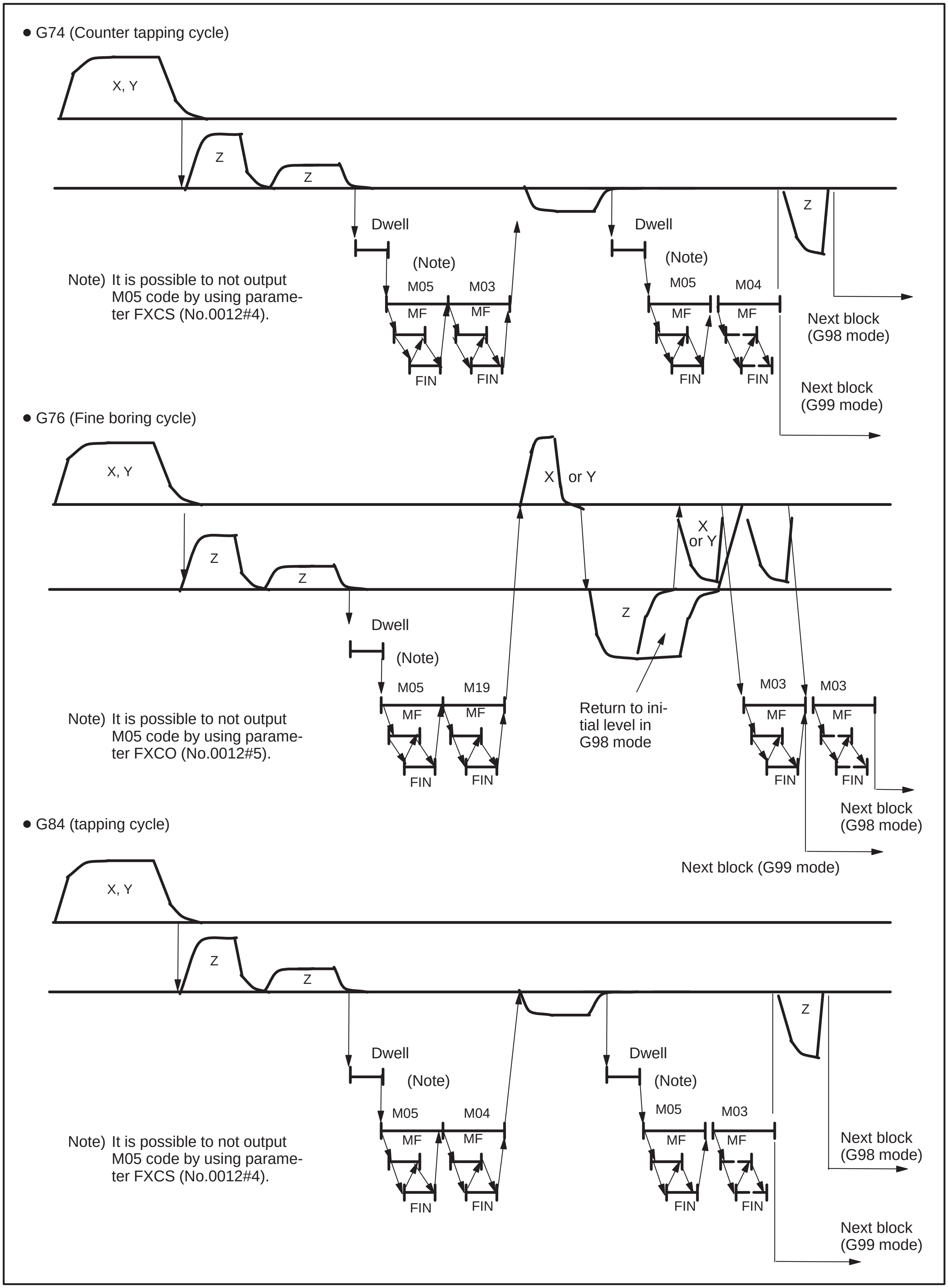


Fig.11.7 (b) Canned cycle for M series (1/2)

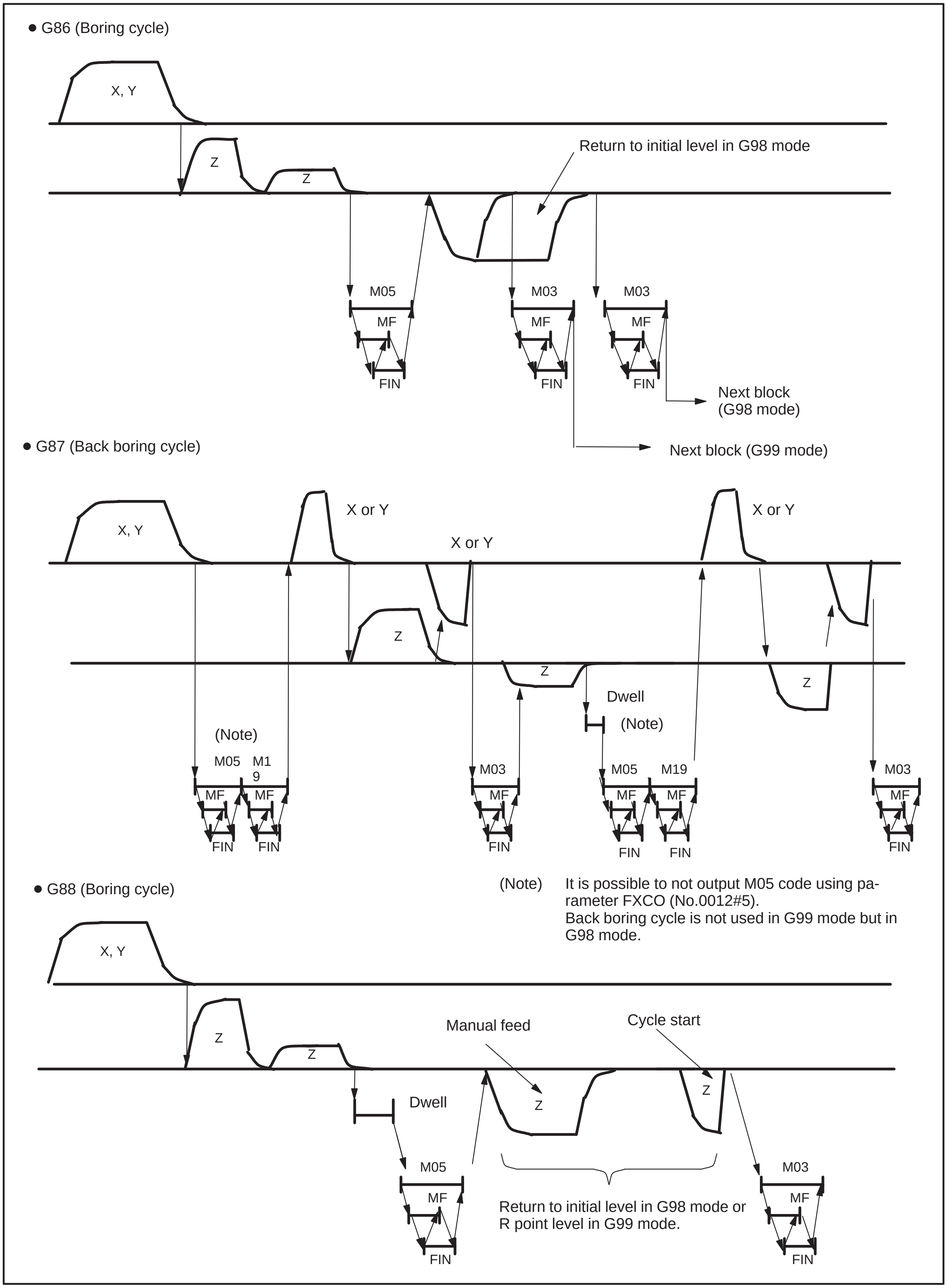


Fig.11.7 (b) Canned cycle for M series (2/2)

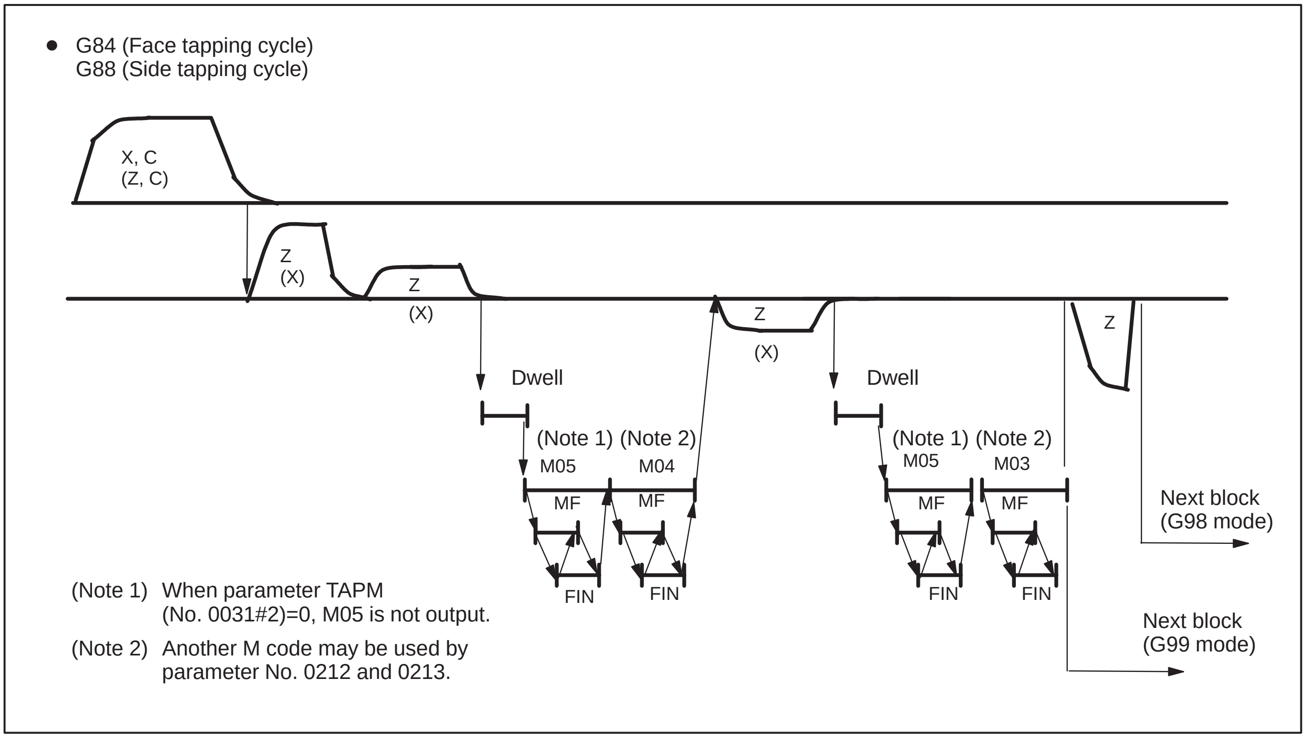


Fig 11.7 (c) Canned cycle for T series

- **M code used for C–axis clamp/unclamp (T series)**

When an M code specified in parameter No.0204 for C–axis clamp/unclamp is coded in a program, 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. 0591.

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

In M series, the TAPDRN (bit 5 of parameter No.0065) specifies whether dry run is valid during tapping.

Signal

Tapping signal TAP<F149#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
F149			TAP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0002			PMXY2	PMXY1					(M series)

[Data type] Bit

PMXY2, PMXY1 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.

PMXY2	PMXY1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	–X	–Z	–Y
1	0	+Y	+X	+Z
1	1	–Y	–X	–Z

	#7	#6	#5	#4	#3	#2	#1	#0	
0011				MCF					(M series)

[Data type] Bit

MCF G81
0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

		#7	#6	#5	#4	#3	#2	#1	#0	
0012				FXCO	FXCS					(M series)

[Data type] Bit

FXCS 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:
0 : Outputs M05
1 : Not output M05

FXCO 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
0031					RTCT	ILVL	TAPM		

[Data type] Bit

TAPM 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:
0 : Not output M05
1 : Outputs M05

ILVL Initial point position in drilling canned cycle
0 : Not updated by reset
1 : Updated by reset

RTCT For drilling, G83 and G87
0 : Make a return by a specified escape amount.
1 : Make a return to the R point level.

		#7	#6	#5	#4	#3	#2	#1	#0	
0057			FXY							(M series)

[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

0204		C–axis clamp M code in drilling canned cycle	(T series)

[Data type] Byte

[Valid data range] 0 to 99

This parameter sets the C–axis clamp M code in a drilling canned cycle.

0212	Spindle forward–rotation M code in drilling canned cycle	(T series)

[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.

0213	Spindle reverse–rotation M code in drilling canned cycle	(T series)

[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.

0591	Dwell time when C–axis unclamping is specified in drilling canned cycle	(T series)

[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.

0592	Return value of canned cycle G83 for drilling	(T series)
0531	Return value of high–speed peck drilling cycle G73	(M series)

[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 32767

For M series, this parameter sets the return value in high–speed peck drilling cycle G73 (G83 for T series).

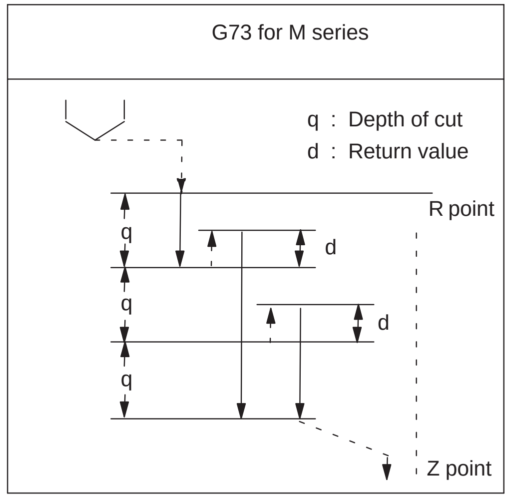


Fig. 11.7 (d) High–speed peck drilling cycle (G73) for M series

For T series, this parameter sets the return or clearance value in drilling canned cycle G83.

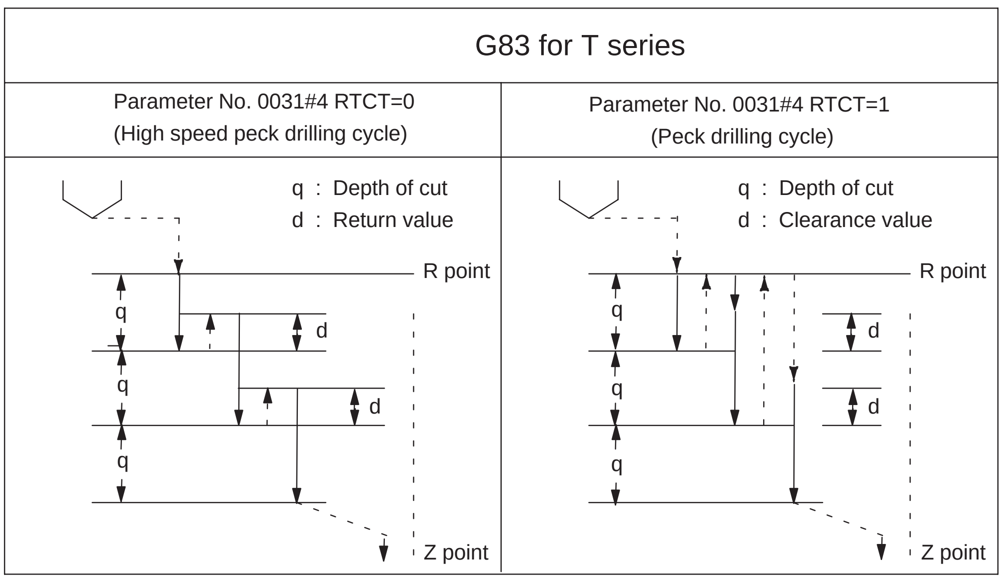


Fig. 11.7 (e) Drilling canned cycle (G83) for T series

0532	Clearance canned cycle G83	(M series)

[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 32767

This parameter sets the clearance value in peck drilling cycle G83.

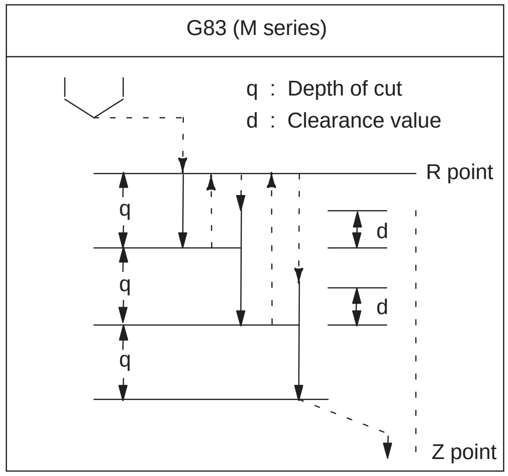


Fig. 11.7 (f) Peck drilling cycle (G83)

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.0057#6) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis. (M series)

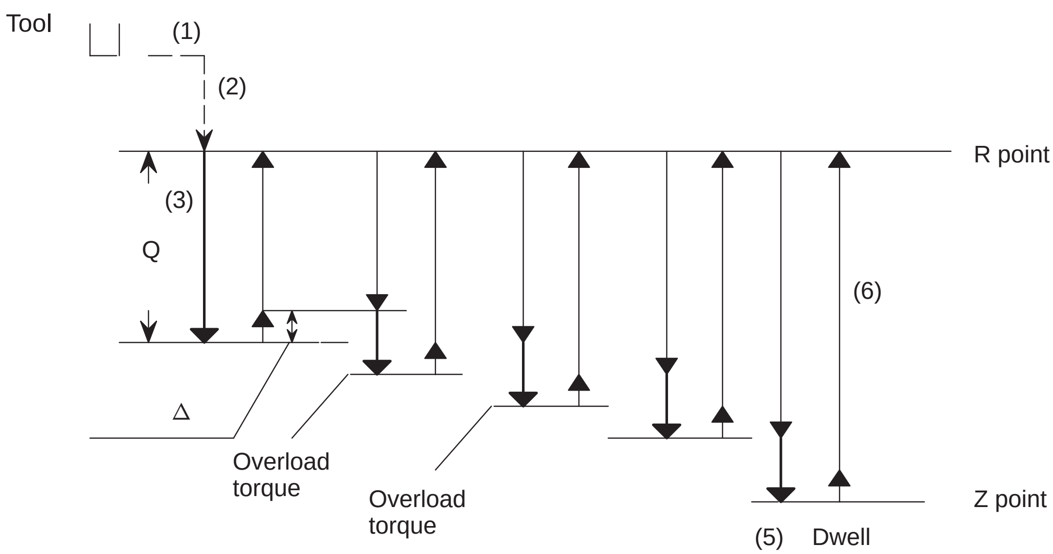
Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.13.1	CANNED CYCLE
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.13.3	CANNED CYCLE FOR DRILLING

11.8 SMALL–DIAMETER PECK DRILLING CYCLE (M SERIES)

General

A tool with an overload torque detection function can change the spindle speed and cutting feedrate during peck drilling, using an overload torque detection signal received from the tool as a skip signal. Such peck drilling enables small–diameter peck drilling. The sequence chart of a small–diameter peck drilling cycle is shown below.



Δ : The first minute escape amount used in moving back to the R point, and an amount of clearance at the start point (end point of the previous cycle) of the second and subsequent cycles

Q : Cutting depth per cycle

 : Movement at a rapid traverse rate

 : Movement at a parameter– or program–specified rate

 : Movement at a program–specified cutting feedrate

The steps of the small–diameter peck drilling cycle are as follows:

(1) Position along the X– and Y–axes.

(2) Position to the R point along the Z–axis.

(3) Cut along the Z–axis (maximum cutting depth Q).

(4) Repeat the following steps until the hole bottom along the Z–axis is reached.

- Reverse (→ minute escape amount Δ).
- Reverse (→ R point).
- Advance (R point → start point + clearance Δ).
- Cut (cutting depth Q + Δ).

(5) Dwell at the hole bottom.

(6) Return to the R point (or initial point) along the Z–axis.

Specification

(1) A small–diameter peck drilling cycle is executed by issuing a G83 after a small–diameter peck drilling cycle M code. The small–diameter peck drilling cycle in progress signal is output when R point positioning along the drill axis begins, and kept output until a return to the R point/initial level is completed.

(2) A skip signal (X008#7) is used as the overload torque detection signal. The skip signal is valid only when the drill is moving forward or cutting between the R and Z points. If the skip signal is input when the drill is moving forward or cutting, the drill starts to move backward immediately.

(3) Cutting conditions are changed for each pecking cycle (advance → cut → reverse) during one G83 cycle.

- Changing the cutting feedrate

  A cutting feedrate specified in a program using an F code is changed for each of the second and subsequent cutting cycles. Parameter No. 307 specifies the ratio of a change to be used when a skip signal has been received during the previous cutting cycle. Parameter No. 308 specifies that to be used when a skip signal has not been received during the previous cutting cycle.

  Cutting feedrate $= F \times \alpha$

  (First cutting cycle) $\alpha = 1.0$
  (Second and subsequent cutting cycles) $\alpha = \alpha \times \beta \div 100$
  where β is a change ratio for one cutting cycle
  If a skip signal is input during the previous cutting cycle:
  β = parameter No. 307
  If a skip signal is not input during the previous cutting cycle:
  β = parameter No. 308
  If the cutting feedrate change ratio (α) becomes less than a value specified in parameter No. 309, no further change will be made to the cutting feedrate.

- Changing the spindle speed

  A spindle speed specified in a program using an S code is changed for each of the second and subsequent cutting cycles. Parameter No. 305 specifies the ratio of a change to be used when a skip signal has been received during the previous cutting cycle. Parameter No. 306 specifies that to be used when a skip signal has not been received during the previous cutting cycle.

  Spindle speed $= S \times \gamma$

  (First cutting cycle) $\gamma = 1.0$
  (Second and subsequent cutting cycles) $\gamma = \gamma \times \delta \div 100$
  where δ is a change ratio for one cutting cycle
  If a skip signal is input during the previous cutting cycle:
  δ= parameter No. 305
  If a skip signal is not input during the previous cutting cycle:
  δ= parameter No. 306
  If the cutting feedrate is clamped at the lowest limit, no change will be made to the spindle speed.

(4) During pecking, advance and reverse movements are performed as cutting feed rather than rapid positioning.

(5) If pecking advance and reverse feedrates are specified using an I code, the same format as for F codes is used. An I code specified using a G83 remains valid until a G80 is issued or a reset occurs.

Signal

Skip signal SKIP <X008#7>

[Classification] Input signal

[Function] Used as a spindle overload torque detection signal during the execution of a small–diameter peck drilling cycle.

[Operation] If the drill is moving forward or cutting between the R and Z points during the execution of a small–diameter peck drilling cycle, setting the skip signal to 1 causes the drill to move backward.

Small–diameter peck drilling cycle execution in progress signal PECK2 <F180#7>

[Classification] Output signal

[Function] Indicates the operating status of a small–diameter peck drilling cycle.

[Operation] The signal is 1 during the execution of a small–diameter peck drilling cycle. It is 0 when a small–diameter peck drilling cycle is not being executed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
X008	SKIP								(M series)
F180	PECK2								(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0398	CHGNRM	CHGSKP						

CHGSKP If an overload torque signal is received:
1 : The feedrate and spindle speed will be changed.
0 : Neither the feedrate nor spindle speed will be changed.

CHGNRM If a cutting depth per cycle is completed before an overload torque signal is received:
1 : The feedrate and spindle speed will be changed.
0 : Neither the feedrate nor spindle speed will be changed.

0304	PECK_M

Specify a small–diameter peck drilling cycle M code.

[Valid data range] 3 to 999

0305	SSOPR1

Specify a percentage at which the spindle speed is to be changed when the drill is caused to move back as directed by an overload torque signal.

[Valid data range] 0 to 255%

0306	SSOPR2

Specify a percentage at which the spindle speed is to be changed when the drill is caused to move back without receiving an overload torque signal.

[Valid data range] 0 to 255%

0307	FROPR1

Specify a percentage at which the cutting feedrate is to be changed when the drill is caused to move back as directed by an overload torque signal.

[Valid data range] 0 to 255%

0308	FROPR2

Specify a percentage at which the cutting feedrate is to be changed when the drill is caused to move back without receiving an overload torque signal.

[Valid data range] 0 to 255%

0309	FROLMT

Specify a lower limit to the percentage of the after–change cutting feedrate to the specified cutting feedrate.

[Valid data range] 0 to 255%

0327	PEKMCV

Specify a custom macro common variable number used to output the total number of times that the drill reverses during cutting. If 0 is specified, no output is made.

[Valid data range] 100 to 149

0328	SKPMCV

Specify a custom macro common variable number used to output the total number of times that the drill reverses during cutting as directed by an overload signal. If 0 is specified, no output is made.

[Valid data range] 100 to 149

0492	PEKFD1

Specify the speed at which the drill moves back to the R point if I is omitted.

[Valid data range] 1mm/min (mm input)
0.01inch/min (inch input)

0493	PEKFD2

Specify the speed at which the drill moves to the previous machined point if I is omitted.

[Valid data range] 1mm/min (mm input)
0.01inch.min (inch input)

0941	DLTPRM

Specify the first minute escape amount to be used in moving back to the R point during a cycle, and a clearance amount at the start point of the second and subsequent cutting cycles.

[Valid data range] Least input increment

Self–diagnosis

The following self–diagnosis information is displayed on the diagnosis screen. Parameter Nos. 520 and 521 are reset using a G83 command.

No. 520 : Total number of times the drill reverses during cutting after a G83 is issued.

No. 521 : Total number of times the drill reverses as directed by an overload signal during cutting after a G83 is issued.

No. 522 : Coordinate from which the drill began to reverse (in least input increments).

No. 523 : Difference between the previous coordinate from which the drill began to reverse and the current coordinate from which the drill began to reverse (in least input increments).

Note

NOTE

1 A small–diameter peck drilling cycle M code is issued in an M code–only block before a G83 is issued. The M code is processed internally; it will not be output.
2 The advance and reverse movements are accelerated/decelerated according to the cutting feedrate acceleration/deceleration time constant. For the reverse movement, a position check is performed at the R point.
3 When the drill is moving forward, if a skip signal is received, the drill is caused to move backward.
4 During a small–diameter peck drilling cycle, changing the spindle speed during pecking requires a 4– or 5–digit S command.
5 The R point must be separated by a specified clearance amount (Δ) from the workpiece plane.

11.9 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.0009 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:

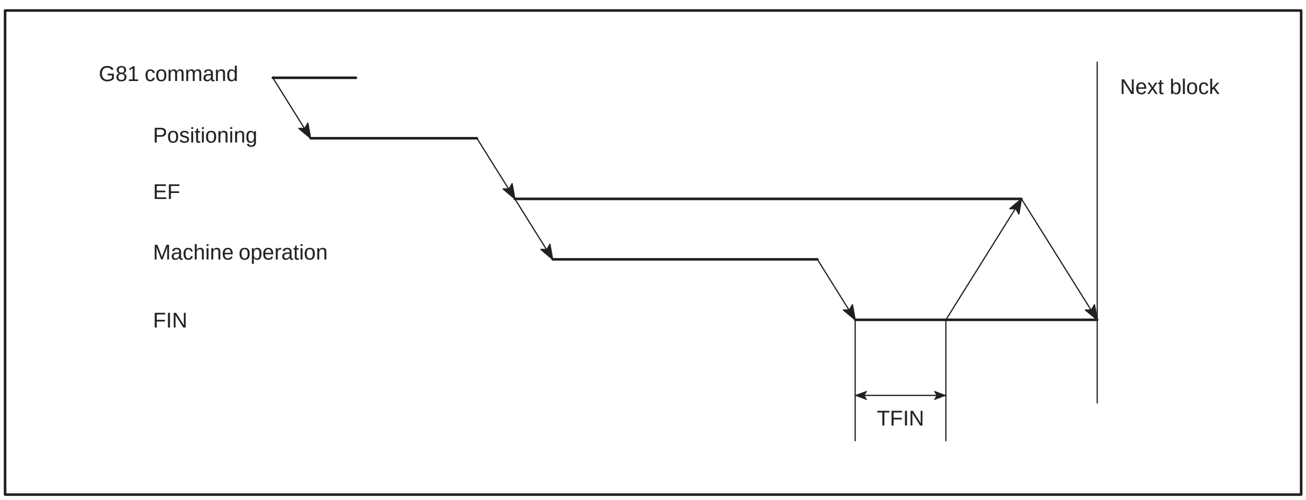


Fig. 11.9 Timing diagram of basic procedure

Signal

External Operation Signal EF<F150#1> (M series)

[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
F150							EF	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0011				MCF				

[Data type] Bit

MCF 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

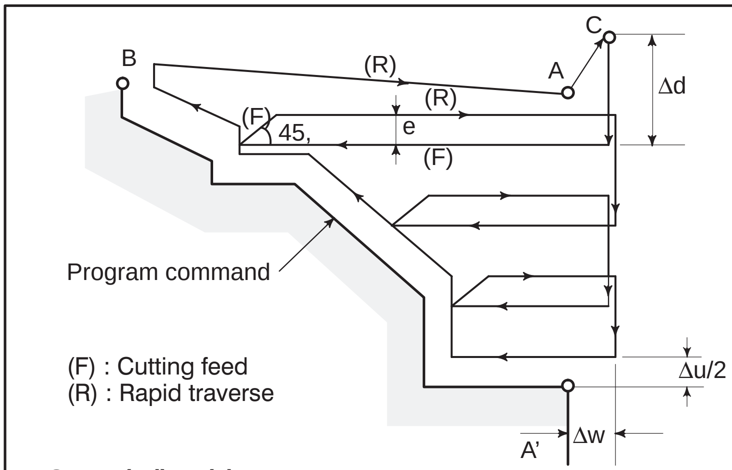
OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.13.8	EXTERNAL MOTION FUNCTION

11.10 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (M SERIES)

General

This option canned cycles to make 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 Δu/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.0717), 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.0718), 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<G126#7> (T Series)

[Classification] Input signal

[Function] Executes chamfering in a threading cycle. Specify the chamfering distance in parameter No.0109.

[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.

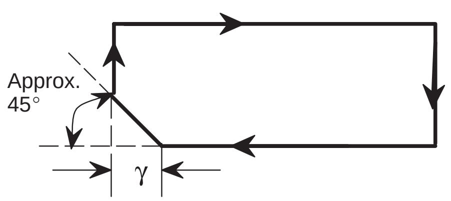


Fig.11.10 (a) Straight thread cutting cycle

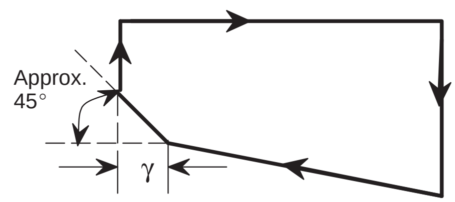


Fig.11.10 (b)Taper thread cutting cycle

Set the chamfering distance γ to the parameter No.0109. When the optional multiple repetitive canned cycle is provided, 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

F126	CDZ								(T series)

Parameter

- **Various setting for multiple repetitive canned cycle**

	#7	#6	#5	#4	#3	#2	#1	#0
0393				CHKMRC			MCQSCH	

[Data type] Bit

CHKMRC 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.

MCQSCH 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**

0109	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**

0717	Depth of cut in multiple repetitive canned cycles G71 and G72

[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 depth of cut in multiple repetitive canned cycles G71 and G72.

- **Escape in multiple repetitive canned cycles G71 and G72.**

0718	Escape in multiple repetitive canned cycles G71 and G72.

[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 escape in multiple repetitive canned cycles G71 and G72.

● Escape in multiple repetitive canned cycles G73

0719	Escape in multiple repetitive canned cycle G73 in X–axis direction
0720	Escape in multiple repetitive canned cycle G73 in Z–axis direction

[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

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

0721	Division count in multiple repetitive canned cycle G73

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the division count in multiple repetitive canned cycle G73.

● Return in multiple canned cycles G74 and G75

0722	Return in multiple canned cycles G74 and G75

[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 return in multiple repetitive canned cycles G74 and G75.

- **Repetition count of final finishing in multiple repetitive canned cycle G76**

0723	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**

0724	Tool nose angle in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 when FS15 tape format is used
0, 29, 30, 55, 60 and 80 when FS15 tape format is not used.

This parameter sets the tool nose angle in multiple repetitive canned cycle G76.

- **Minimum depth of cut in multiple repetitive canned cycle G76**

0725	Minimum depth of cut in 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 multiple repetitive canned cycle G76.

- **Finishing allowance in multiple repetitive canned cycle G76**

0726	Finishing allowance in 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 finishing allowance 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.

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.13.1 II.13.2	CANNED CYCLE MULTIPLE REPETITIVE CYCLE

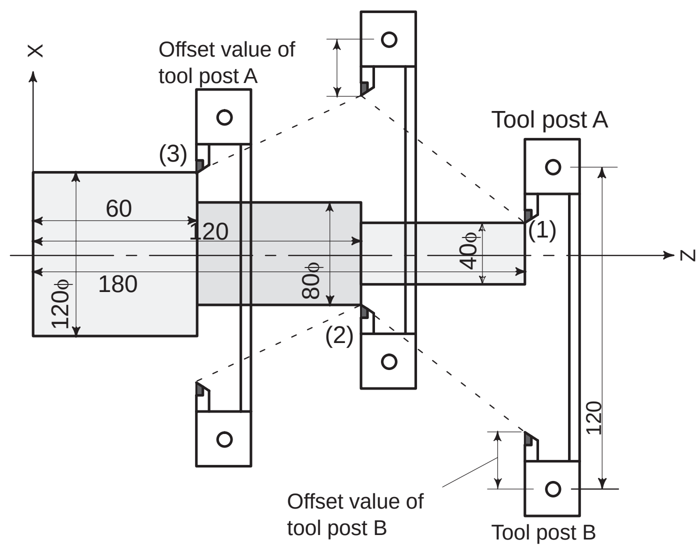
11.11 MIRROR IMAGE FOR DOUBLE TURRETS (T SERIES)

General

Mirror image can be applied to X–axis with G code.

G68 : Double turret mirror image on
G69 : Mirror image cancel

When G68 is designated, the coordinate system is shifted to the mating turret side, and the X–axis sign is reversed from the programmed command to perform symmetrical cutting. To use this function, set the distance between the two turrets to a parameter (No. 0730).
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**

0730	Distance between two turrets in 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 in mirror image.

Reference Item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.13.6	MIRROR IMAGE FOR DOUBLE TURRETS (G68, G69)

11.12 INDEX TABLE INDEXING FUNCTION (M SERIES)

General

By specifying indexing positions (angles) for the indexing axis (one rotation axis, A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped .

Basic Procedure

The control axis that indexes the index table can be named A, B or C. It will be referred to as "B" in the following discussion.

The positioning angle for the index table is commanded by the numerics following "B" in the program command, which is an independent block. Both absolute and incremental commands are possible, but the value after "B" is the integer times the numeric set by the parameter:

(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 and the difference between the variations are described below, followed by time charts showing them graphically.

(1) Assume Bbbbb is ordered by the command program.

(2) The CNC turns the B axis unclamp signal BUCLP <F188#2> to "1". (Type B -- When BUCLP is turned to "1", the position control servo for the B axis is turned ON.)

(3) On the PMC side, the clamp of the B axis is released; when completed, the B axis unclamp completion signal *BEUCP <G143#6> turns to "0".

(4) The CNC then turns the B axis unclamp signal BEUCP to "0", to indicate it received the *BEUCP signal.

(5) When the PMC is notified that BUCLP has been turned to "0", the PMC should turn *BEUCP to "1".
In type B, B–axis unclamp signal BUCLP is turned to "0", B–axis position control is made in servo–on state, 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, CNC turns B–axis clamp signal BCLP<F188#3>to 1. In type A, signal BCLP is set to "1" and B–axis position control is made in servo–off state.

(7) When BCLP is turned 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 <G143#7> is turned to "0".

(8) When *BECLP is turned to “0”, the CNC then turns BCLP to “0”, informing it received the *BECLP signal. (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 turned to “1”. This completes the sequence.

The time charts for these operations are shown in the figures below.

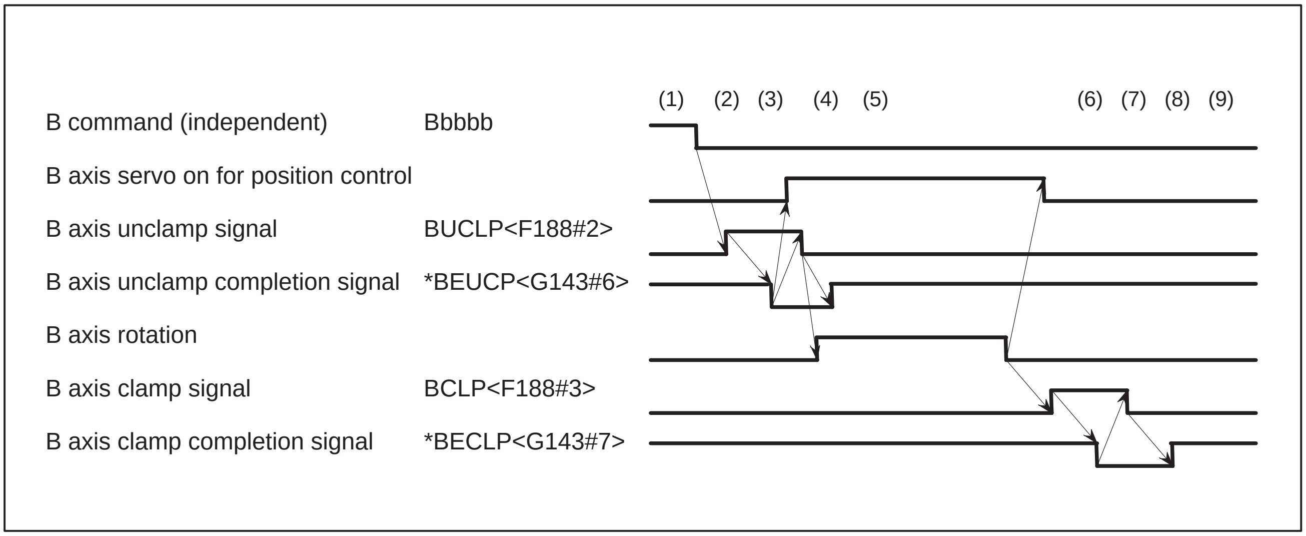


Fig. 11.12 (a) Time chart for positioning index table (type A)

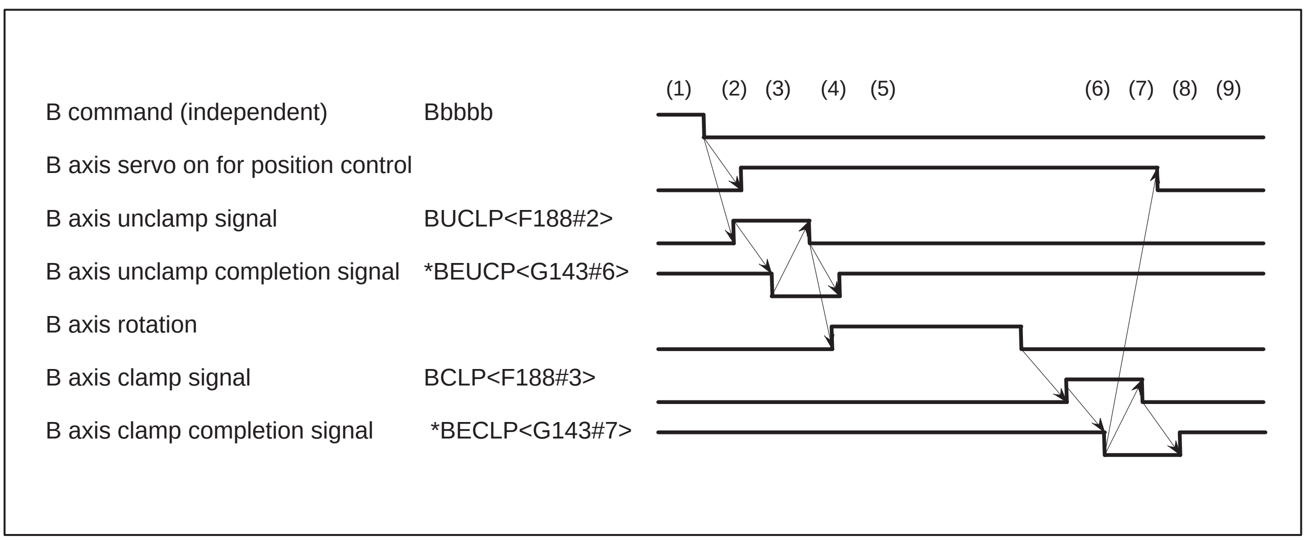


Fig. 11.12 (b) Time chart for positioning index table (type B)

Manual reference position return of B axis time chart (type A) is shown in the figure below.

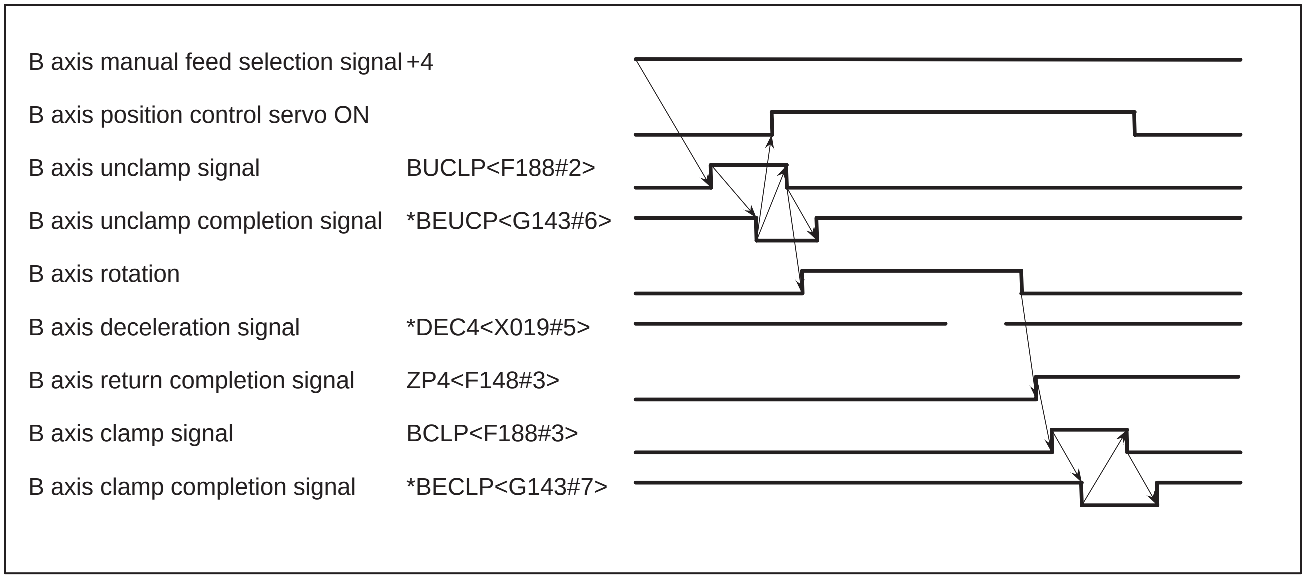


Fig. 11.12 (c) Manual reference position return of B axis time chart (type A)

Type A and Type B

As described in the basic procedure, type A differs from type B in that the servo used for B–axis position control is turned on or off at the different timing.
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. 0839. If an angle which is not a multiple of this minimum indexing angle is specified in indexing, alarm No. 135 is issued.

Direction of rotation

The direction of rotation can be set to one of the following.

– Whichever direction has the shorter distance (IXINC, bit 3 of parameter No. 0079)
– Direction specified with a command
– Usually the positive direction. Only when a particular M code is specified in the same block, the axis rotates in the negative direction (parameter No. 0249).

Absolute/incremental programming

Setting IXG90, bit 4 of parameter No. 0079, specifies absolute programming, irrespective of G90/G91 mode.

Signal

B axis clamp signal BCLP<F188#3>

[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<G143#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<F188#2>

[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<G143#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
G143	*BECLP	*BEUCP						

	#7	#6	#5	#4	#3	#2	#1	#0
F188					BCLP	BUCLP		

Parameter

- **Setting linear or rotation axis**

	#7	#6	#5	#4	#3	#2	#1	#0
0011						ADLN		

[Data type] Bit

ADLN 0 : 4th axis is rotation axis.
1 : 4th axis is linear axis.

- **Various setting for index table indexing**

	#7	#6	#5	#4	#3	#2	#1	#0
0079	IXTYP			IXG90	IXINC	IXABS	IXREL	IXDDP

[Data type] Bit

IXDDP Selection of decimal–point input method of index table indexing axis
0 : Conventional method (Example IS–B: B1; = 0.001 deg)
1 : Pocket calculator method (Example IS–B: B1; = 1.000 deg)

IXREL Relative position display of index table indexing axis
0 : Not rounded by 360 degrees
1 : Rounded by 360 degrees

IXABS Displaying absolute coordinate value of index table indexing axis
0 : Not rounded by 360 degrees
The index table indexing 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 table indexing axis is positioned in 40 degrees when G90 B400.0; is specified from the 0–degree position. The index table indexing 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.

IXINC Rotation in the G90 mode when negative–direction rotation command M code (parameter No. 0249) is not set
0 : Not set to the shorter way around the circumference
1 : Set to the shorter way around the circumference (Set IXABS, #2 of parameter No. 0079, to 1.)

IXG90 Index table indexing command
0 : Judged to be an absolute/increment command according to the G90/G91 mode
1 : Judged to be an absolute command

IXTYP Index table indexing sequence
0 : Type A
1 : Type B

- **Negative direction rotation command M code**

0249	Negative 0 direction rotation command M code

[Data type] Byte

[Valid data range] 0 to 255

0 : Not use an M code that sets the index table rotation to the negative direction. The rotation direction is specified using a command and parameter (IXINC, #3 of parameter No. 0079).

1 to 255:
Sets an M code that sets the index table rotation to the negative direction. The rotation is set to the negative direction only when an M code set here 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.

> **WARNING**
> Set IXABS, #2 of parameter No. 0079, to 1.

- **Unit of index table indexing angle**

0839	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. A P/S alarm is generated when movement other than integer multiple of the setting value is specified.

> **NOTE**
> If zero is specified as the setting value, any command can be specified irrespective of the unit of angle.

Alarm and Message

Number	Message	Description
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	ILLEGAL AXIS COMMAND	In index table indexing , another control axis was instructed together with the B axis. Modify the program.

Warning

WARNING

1 The secondary auxiliary function can be used if the address is 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 is ineffective during positioning of the B axis.
4 The machine lock signal MLK is functional during positioning of the B axis. However, while the B axis is moving, after the movement ends, the MLK is functional.

Note

NOTE

1 Specify a rotation axis as the index table indexing axis.
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, this operation 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 B axis, but manual reference position return is possible. If reset is applied during the movement of B axis, the manual reference position return operation is 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).

Reference Item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	II.14.10	INDEX TABLE INDEXING FUNCTION

11.13 SCALING (M SERIES)

General

A programmed figure can be magnified or reduced (scaling).
The dimensions specified with X_, Y_, and Z_ can each be scaled up or down with the same or different rates of magnification.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

- **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.
It depends on parameter SCR (No. 0036#07) which value is selected. If scaling P is not specified on the block of scaling (G51X_Y_Z_P_ ;), the scaling magnification set to parameter (No. 0731) is applicable. 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

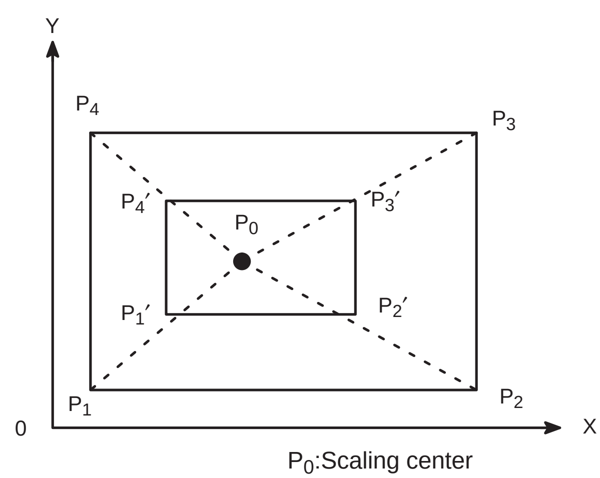


Fig. 11.13 (a) Scaling (P_1 P_2 P_3 P_4→P_1' P_2' P_3' P_4')

- **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 ESCAL (No.0063#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 0036#0 to #2) 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.0036#7)).
Magnification is set within the range ±0.00001 to ±9.99999 or ±0.001 to ±9.999.
If a negative value is set, mirror image is effected.
If magnification I, J or K is not commanded, a magnification value set to parameter (Nos. 0731 to 0733) is effective. However, a value other than 0 must be set to the parameter.

SCALING UP OR DOWN ALONG EACH AXES AT A DIFFERENT RATE OF MAGNIFICATION (MIRROR IMAGE)		
Format		Meaning of command
G51_X_Y_Z_I_J_K_;	Scaling start	**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
⋮	Scaling is effective. (Scaling mode)	
G50	Scaling cancel	

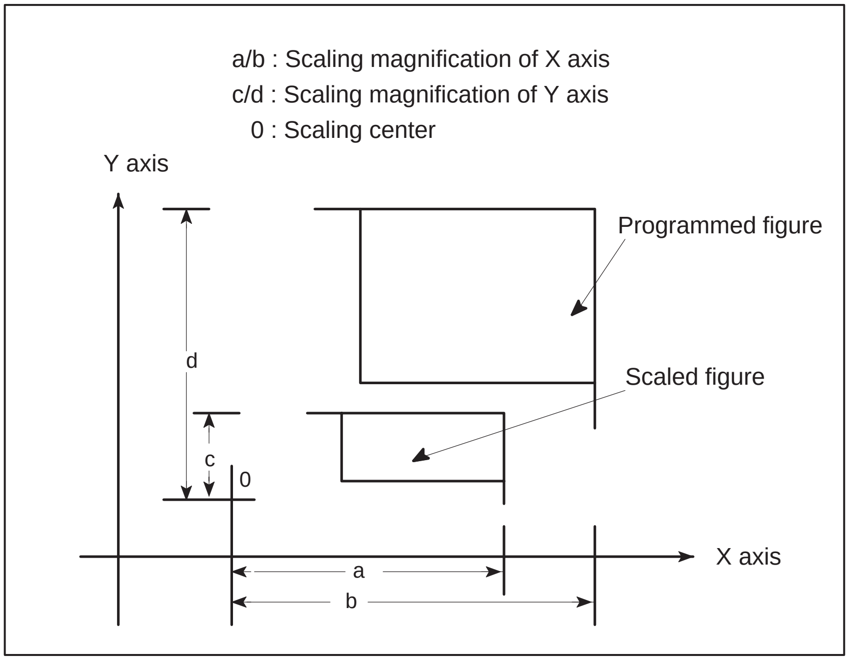


Fig. 11.13 (b) Scaling of each axis

Parameter

- **Setting valid/invalid and magnification of scaling**

	#7	#6	#5	#4	#3	#2	#1	#0
0036	SCR					SCLZ	SCLY	SCLX

[Data type] Bit

SCR Scaling magnification unit
0 : 0.00001 times (1/100,000)
1 : 0.001 times

SCLx Scaling for each axis
0 : Invalidated
1 : Validated

	#7	#6	#5	#4	#3	#2	#1	#0
0063		ESCAL						

[Data type] Bit

ESCAL Axis scaling and programmable mirror image
0 : Invalidated (The scaling magnification is specified by P.)
1 : Validated

- **Magnification used when scaling magnification is not specified**

0731	Magnification used when scaling magnification is not specified

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 0036)

[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.

> **WARNING**
> Parameter Nos.0731 to 0733 becomes valid when scaling for every axis is valid. (ESCAL, #6 of parameter No. 0063 is "1".)

- **Scaling magnification for each axis**

0731	Scaling magnification for X axis
to	
0733	Scaling magnification for Z axis

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 0036)

[Valid data range] –999999 to –1, 1 to 999999

This parameter sets the scaling magnification for each 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.
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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.14.8	SCALING (G50, G51)

11.14 COORDINATE SYSTEM ROTATION (M SERIES)

General

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

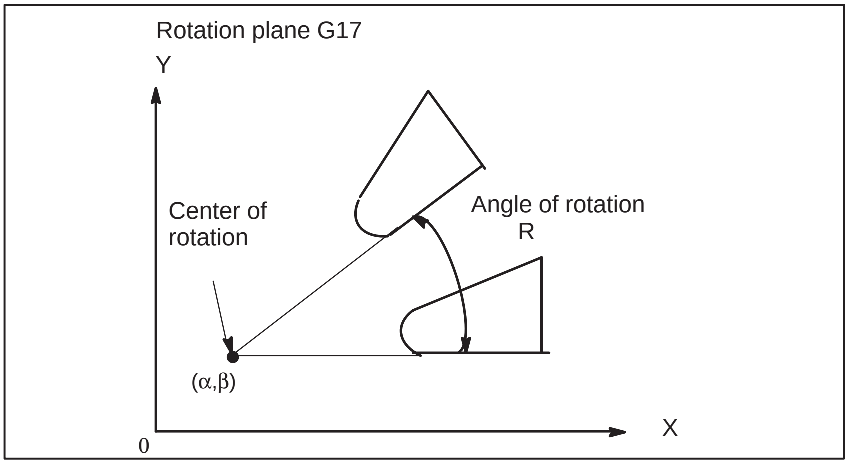


Fig. 11.14 Coordinate system rotation

FORMAT

$\left\{\begin{array}{l}\textbf{G17}\\ \textbf{G18}\\ \textbf{G19}\end{array}\right\}$ **G68** α_β_**R_** ; Start rotation of a coordinate system.

Coordinate system rotation mode
(The coordinate system is rotated.)

G69 ; Coordinate system rotation cancel command

MEANING OF COMMAND

G17 (G18 or G19) : Select the plane in which 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 by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.

R_ Angular displacement with a positive value indicates counter clockwise rotation. Parameter 0041#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

Parameter

- **Angle specification method of coordinate system rotation**

	#7	#6	#5	#4	#3	#2	#1	#0
0041								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**

0730	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 for coordinate system rotation is not specified with address R in the block where G68 is specified, the setting of this parameter is used as the angular displacement for coordinate system rotation.

Alarm and Message

Number	Message	Description
144	ILLEGAL PLANE SE-LECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.

Reference Item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.14.9	COORDINATE SYSTEM ROTATION (G68, G69)

11.15 MACRO COMPILER/ MACRO EXECUTER

General

There are two types of NC programs; those which, once created, are scarcely changed, and those which are changed for each machining type. The former are programs created by the custom macro, and the latter are machining programs. If programs of these types are executed simultaneously, a battery may run out or the custom macro may be destroyed by error operation.

Such problems can be solved by this function. The custom macro created by a machine tool builder is converted to an execute-form program, be stored in the ROM cassette, and be executed.

Features

(1) Since the program is stored after converted to an execute-form program, the execution speed is high. The machining time is then reduced, and the precision is improved.

(2) Since the program is stored in ROM cassette, there is no problem of battery extinction or custom macro destruction by error operation. The reliability is improved.

(3) Since the stored program is not displayed on a program screen, the know-how of the machine tool builder is protected.

(4) Since the custom macro is stored in ROM cassette, the program edit memory can be used efficiently.

(5) The user can call the macro easily without knowing the stored program. A custom macro can be created and executed in the program edit memory as usual.

(6) An original screen can be created by using the graphic display or selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machining program creation and edit control, reader/punch interface control, and PMC data read/write functions.

Note

NOTE

When the macro executor is attached, the order-made macro cannot be specified.

Reference Item

Macro compiler/executer programming manual (B–61393E–1)

12 DISPLAY/SETTING/EDIT

12.1 DISPLAY/SETTING

Signal

Relative coordinate update disable signal DLK<G127#6>

[Classification] Input signal

[Function] Setting this signal to 1 disables the updating of relative coordinates. Subsequently setting this signal to 0 causes the updating of relative coordinates to be resumed.

Ladder display signal *LDSP<G119#0>

[Classification] Input signal

[Function] This signal enables or disables the display of ladder diagrams.
*LDSP = 1: Does not display ladder diagrams.
*LDSP = 0: Displays ladder diagrams.

> **NOTE**
> This signal is enabled only when bit 2 (LDDSPG) of parameter No. 0060 is set to 1. When LDDSPG is 0, dynamic ladder display is disabled regardless of the state of this signal.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G119								*LDSP
G127		DLK						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0060						LDDSPG		

[Data type] Bit

LDDSPG Dynamic ladder display is:
1 : Performed.
0 : Not performed.

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. The time will be told through the window on the PMC side.

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 65535 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 an apparent decimal number. For example, March 28, 1995 is represented as 19950328.
#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

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.11.5.5	Displaying and Setting Run Time,Parts Count, and Time
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.11.5.3	Displaying and Setting Run Time,Parts Count, and Time

12.1.2 Servo Tuning Screen

General

On the servo tuning screen, parameters required for basic adjustment of the servo motor and statuses being monitored are listed for each axis.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0389								SRVSET

[Data type] Bit

SRVSET Servo tuning screen
0 : Displayed
1 : Not displayed

Reference item

MAINTENANCE MANUAL (B–61395E)	5.2	SERVO TUNING SCREEN

12.1.3 Spindle Tuning Screen

General

On the spindle tuning screen, parameters required for basic adjustment of the serial spindle and statuses being monitored are listed. The screen is only for the main spindle connected to the first amplifier.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0389							SPPRM	

[Data type] Bit type

SPPRM Spindle tuning screen
0 : Not displayed
1 : Displayed

Reference item

MAINTENANCE MANUAL (B–61395E)	6.4	SPINDLE TUNING SCREEN

12.1.4 Servo Waveform Display

General

Servo waveform display provides graphs of waveforms to illustrate changes in the following data.

a. Servo motor error value along each axis, number of distributed pulses and torque

b. On/off status of the machine signal, specified by a signal address

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0077						SGD		

[Data type] Bit

SGD Servo waveform
0 : Not displayed
1 : Displayed

Note

NOTE
Servo waveform display is enabled when bit 2 (SGD) of parameter No. 0077 is set to 1. To perform waveform diagnosis, a graphics board is required.

Reference item

MAINTENANCE MANUAL (B–61395E)	1.8	WAVEFORM DIAGNOSTIC DISPLAY

12.1.5 Self–diagnosis

General

When a breakdown occurs, in order to quickly determine the cause, the following should be done.
First, it has to be determined as to whether the breakdown occurred in the NC internal section, or the PMC or machine side.
There are times when it appears that a breakdown has occurred even when the breakdown has not actually occurred. For example, when the machinery ceases to operate because it is waiting for an external signal. In this case, the condition of the interface between the CNC and PMC, or between the CNC and the machinery, and the conditions within the CNC need to be investigated.
The NC checks the following itself.

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 CRT/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 from PMC to CNC, or vice versa, and inner status of the NC can be displayed on the CRT screen.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.7.2	CHECKING BY SELF–DIAGNOSTIC SCREEN
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.7.2	CHECKING BY SELF–DIAGNOSTIC SCREEN

12.1.6 Position Display Neglect

General

Indication of the current position can be suppressed by setting parameters. (except for basic axes of T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0029		DSPSUB					DSP4	DSP3	(T series)

[Data type] Bit

DSPx Specifies whether to display the current position of the third axis, as follows:
1 : Display
0 : Do not display

DSPSUB Fifth and Sixth axes in absolute and relative screens
0 : Display
1 : Do not display

	#7	#6	#5	#4	#3	#2	#1	#0	
0035					NDSP4	NDSP3	NDSPY	NDSPX	(M series)

[Data type] Bit

NDSPx Display of the current position for each axis
0 : The current position is displayed.
1 : The current position is not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
0061		DSP78						

[Data type] Bit

DSP78 Seventh and Eighth axes in absolute and relative screens
0 : Display
1 : Do not display

12.1.7 Run Hour and Parts Count Display

General

This function displays the integrated cycle operation time and the integrated cutting time on the CRT display screen. The integrated cycle operation time, the integrated cutting time can be altered and preset, using the MDI.

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 parts machined on the CRT 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 machined in memory 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 to change and preset the number of parts required and the number of parts machined using MDI.

Signal

Required parts count reached signal PRTSF<F164#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 0 (infinity) is set as the required number of parts, this signal is not output.

The PRTSF signal is set to 0 when:

- Machining of the specified number of parts has not yet been completed.
- The system is reset.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F164	PRTSF							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0040					RWCNT			

[Data type] Bit

RWCNT M code that counts the total number of machined parts and the number of machined parts

0 : M02, or M30, or an M code specified by parameter No.0219

1 : Only M code specified by parameter No.0219

0219	M code that counts the total number of machined parts and the number of machined parts

[Data type] Byte

[Valid data range] 0 to 255 except 98 and 99

The total number of machined parts and the number of machined parts are counted (+1) when the M code set is executed.

NOTE
Set value 0 is invalid (the number of parts is not counted for M00). Data 98 and 99 cannot be set.

0600	Number of required parts

[Data type] Word

[Unit of data] One piece

[Valid data range] 0 to 9999

This parameter sets the number of required machined parts.

Required parts finish signal PRTSF is output to 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.

0779	Total number of machined parts

[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.0219 is executed.

Reference item

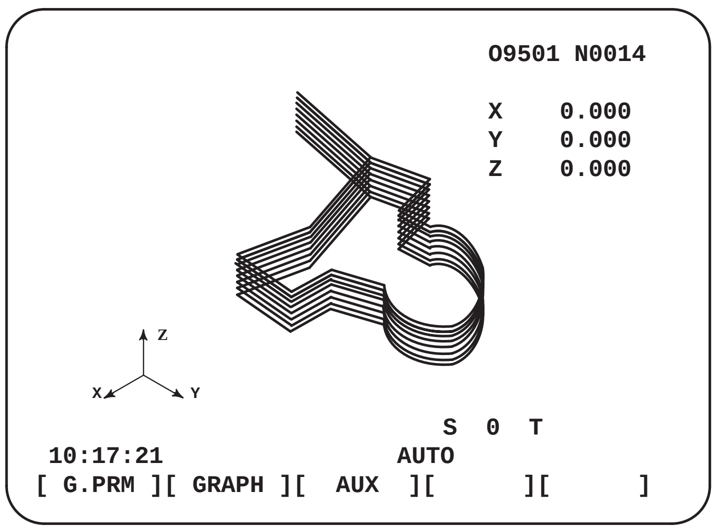
OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.11.5.5	Displaying and Setting Run Time, Parts Count, and Time
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.11.5.3	Displaying and Setting Run Time, Parts Count, and Time

12.1.8 Graphic Display/ Dynamic Graphic Display

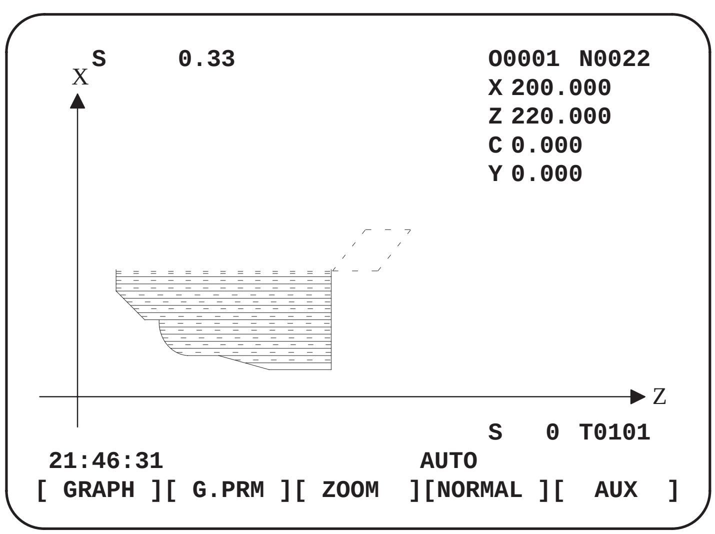
General

Graphic Display

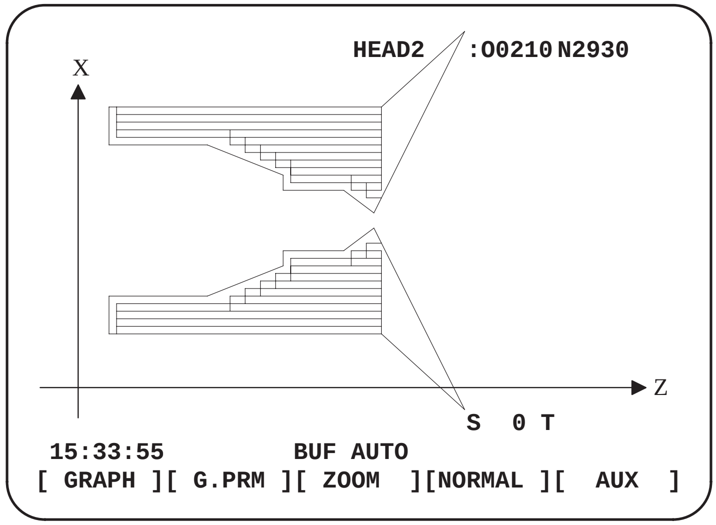
It is possible to draw the programmed tool path on the CRT screen, which makes it possible to check the progress of machining, while observing the path on the CRT screen.
In addition, it is also possible to enlarge/reduce the screen.
The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.
In the 0–TTC, the tool paths of two tool posts are displayed on the same screen, one on the right and the other on the left.



0–MC



0–TC



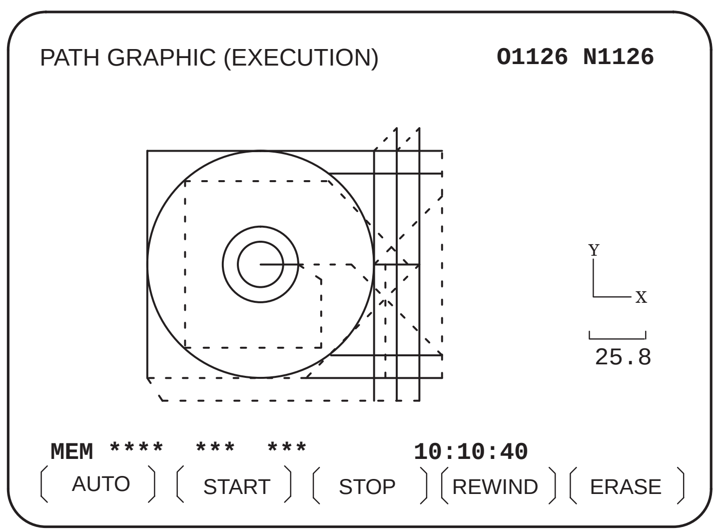
0–TTC

Dynamic graphic display (M series)

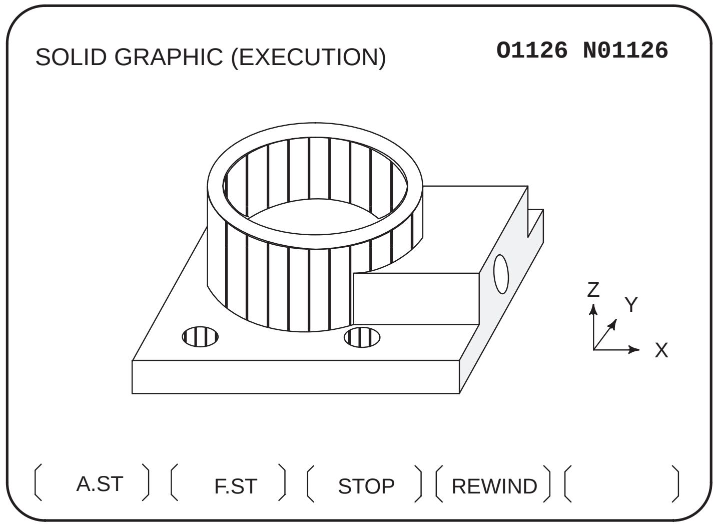
There are the following two functions in Dynamic Graphics.

Path graphic	This is used to draw the path of tool center commanded by the part program.
Solid graphic	This is used to draw the workpiece figure machined by tool movement commanded by the part program.

The path graphic function is used to precisely check the part program for drawing the tool path with a line. The solid graphic function is used to draw the workpiece figure to be machined with a program. Thus, it is easy to recognize roughly the part program. These two functions can be used freely by switching them.



Tool path Graph



Part Machined

Signal

Drawing signal CKGRP <F164#5>

[Classification] Output signal

[Function] Reports that a machining drawing is being created.

[Operation] The signal becomes 1 when:

- Creation of a machining drawing is started.

The signal becomes 0 when:

- Creation of a machining drawing is stopped.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F164			CKGRP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0024					GNSR				(T series)

[Data type] Bit

GNSR Current position display on the graphic display screen

0 : Displays the actual position to ensure tool nose radius compensation

1 : Displays the programmed position

	#7	#6	#5	#4	#3	#2	#1	#0
0046			GRPOS					

[Data type] Bit

GRPOS Current position on the graphic display screen
0 : Not appear
1 : Appears

	#7	#6	#5	#4	#3	#2	#1	#0	
0047							SP2C		(0–TTC)

[Data type] Bit

SP2C Graphic display (0–TTC) is done
0 : on two spindles and two tool posts
1 : on one spindle and two tool posts

	#7	#6	#5	#4	#3	#2	#1	#0	
0058			SGCSR	SGFIN	SGPLN	SG3PL	SGTLC	SGORG	(M series)

[Data type] Bit

SGORG 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)

SGTLC In solid drawing
0 : Not compensate the tool length
1 : Compensates the tool length

SG3PL Tri–plane drawing in solid drawing
0 : Drawn by the first angle projection
1 : Drawn by the third angle projection

SGPLN In solid drawing
0 : Draws a plane without edges.
1 : Draws a plane with edges.

SGFIN Machining profile drawing in solid drawing
0 : Displayed in the coarse mode
1 : Displayed in the fine mode

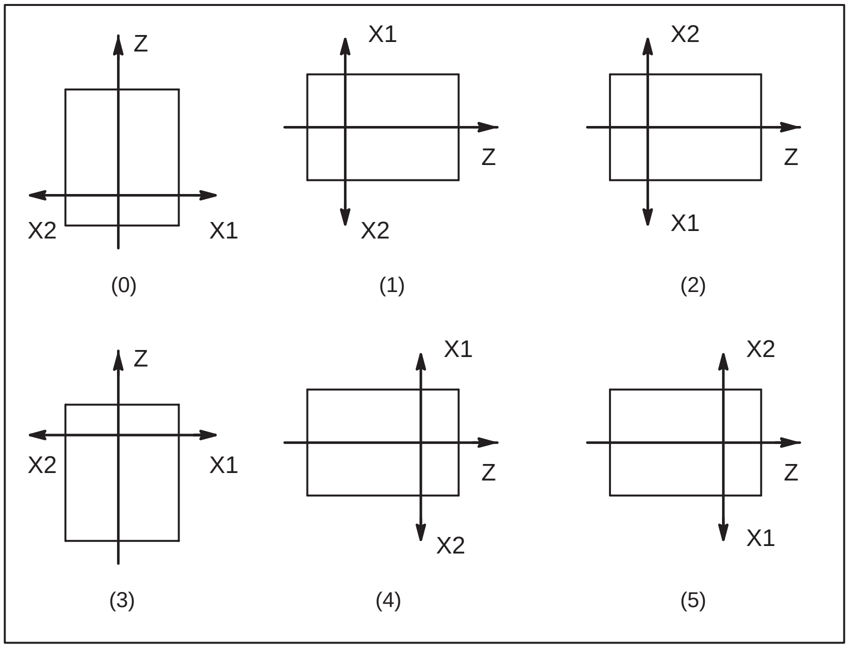
SGCSR While the screen image is enlarged, the shape of the graphic cursor is:
0 : A square. (■)
1 : An X. (×)

0123	Coordinate system for drawing	(0–TTC)

[Data type] Byte

[Valid data range] 0 to 5

The following shows the relationship between the settings and the drawing coordinate systems:



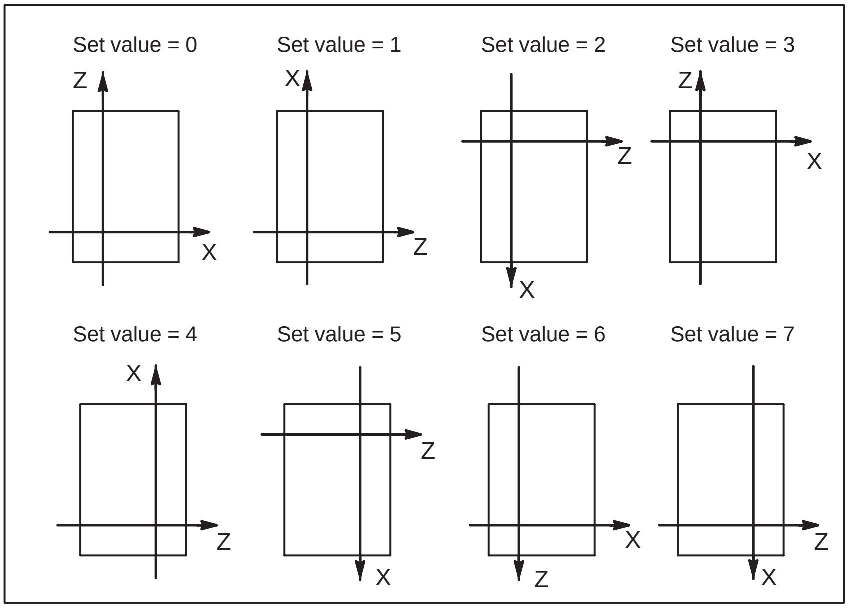
0123	Coordinate system for drawing	(T series)

[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 set values and the drawing coordinate systems.



NOTE
This parameter is specified for each tool post in the 0–TTC. A different drawing coordinate system can be selected for each tool post.

0253	Change in cross–section position in tri–plane drawing	(M series)

[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 set to 1.

0589	Right margin in solid drawing	(M series)
0590	Left margin in solid drawing	(M series)
0591	Upper margin in solid drawing	(M series)
0592	Lower margin in solid drawing	(M series)

[Data type] Word

[Unit of data] Dot

These parameters set the machining profile drawing position in margins on the CRT screen. The unit is a dot.

Parameter No.	Margin area	Standard set value			
		GRPOS (No.0046#5) is 0		GRPOS (No.0046#5) is 1	
		9” CRT	14” CRT	9” CRT	14” CRT
0589	Right	0	0	200	100
0590	Left	0	0	0	0
0591	Upper	25	32	25	32
0592	Lower	0	10	0	10

Note

NOTE
When the dynamic graphics function is used, the graphics function cannot be used. (M series)

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.12.1	GRAPHICS DISPLAY
	III.12.2	DYNAMIC GRAPHIC DISPLAY
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.12.1	GRAPHICS DISPLAY

12.1.9 Operating Monitor Display

General

The reading on the load meter can be displayed for each servo axis and the serial spindle.

- **Display of the servo axes**
  The reading on the load meter can be displayed for servo axes.
- **Display of the spindle axes**
  When serial spindles are used, the reading on the load meter and speedometer can be displayed.
- **Speedometer**
  Although the speedometer normally indicates the speed of the spindle motor.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0060			OPMNDP					

[Data type] Bit

OPMNDP Operating monitor
0 : Not displayed
1 : Displayed

6627	Load meter displayed value for maximum output

[Data type] Word

8x86	Rated current parameter (RTCURR)

[Data type] Word

Note

> **NOTE**
> The reading on the load meter depends on servo parameter 8x86 and spindle parameter 6627.
> These parameters are set by the automatic setting.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.11.1.6	Operating Monitor Display
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.11.1.6	Operating Monitor Display

12.1.10 Software Operator's Panel

General

The software operator's panel function replaces part of the control switches on the machine operator's panel with soft switches which can be turned on or off using the CRT/MDI of the control unit.

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 used additionally 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.0017) can be used to select whether the control switches on the machine operator's panel or soft switches on the CRT/MDI of the control unit are used for each group.

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, 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 informed to the PMC by output signals. Based on these output signals, the PMC should turn "1" or "0" input signals related to soft switch functions. In other words, turning "1" the soft switch assigned to single block operation, for example, does not cause the control unit to select single block operation internally. Single block operation is selected when the PMC turns to "1" the input signal for single block operation, instead.

Signal

Group	Function	Output signal	Related input signal
1	Mode selection	MD1O<F174#0> MD2O<F174#1> MD4O<F174#2> ZRNO<F174#3>	MD1 MD2 MD4 ZRN
2	Jog feed axis select	+XO to +4O –XO to –4O <F177>	+X to +4 –X to –4
	Manual rapid traverse	RTO<F178#6>	RT
3	Handle feed	HXO to H4O <F174#4 to #7>	HX HY HZ H4
	Handle feed magnification	MP1O<F175#6> MP2O<F175#7>	MP1 MP2
4	Jog feed rate override	JV1O to JV8O <F175#0 to 3>	*OV1 to *OV8
	Feed rate override	OV1O to OV8O <F176#0 to 3>	*OV1 to *OV8
	Rapid traverse override	ROV1O<F175#4> ROV2O<F175#5>	ROV1 ROV2
5	Optional block skip	BDTO<F176#4>	BDT
	Single block	SBKO<F176#5>	SBK
	Machine lock	MLKO<F176#6>	MLK
	Dryrun	DRNO<F176#7>	DRN
6	Protect key	KEYO<F178#5>	KEY
7	Feed hold	SPO<F178#7>	*SP
8	General purpose (Switch from 1st line to the 8th line on CRT)	OUT0 to OUT7 <F171>	

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
F171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0	
F174	H4O	H3O	HZO	HXO	ZRNO	MD4O	MD2O	MD1O	(T series)
F174	H4O	HZO	HYO	HXO	ZRNO	MD4O	MD2O	MD1O	(M series)
F175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O	
F176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O	
F177	–4O	+4O	–3O	+3O	–ZO	+ZO	–XO	+XO	(T series)
F177	–4O	+4O	–ZO	+ZO	–YO	+YO	–XO	+XO	(M series)
F178	SPO	RTO	KEYO						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0017		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Data type] Bit

OP1 Mode selection on software operator’s panel
0 : Not performed
1 : Performed

OP2 JOG feed axis select and manual continous rapid traverse buttons on software operator’s panel
0 : Not performed
1 : Performed

OP3 Manual pulse generator’s axis select and manual pulse generator’s magnification switches on software operator’s panel
0 : Not performed
1 : Performed

OP4 Jog feedrate override, feedrate override, and rapid traverse override switches on software operator’s panel
0 : Not performed
1 : Performed

OP5 Optional block skip, single block, machine lock, and dry run switches on software operator’s panel
0 : Not performed
1 : Performed

OP6 Protect key on software operator’s panel
0 : Not performed
1 : Performed

OP7 Feed hold on software operator’s panel
0 : Not performed
1 : Performed

0130	Jog–movement axis and its direction on software operator’s panel [↑]
0131	Jog–movement axis and its direction on software operator’s panel [↓]
0132	Jog–movement axis and its direction on software operator’s panel [→]
0133	Jog–movement axis and its direction on software operator’s panel [←]
0134	Jog–movement axis and its direction on software operator’s panel [↙]
0135	Jog–movement axis and its direction on software operator’s panel [↗]
0136	Jog–movement axis and its direction on software operator’s panel [⤴]
0137	Jog–movement axis and its direction on software operator’s panel [↷]

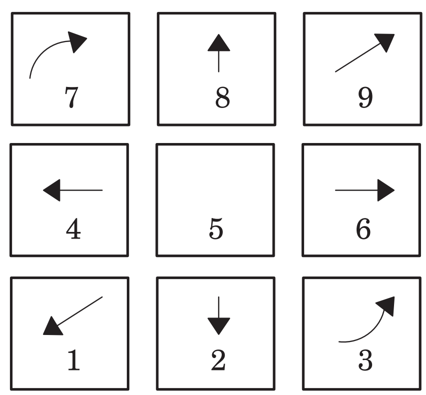
[Data type] Byte

[Valid data range] 0 to 8

On software operator’s panel, set a feed axis corresponding to an arrow key on the CRT/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 CRT/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.0130 = 5 (Z axis, positive direction)
Parameter No.0131 = 6 (Z axis, negative direction)
Parameter No.0132 = 1 (X axis, positive direction)
Parameter No.0133 = 2 (X axis, negative direction)
Parameter No.0134 = 3 (Y axis, positive direction)
Parameter No.0135 = 4 (Y axis, negative direction)
Parameter No.0136 = 0 (Not used)
Parameter No.0137 = 0 (Not used)

0140	Name of general–purpose switch on software operator's panel
⋮	⋮
0203	Name of general–purpose switch on software operator's panel

[Data type] Byte

(Exapmle)

These parameters set the names of the general–purpose switches (SIGNAL 1 through SIGNAL 8) on the software operator's panel as described below.

```
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. 0140 to 0203.

Parameter No.0140:
Sets the character code (083) corresponding to S of SIGNAL 1.

Parameter No.0141:
Sets the character code (073) corresponding to I of SIGNAL 1.

Parameter No.0142:
Sets the character code (071) corresponding to G of SIGNAL 1.

Parameter No.0143:
Sets the character code (078) corresponding to N of SIGNAL 1.

Parameter No.0144:
Sets the character code (065) corresponding to A of SIGNAL 1.

Parameter No.0145:
Sets the character code (076) corresponding to L of SIGNAL 1.

Parameter No.0146:
Sets the character code (032) corresponding to (space) of SIGNAL 1.

Parameter No.0147:
Sets the character code (049) corresponding to 1 of SIGNAL 1.

Parameter Nos.0148 to 0155:
Set the character codes of SIGNAL 2 shown in the figure above.

Parameter Nos.0156 to 0163:
Set the character codes of SIGNAL 3 shown in the figure above.

Parameter Nos.0164 to 0171:
Set the character codes of SIGNAL 4 shown in the figure above.

Parameter Nos.0172 to 0179:
Set the character codes of SIGNAL 5 shown in the figure above.

Parameter Nos.0180 to 0187:
Set the character codes of SIGNAL 6 shown in the figure above.

Parameter Nos.0188 to 0195:
Set the character codes of SIGNAL 7 shown in the figure above.

Parameter Nos.0196 to 0203:
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

Char-acter	Code	Comment	Char-acter	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	Sharpe
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	
3	051		¥	093	Yen mark
4	052		]	094	Right square bracket
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 equipped, for example, 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 When the soft switch for feed hold is turned on, output signal SPO is turned to “1”. At this time, 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 informing the state of a soft switch is turned to “1”, the corresponding input signal is turned to “1”.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.11.6.2	Displaying and Setting the Software Operator’s Panel
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.11.6.2	Displaying and Setting the Software Operator’s Panel

12.1.11 Multi–language Display

General

The CRT or LCD screens are displayed in a parameter–set language.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0023		DSPN	DHNG	DITA	DCHI	DFRN	DGRM	DJPN

[Data type] Bit type

NOTE
When this parameter is set, turn off the power once.

The language used in the display on the CRT is selected.

DSPN	DHNG	DITA	DCHI	DFRN	DGRM	DJPN	CRT display language
0	0	0	0	0	0	0	English
0	0	0	0	0	0	1	Japanese
0	0	0	0	0	1	0	German
0	0	0	0	1	0	0	French
0	0	0	1	0	0	0	Chinese (Taiwanese)
0	0	1	0	0	0	0	Italian
0	1	0	0	0	0	0	Hangul
1	0	0	0	0	0	0	Spanish

12.2 EDIT

12.2.1 Part Program Storage Length

General

One of the following part program length.

M series (m)	10	20	40	80	120	320
T series (m)	10	20	40	80	120	320

Alarm and message

Number	Message	Description
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.11.3.1	Displaying Memory Used and a List of Programs
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.11.3.1	Displaying Memory Used and a List of Programs

12.2.2 No. of Registered Programs

General

One of the following no. of registered programs can be selected.
63/125/200.

Alarm and message

Number	Message	Description
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), or 200 (option). Delete unnecessary programs and execute program registeration again.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.11.3.1	Displaying Memory Used and a List of Programs
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 from being registered or deleted erroneously.

Signal

Memory protection signal KEY<G122#3>

[Classification] Input signal

[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
G122					KEY			

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.11	SETTING AND DISPLAYING DATA
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.11	SETTING AND DISPLAYING DATA

12.2.4 Password Function

General

The password function locks PRG9 (bit 4 of parameter No.0010), used to protect program Nos. 9000 to 9999, by using the PASSWD (No.0797) and KEYWD (No.0798) parameters. When PRG9 is locked, PRG9 cannot be set to 0. Therefore, the protection for programs numbered 9000 to 9999 cannot be released unless the correct keyword is entered.

PRG9 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.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0010				PRG9				

[Data type] Bit

PRG9 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

0797	Password(PASSWD)

[Data type] Two–word

Set a secret number to this parameter. Its value is not displayed.

CAUTION
Once a key is lock, parameter PRG9 cannot become 0 and PASSWD cannot be changed unless you perform an unlock operation or perform the memory all clear operation. Special care should be exercised when setting the PASSWD parameter.

0798	Keyword(KEYWD)

[Data type] Two–word

When the value set as the password (set in parameter No. 0797) is set to this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (PRG9) of parameter No. 0010 becomes 0.

> **NOTE**
> The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

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, and 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.

> **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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.9.7	BACKGROUND EDITING
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.9.7	BACKGROUND EDITING

12.2.6 Playback

General

When the playback option is selected, the **TEACH IN JOG** mode (TJOG) and **TEACH IN HANDLE** mode (THND) are added. In these modes, 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
0002		TJHD						

[Data type] Bit

TJHD Manual pulse generator in TEACH IN JOG mode
0 : Valid
1 : Invalid

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.10.5	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.10.5	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

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. Custom macro common variables.
5. PMC D area data

Parameter

This CNC has four channels of input/output device interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel 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.

Input/output channel number
↓

I/O

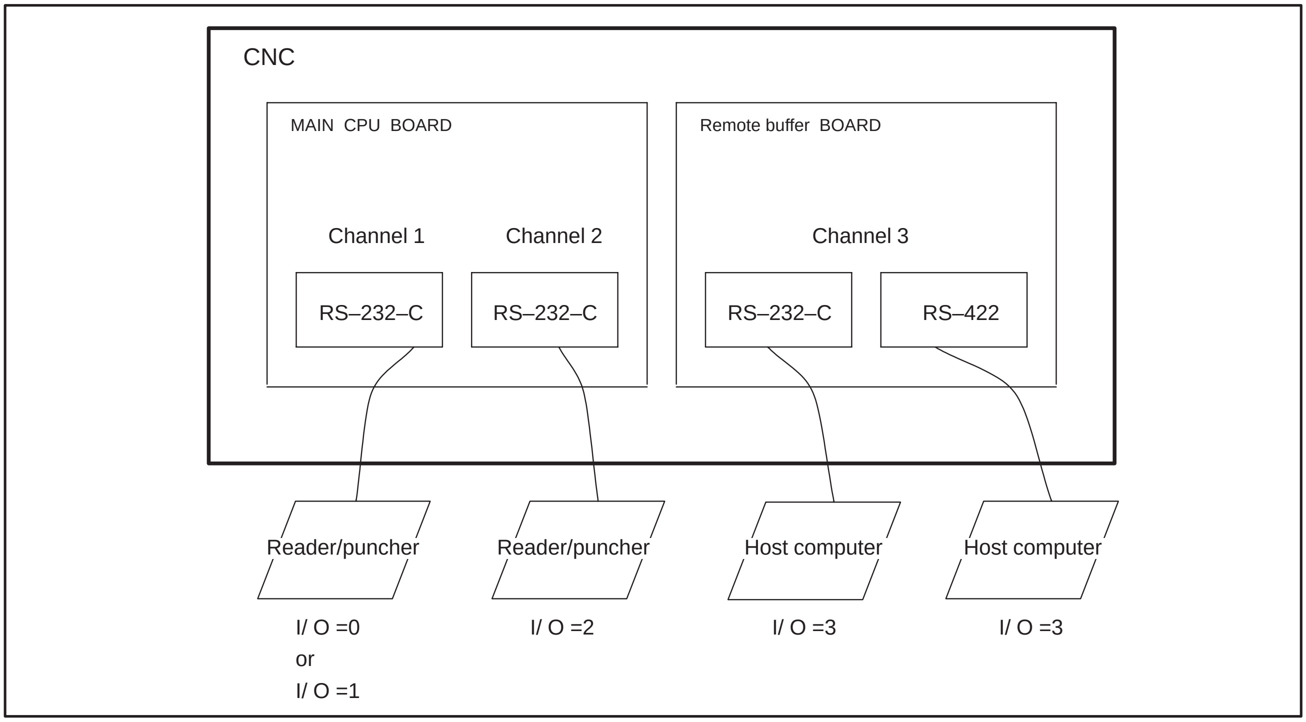
Specify a channel for an input/output device.
I/O =0 : Channel 1
=1 : Channel 1
=2 : Channel 2
=3 : Channel 3

I/O =0 (channel 1)	0002#0	Stop bit and other data
	0038#6, #7	Number specified for the input/output device
	0552	Baud rate
I/O =1 (channel 1)	0012#0	Stop bit and other data
	0038#6, #7	Number specified for the input/output device
	0553	Baud rate
I/O =2 (channel 2)	0050#0	Stop bit and other data
	0038#4, #5	Number specified for the input/output device
	0250	Baud rate
I/O =3 (channel 3) * Remote buffer	0051#0	Stop bit and other data
	0038#1, #2	Number specified for the input/output device
	0251	Baud rate
	0055#2	Selection of protocol
	0055#3	Selection of RS–422 or RS–232C, and other data

Setting entry

TVON TV check
0 : Not performed
1 : Performed

ISO Code used for data output
0 : EIA code
1 : ISO code



(1) Parameters common to all channels

	#7	#6	#5	#4	#3	#2	#1	#0
0018		TVC						

[Data type] Bit

TVC Character counting for TV check in the comment section of a program.
0 : Performed
1 : Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
0070	ICR							

[Data type] Bit

ICR Output of the end of block (EOB) in ISO code
0 : LF, CR, CR are output.
1 : Only LF is output.

	#7	#6	#5	#4	#3	#2	#1	#0
0075	IONUL							

[Data type] Bit

IONUL Action taken when a NULL code is found during read of EIA code
0 : An alarm is generated.
1: The NULL code is ignored.

(2) Parameters for channel 1 (I/O =0)

	#7	#6	#5	#4	#3	#2	#1	#0
0002	NFED				RSASCI			STP2

[Data type] Bit

STP2 The number of stop bits
0 : 1
1 : 2

RSASCI Code used at data input
0 : EIA or ISO code (automatically distinguished)
1 : ASCII code

NFED 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 NFED to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
0038	RSCMD1	DEVFL1	RSCMD2	DEVFL2		RSCMD3	DEVFL3	

[Data type] Bit

Set the specification of the input/output device for I/O = 0, using the values shown in Table 13.1 (a).

Table 13.1 (a)

(I/O=0, 1, or 2)

RSCMD*	DEVFL*	I/O device used
0	0	Bubble cassette
0	1	Floppy cassette
1	0	RS232C, PPR
1	1	New type interface

(I/O=3)

RSCMD3	DEVFL3	I/O device used
0	0	Bubble cassette
0	1	Floppy cassette
1	0	Portable tape reader, etc.
1	1	Portable tape reader, etc.

0552	Baud rate (when the I/O is set to 0)

[Data type] Byte

Set baud rate of the input/output device used when the I/O 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)

* : I/O channel 3 or 2

(3) Parameters for channel 1 (I/O =1)

	#7	#6	#5	#4	#3	#2	#1	#0
0012	NFED				RSASCI			STP2

[Data type] Bit

These parameters are used when I/O is set to 1. The meanings of the bits are the same as for parameter 0002.

	#7	#6	#5	#4	#3	#2	#1	#0
0038	RSCMD1	DEVFL1						

[Data type] Bit

Set the number specified for the input/output device used when the I/O is set to 1, with one of the set values listed in Table 13.1 (a).

0553	Baud rate (when I/O 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 for channel 2 (I/O =2)

	#7	#6	#5	#4	#3	#2	#1	#0
0038			RSCMD2	DEVFL2				

[Data type] Bit

Set the number specified for the input/output device used when I/O is set to 2, with a value in Table 13.1 (a).

	#7	#6	#5	#4	#3	#2	#1	#0
0050	NFED				RSASCI			STP2

[Data type] Bit

These parameters are used when I/O is set to 2. The meanings of the bits are the same as for parameter 0002.

0250	Baud rate (when the I/O 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).

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.

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	III.8	DATA INPUT/OUTPUT
OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.8	DATA INPUT/OUTPUT

13.2 REMOTE BUFFER

Refer to Descriptions (B–61392EN–1) for Remote Buffer for detailed informetion of remote buffer.

13.3 DNC1 INTERFACE

General

Refer to FANUC DNC1 DESCRIPTIONS(B–61782E) for detailed information of DNC1 interface.

Parameter

- Setting entry
  An input/output unit is selected at I/O on the setting screen.
  Setting value 10

0251	Baud rate

[Data type] Byte

The baud rate of HDLC is fixed to 460 kbps for DNC1.
Set following value:

Set value. :51

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

0347	System for connection between the CNC and host (DNC1 interface)

[Data type] Byte

[Valid data range] 1 or 2

This parameter specifies the system for connection (DNC1 interface) between the CNC and host.

Set value

1 : Point–to–point connection

2 : Multipoint connection

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

0348	Station address of the CNC (DNC1 interface)

[Data type] Byte

[Valid data range] 2 to 52

This parameter specifies the station address of the CNC when the CNC is connected via the DNC1 interface using multipoint connection.

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

MAP parameter description

(SETTING)

DNC FILE SELECTION

The parameter is used to specify a file name for a part program used in DNC operation.

Format: OXXXX.PRG (where XXXX is a four–digit number)

Setting method

(1) Full keypad

1. Press the rightmost soft key [▷].
   * Skip this step if the [STRING] soft key has already appeared.
2. Press the [STRING] soft key.
3. Specify a file name using the MDI panel keys.
4. Assert the entry using the [INPUT] key.

   (Example) To specify O0100:

Key in ”O0100.PRG” using the MDI panel keys, then press [INPUT].

(2) Standard keypad

1. Press the rightmost soft key [▷].
   * Skip this step if the [CODE] soft key has already appeared.
2. Press the [CODE] soft key.
   (SETTING)
   DNC FILE SELECTION
   _
   ↑
   The cursor moves to here.
3. Specify a file name in ASCII code using numeric keys.
4. Assert the entry using the [INPUT] key.
5. Pressing [▷] again moves the cursor back to the previous position and cancels the code entry.

```
MAP PARAMETER                    O0001 N0001

 (SETTING)

DNC FILE SELECTION
    __

NUM.
                         MDI
[ PARAM  ][ DGNOS  ][  MAP  ][      ][      ]
[ STRING ][  CODE ][ CLEAR ][VRFY.H][ VRFY ]
[       ][       ][       ][INS.CH][ DEL.CH ]
```

← [STRING] is not displayed with the standard keypad.

NC APPLICATION NAME
NC APPLICATION PASS WORD
MAX MESSAGE LENGTH
MAX INVOKES
DNC1 does not use these items.

```
MAP PARAMETER                    O0001 N0001

 NC APPLICATION NAME


 NC APPLICATION PASS WORD
                                         0

 MAX MESSAGE LENGTH
                                         0
 MAX INVOKES


NUM.
                         MDI
[ PARAM ][ DGNOS ][  MAP  ][       ][       ]
```

HOST APPLICATION NAME
HOST APPLICATION INSTACE
HOST APPLICATION PASSWORD
DNC1 does not use these items.

```
MAP PARAMETER                     O0001 N0001

  HOST APPLICATION NAME


  HOST APPLICATION INSTACE
                                           0

  HOST APPLICATION PASSWORD




NUM.
                         MDI
[ PARAM ][ DGNOS ][  MAP  ][       ][       ]
```

INFORMATION REPORT ENABLE

RISING EDGE 00000000 00000000
FALLING EDGE 00000000 00000000

With these parameters, it is necessary to set the bit information needed when the status information in the CNC is sent to DNC1 at a local request.

The CNC status information consists of the following bit pattern, and corresponds to the RISING EDGE and FALLING EDGE bit parameters. If the bit is 0, it specifies that the information be masked. If the bit is 1, it specifies that the information be sent to DNC1.

RISING EDGE means that when the status information bit changes from 0 to 1, the change is reported.

FALLING EDGE means that when the status information bit changes from 1 to 0, the change is reported.

CNC status information bit pattern configuration

15	14	13	12	11	10	09	08

07	06	05	04	03	02	01	00

Bit position:signal name	Description
00 : RWD	Rewind signal
01 : AL	Alarm output signal
02 : RST	Reset in progress signal
03 : SPL	Automatic operation at pause signal
04 : STL	Automatic operation being started signal
05 : OP	Automatic operation in progress signal
06 : SA	Servo ready
07 : MA	CNC ready
08 :	Not used
09 :	Not used

10 :	Not used
11 :	Not used
12 : M00	M00 decode output signal (*)
13 : M01	M01 decode output signal (*)
14 : M02	M02 decode output signal (*)
15 : M30	M30 decode output signal (*)

* Signals for the 0–TTC
#08 to #11 correspond to M00 to M30 for HEAD2.
#12 to #15 correspond to M00 to M30 for HEAD1.

```
MAP PARAMETER                      O0001 N0001

  INFORMATION REPORT ENABLE
     RISING  EDGE   00000000  00000000
     FALLING EDGE   00000000  00000000

  ALARM NOTIFICATION ENABLE
     UPPER WORD     00000000  00000000
     LOWER WORD     00000000  00000000




NUM.
                                 MDI
[ PARAM ][ DGNOS ][  MAP  ][       ][       ]
```

ALARM NOTIFICATION ENABLE

UPPER WORD	00000000	00000000
LOWER WORD	00000000	00000000

The parameter is used to specify an alarm type that causes the CNC to inform the host of the CNC status change that is induced by the alarm.

Setting value

0: The occurrence of the alarm does not trigger notification by the CNC.

1: The occurrence of the alarm triggers notification by the CNC.

The parameter bits correspond to each alarm type as listed below.

Upper word bit parameter

15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00

Bit position	Alarm type
01 :	P/S alarm
02 :	Overheat alarm
05 :	P/S 100 alarm
06 :	Overtravel
12 :	Servo alarm
13 :	P/S 101 alarm
14 :	P/S 000 alarm

Lower word bit parameter

15	14	13	12	11	10	09	08

07	06	05	04	03	02	01	00

00 : Battery alarm

MAC PRINT FILE → Not used by DNC1.

LOGGING MODE → Not used by DNC1.
00000000 00000000

MAP SERVICE MODE → Not used by DNC1.
00000000 00000000

NC SERVICE MODE → Not used by DNC1.
00000000 00000000

This parameter specifies the behavior of the all–file directory information read function of the CNC as follows:

00000000 00000001 : Only the file number is read.

00000000 00000001 : Both the file number and size are read.

```
MAP PARAMETER                      O0001 N0001

  MAC PRINT FILE


  LOGGING MODE
                      00000000  00000000
  MAP SERVIEC MODE
                      00000000  00000000
  NC SERVIEC MODE
                      00000000  00000000


NUM.
                                 MDI
[ PARAM ][ DGNOS ][  MAP  ][       ][       ]
```

MAP LOG MESSAGE screen

```
MAP LOG MESSAGE                      O0001 N0001










NUM.
                                 MDI
[ ALARM ][ MSG ][MAPMSG][       ][       ]
```

Not used with the DNC1 function

13.4 DNC2 INTERFACE

Refer to an item of FANUC DNC2 DESCRIPTIONS (B–61992E) for detailed information of DNC2 interface.

13.5 EXTERNAL I/O DEVICE CONTROL

General

It is possible to request from the outside 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 onto tape 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<G134#1>

[Classification] Input signal

[Function] Programs are registered through the reader/punch interface or remote buffer. Or the 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 KEY<G122#3> determines whether to register or collate.)
- Bit 1 (RAL) of parameter No.0045 selects whether to register all programs in a file or one program at a time. Bit 0 (RDL) of parameter No.0045 can be used to delete all programs previously stored in the part program memory. However, it is impossible to delete programs protected by bit 4 (PRG9) of parameter No.0010 and bit 2 (PRG8) of parameter No.0389.
- 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/punch 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<G134#3>

[Classification] Input signal

[Function] Programs stored in the part program memory are output to an external unit via the reader/punch 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/punch 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 bit 4 (PRG9) of parameter No.0010 and bit 2 (PRG8) of parameter No.0389.

External read/punch stop signal EXSTP<G134#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/punch interface and program registeration and collation via the remote buffer.

[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<F180#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.
- When the uploading or downloading of a program from DNC1, DNC2, or MMC is started.

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).
- When the uploading or downloading of a program from DNC1, DNC2, or MMC is completed.

Read/punch busy signal RPBSY<F180#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<F180#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
G134					EXWT	EXSTP	EXRD	

	#7	#6	#5	#4	#3	#2	#1	#0
F180				BGEACT	RPALM	RPBSY		

Parameter

Input/output channel number
↓

I/O

Specify a channel for an input/output device.

I/O =0 : Channel 1
=1 : Channel 1
=2 : Channel 2
=3 : Channel 3

I/O =0 (channel 1)	0002#0	Stop bit and other data
	0038#6, #7	Number specified for the input/output device
	0552	Baud rate
I/O =1 (channel 1)	0012#0	Stop bit and other data
	0038#6, #7	Number specified for the input/output device
	0553	Baud rate
I/O =2 (channel 2)	0050#0	Stop bit and other data
	0038#4, #5	Number specified for the input/output device
	0250	Baud rate
I/O =3 (channel 3) * Remote buffer	0051#0	Stop bit and other data
	0038#1, #2	Number specified for the input/output device
	0251	Baud rate
	0055#2	Selection of protocol
	0055#3	Selection of RS–422 or RS–232C, and other data

	#7	#6	#5	#4	#3	#2	#1	#0
0010				PRG9				

[Data type] Bit

PRG9 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

	#7	#6	#5	#4	#3	#2	#1	#0
0015		REP						

[Data type] Bit

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.

	#7	#6	#5	#4	#3	#2	#1	#0
0019		NEOP						

[Data type] Bit

NEOP 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
0045								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.

		#7	#6	#5	#4	#3	#2	#1	#0
0389							PRG8		

[Data type] Bit

PRG8 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

Alarm and message

Number	Message	Description
079	BP/S ALARM	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	BP/S ALARM	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	BP/S ALARM	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	BP/S ALARM	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.
180	BP/S ALARM	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.

Reference item

OPERATOR’S MANUAL (For Machining Center) (B–61404E)	III.8.4	PROGRAM INPUT/OUTPUT
OPERATOR’S MANUAL (For Lathe) (B–61394E)	III.8.4	PROGRAM INPUT/OUTPUT

13.6 SIMULTANEOUS INPUT AND OUTPUT OPERATIONS (M SERIES)

General

While an automation operation is being performed, a program input from an I/O device connected to the reader/punch interface can be executed and stored in memory.
Similarly, a program stored in memory can be executed and output through the reader/punch interface at the same time.

Basic procedure for input and run simultaneous operation

(1) Search the head of a program (file) you want to run and input.
(2) Set the signal (G127#5) for DNC operation to logical 1.
(3) Set the input and run simultaneous mode select signal STRD to logical 1.
(4) Activate automatic operation.
(5) The system repeats to input and run one block of data alternately.

Basic procedure for output and run simultaneous operation

(1) Select a program you want to run and output.
(2) Set the signal (G127#5) for DNC operation to logical 1.
(3) Set the output and run simultaneous mode select signal STWD to logical 1.
(4) Activate automatic operation.
(5) The system repeats to output and run one block of data alternately.

Signal

Input and run simultaneous mode select signal STRD <G140#5>

[Classification] Input signal

[Function] When this signal becomes logical 1, the control unit:

- Selects the input and run simultaneous mode.
  To select the input and run simultaneous mode, it is necessary to select the DNC operation mode and to set this signal to logical 1.

Output and run simultaneous mode select signal STWD <G140#6>

[Classification] Input signal

[Function] When this signal becomes logical 1, the control unit:

- Selects the output and run simultaneous mode.
  To select the output and run simultaneous mode, it is necessary to select the DNC operation mode and to set this signal to logical 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G140		STWD	STRD					

Alarm and message

Number	Message	Description
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.

Note

NOTE

1 M198 (file access) cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in alarm No. 210.
2 A macro control command cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in alarm No. 123.
3 If an alarm condition occurs during the input, output and run simultaneous mode, a block being processed when the alarm condition occurs and all blocks before that are input or output.
4 In the output and run simultaneous mode, if a device used is a floppy disk drive or FA card, the file name is the execution program number.
5 When a program is being executed in the output and run simultaneous mode, if a subprogram is called, only the main program is output.

13.7 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.

Signal

External program input start signal MINP<G120#0>(M series)/ <G117#0>(T series)

[Classification] Input signal

[Function] This signal starts 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:

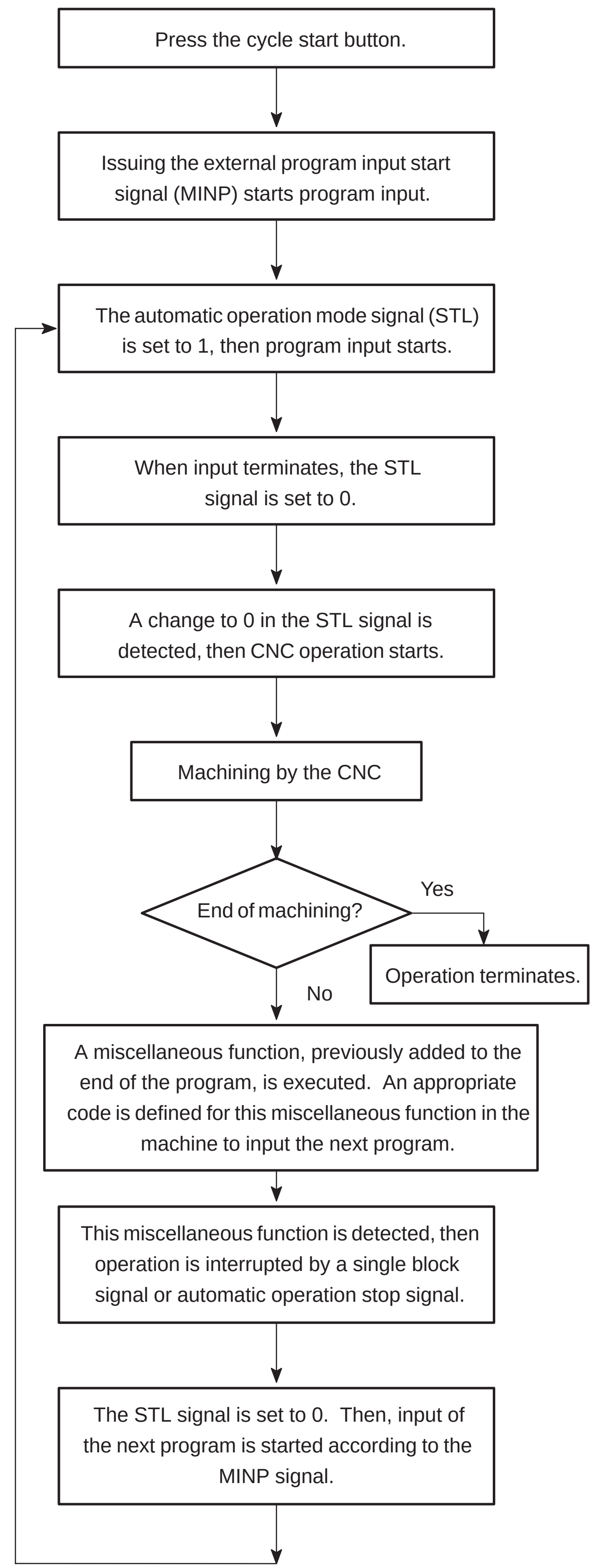
- When memory operation mode is set, but no automatic operation is being performed and program loading is not inhibited by the setting of the memory protection key, the CNC deletes all currently loaded programs, then loads a program from the external input unit into CNC memory.
- When the FANUC Handy 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 PN8), 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.
PN8	PN4	PN2	PN1	
0	0	0	0	00 (*)
0	0	0	1	01
0	0	1	0	02
0	0	1	1	03
0	1	0	0	04
0	1	0	1	05
0	1	1	0	06
0	1	1	1	07
1	0	0	0	08
1	0	0	1	09
1	0	1	0	10
1	0	1	1	11
1	1	0	0	12
1	1	0	1	13
1	1	1	0	14
1	1	1	1	15

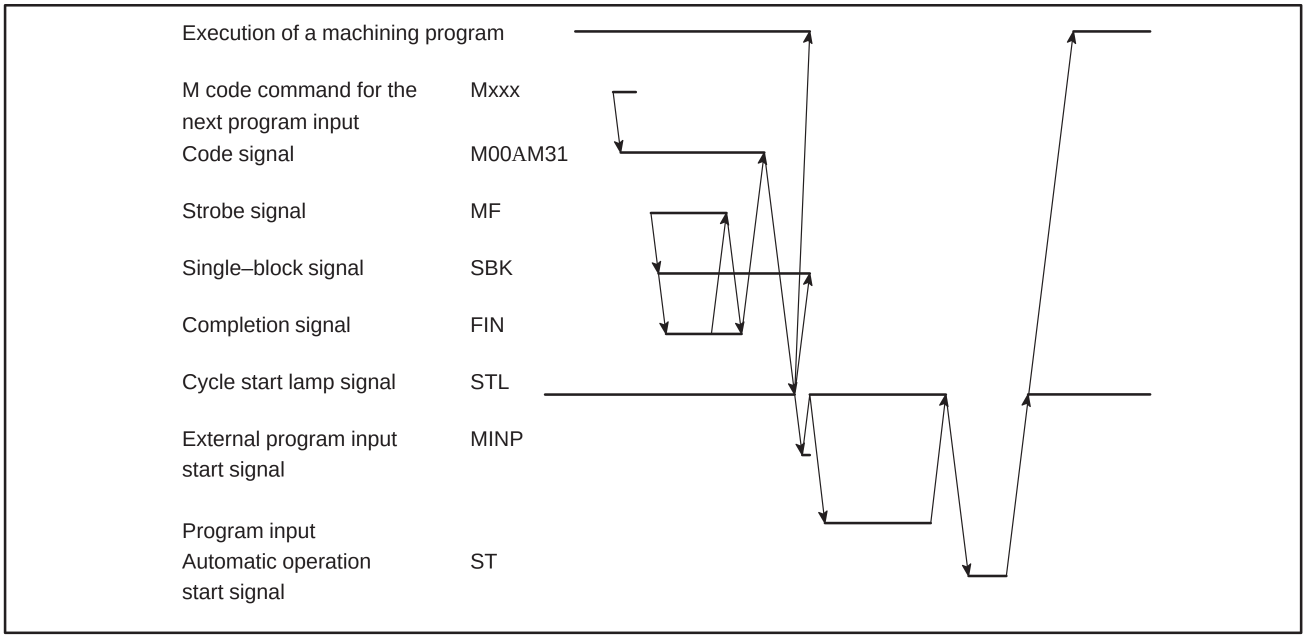
(*) File No. 00 is used for special specification; specifying file No. 00 means that no search operation is to be performed. Therefore, numbers 01 to 15 can be assigned to files.

[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	
G117								MINP	(T series)
G120								MINP	(M series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0011	MCINP							

[Data type] Bit

MCINP 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

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.

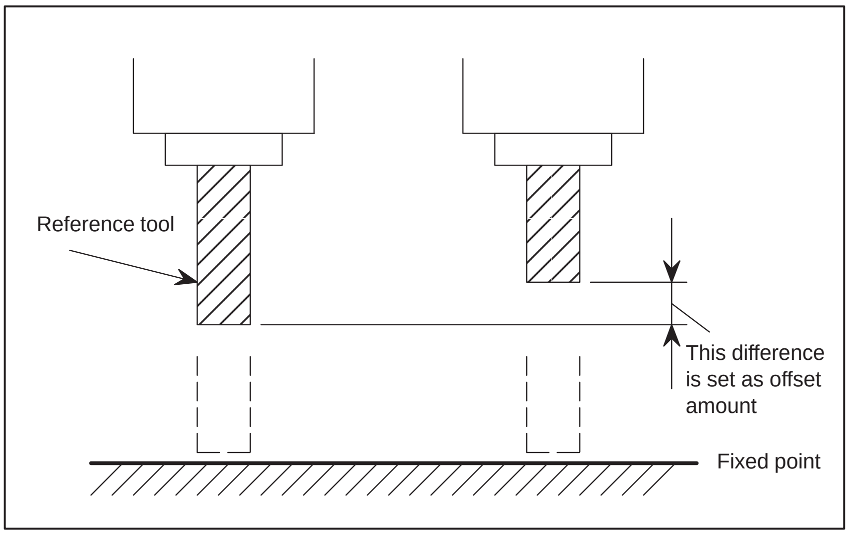
14 MEASUREMENT

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.

Call offset value display screen on the CRT. Relative positions are also displayed on this screen. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by a 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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	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.

Signal

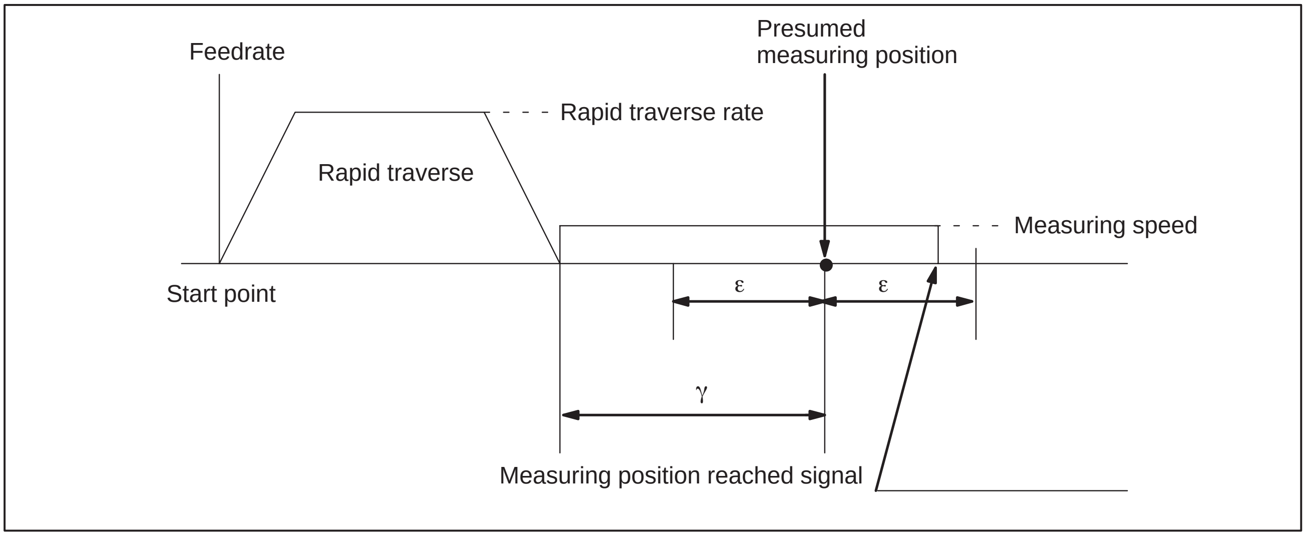
Measuring position reached signals XAE<X008#0>, YAE<X008#1>, ZAE<X008#2>(M series) XAE<X008#0>, ZAE<X008#1>(T series)

[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 O before the measuring position.
The tool then moves to the measuring position at the speed preset by a parameter no. 0558. 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 operates 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 compensation) 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 compensation nor 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	
X008							ZAE	XAE	(T series)
						ZAE	YAE	XAE	(M series)

Parameter

0558	Feedrate during measurement of automatic tool compensation	(T series)
	Feedrate during measurement of tool length automatic compensation	(M series)

[Data type] Word

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 feedrate during measurement of automatic tool compensation (T series) and tool length automatic compensation (M series).

0731	γ value on X axis during automatic tool compensation	(T series)
0732	γ value on Z axis during automatic tool compensation	(T series)
0813	γ value during tool length automatic compensation	(M series)

[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 compensation (T series) or tool length automatic compensation (M series).

> **CAUTION**
> Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

0733	ε value on X axis during automatic tool compensation	(T series)
0734	ε value on Z axis during tool automatic compensation	(T series)
0814	ε value during tool length automatic compensation	(M series)

[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 compensation (T series) or tool length automatic compensation (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 0814 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT ASSERTED (T series)	In the automatic tool compensation function (G36, G37), the measurement position reached signal (XAE or ZAE) is not turned on within an area specified in parameter 0733, and 0734 (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 compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.

Number	Message	Description
082	H–CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool compensation (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 compensation (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 compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.

Note

NOTE

1 Measurement speed, γ, and $\varepsilon$are set as parameters.ε must be positive numbers so that γ>ε.

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 turned on) – (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.
When offset memory A is used, the offset value is changed.
When offset memory B is used, the tool wear compensation value is changed.
When offset memory C is used, 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

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.14.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	II.14.5	AUTOMATIC TOOL OFFSET (G36, 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 #5064, as follows:

#5061 X axis coordinate value
#5062 Z axis coordinate value
#5063 3rd axis coordinate value
#5064 4th axis coordinate value

Signal

Skip signal SKIP<X008#7>

[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 functions as described below.

(1) 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.

(2) 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 requires at least 10 msec.
2 The CNC directly reads the skip signal SKIP<X008#7> from the machine tool; the PMC no longer requires to process the signal.
3 If the skip function G31 is not used, the PMC can use the signal terminal SKIP<X008#7> corresponding to the skip signal as a general purpose input signal.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008	SKIP							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0015					SKPF			

[Data type] Bit

SKPF 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.0015#3 SKPF=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 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:

$$ERRmax = Fm \times \frac{1}{60} \times \frac{4}{1000}$$

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:

$$Qmax = Fm \times \frac{1}{60} \times \frac{1}{1000} (20 + Tc + Ts)$$

Qmax : Overtravel amount (mm or inch)
Fm : Feedrate (mm/min or inch/min)
Tc : Cutting time constant (ms)
Ts : Servo time constant (ms) ($\frac{1}{\text{loop gain}}$)

Reference item

OPERATOR'S MANUAL (For Machining Center) (B–61404E)	II.4.8	SKIP FUNCTION(G31)
OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 instead of an ordinary skip signal (X008#7). In this case, up to eight signals can be input.

Delay and error of skip signal input is 0 - 2 msec at the NC 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.

14.3.3 Multi–step Skip (0/00–GCC)

General

In a block specifying P1 to P4 after G31, the multistage skip function stores coordinates in a custom macro variable when a skip signal (4 points) is turned on.
Also in a block specifying Q1 to Q4 after G04, the multistage skip function skips a dwell when the skip signal (4 points) 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 SKIP4 <X008#7, #2 to #4>

[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 four skip signals is applicable to blocks containing the G codes can be determined by parameter (No.0033 to 0035). 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.

(1) 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 block was supposed to be moved.

(2) 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.

(3) 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 or dwell operation is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008	SKIP			SKIP4	SKIP3	SKIP2		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0033	P2S4	P2S3	P2S2	P2S1	P1S4	P1S3	P1S2	P1S1
0034	P4S4	P4S3	P4S2	P4S1	P3S4	P3S3	P3S2	P3S1
0035					DS4	DS3	DS2	DS1

[Data type] Bit

P1S1 to P1S4, P2S1 to P2S4, P3S1 to P3S4, P4S1 to P4S4, DS1 to DS4

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	1S1	2S1	3S1	4S1	DS1
SKIP2	1S2	2S2	3S2	4S2	DS2
SKIP3	1S3	2S3	3S3	4S3	DS3
SKIP4	1S4	2S4	3S4	4S4	DS4

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 until the SKIP signal is set to “1” or the end point of the specified movement is reached, while performing linear interpolation. 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.

When no Qn command follows the G04 command, and when DS1 to DS4, bit 0 to bit 3 of parameter No.0035, are not set, dwell is not skipped.

Reference item

OPERATOR’S MANUAL (For Lathe) (B–61394E)	II.4.10	MULTI–STEP (0–GCC/00–GCC/0–GCD)

14.3.4 Torque Limit Skip (T series)

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 during the execution of G31 P99, the execution of the next block starts without executing the remaining portion of the move command.

When no torque limit is specified before executing G31 P99, the move command is executed without performing the skip operation.

For G31 P99, 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 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.

Signal

Torque limit reached signals TORQ1 to TORQ8 <F170#0 to #5>

[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 8 indicate the corresponding axis numbers.

Signal address

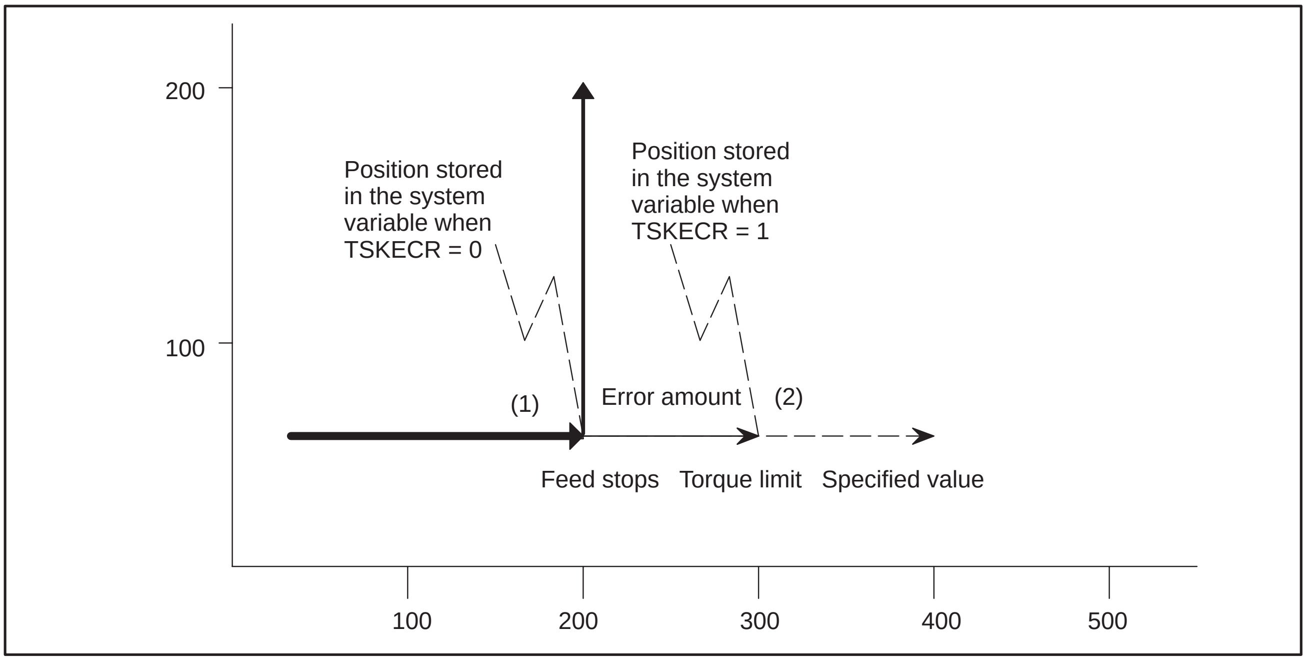
	#7	#6	#5	#4	#3	#2	#1	#0
F170			TORQ8	TORQ7	TORQ4	TOQR3	TOQR2	TORQ1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0389					TSKECR			

[Data type] Bit

TSKECR When a skip operation is performed by the G31 P99 command used to specify torque limit skip:
0 : Corrects servo errors. (1)
1 : Does not correct servo errors. (2)



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.

Warning

WARNING

Specify a torque limit before G31 P99. If G31 P99/P98 is executed with no torque limit specified, the move command is executed without a skip operation.

Caution

CAUTION

1 Before specifying G31 P99, cancel tool–tip radius compensation with G40.
2 Set the SKPF bit (bit 3 of parameter No.0015) to 0 to disable the dry run, override, and automatic acceleration/deceleration functions for the G31 skip command.

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 in the 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 standard 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<G103#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 MORB (No.0015#4) to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G103		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0015			MORB					

[Data type] Bit

MORB 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

OPERATOR'S MANUAL (For Lathe) (B–61394E)	III.11.4.2	Direct Input of Tool Offset Value

14.4.2 Input of Tool Offset Value Measured B (T series)

General

When the touch sensor is provided, the tool offset value is 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 is also automatically settable.

• Touch sensor

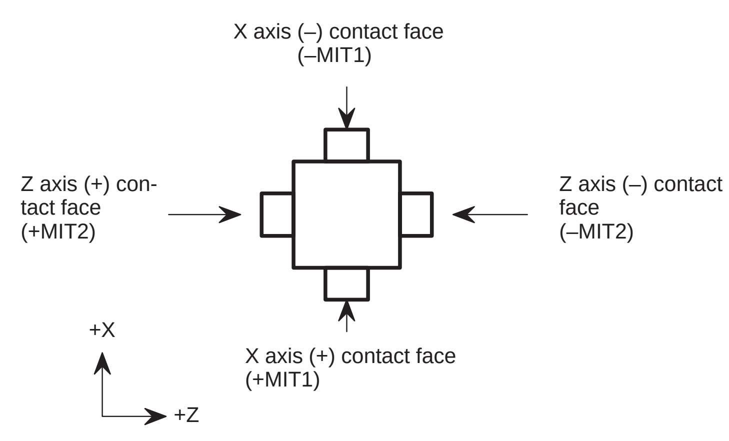
This touch sensor makes contact in two directions in each axis, and outputs four signals when it detects a tool contact. The contact faces are selected according to the tool nose figures to be measured.

a) +MIT1 (+MITX) : Contact to X-axis (+) contact face (Contact in X + direction)

b) –MIT1 (–MITX) : Contact to X-axis (–) contact face (Contact in X – direction)

c) +MIT2 (+MITZ) : Contact to Z-axis (+) contact face (Contact in Z + direction)

d) –MIT2 (–MITZ) : Contact to Z-axis (–) contact face (Contact in Z – direction)



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 nos. 0743 to 0746. 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.

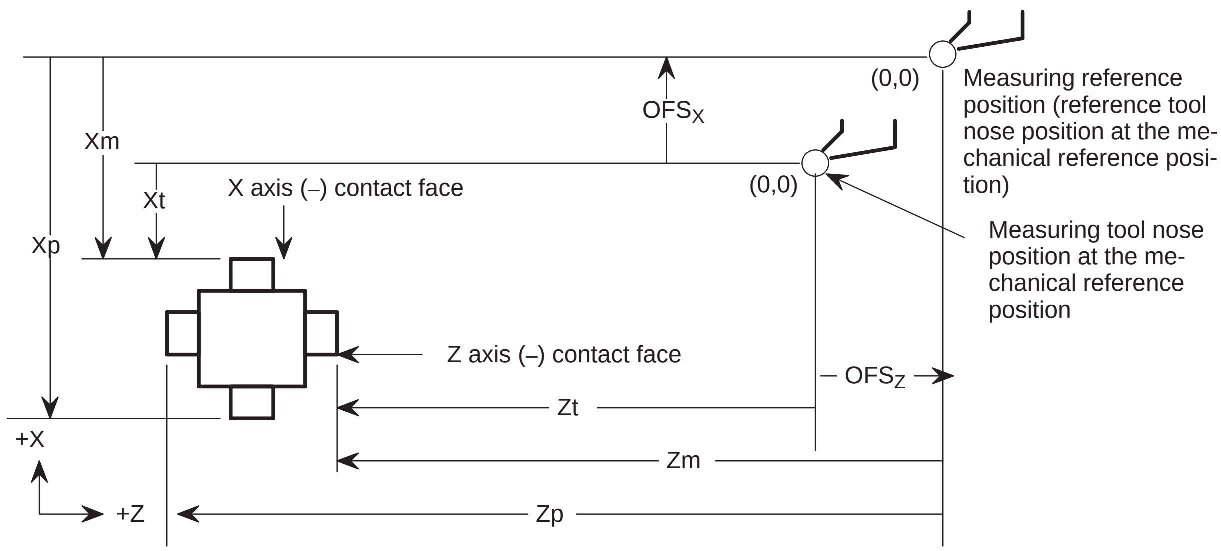
(Tool offset value to be set)

= (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.

(Example 1)

The difference between the reference tool nose tip position and the measuring tool nose tip position is settable 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 as parameters.



XP : Distance from the measuring reference position to X-axis (+) contact face (parameter no. 0743)

Xm : Distance from the measuring reference position to X-axis (–) contact face (parameter no. 0744)

Zp : Distance from the measuring reference position to Z-axis (+) contact face (parameter no. 0745)

Zm : Distance from the measuring reference position to Z-axis (–) contact face (parameter no. 0746)

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)

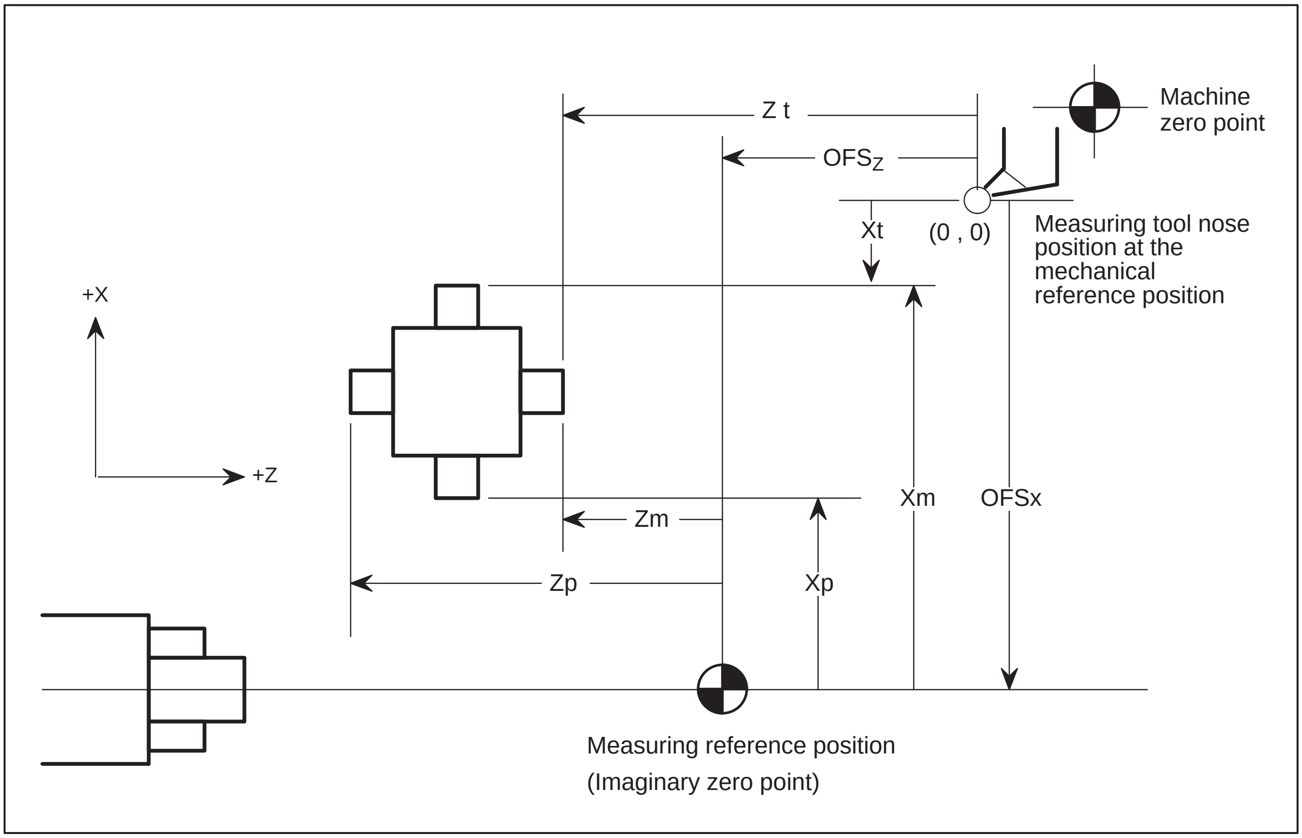
(when Xt and Zt touch the X-axis (–) contact face and Z-axis (–) contact face in the above figure)

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 point zero), 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 is settable as the tool offset value of the measuring tool, by setting the distances from the imaginary zero point to the respective contact faces as parameters.



When the imaginary zero position is set as the measuring reference position

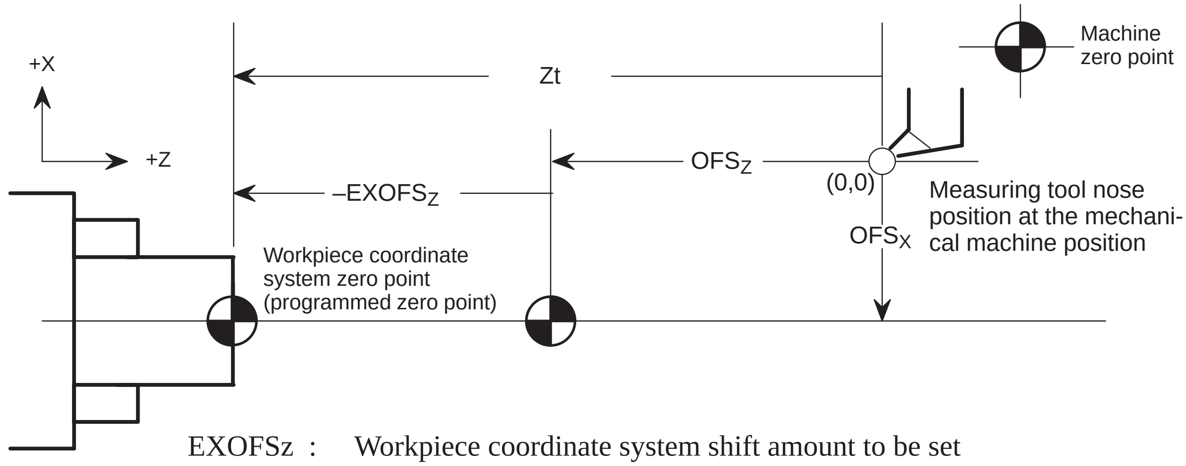
- **Setting the workpiece coordinate system shift amount**

The workpiece coordinate system shift amount for the Z-axis is settable 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))

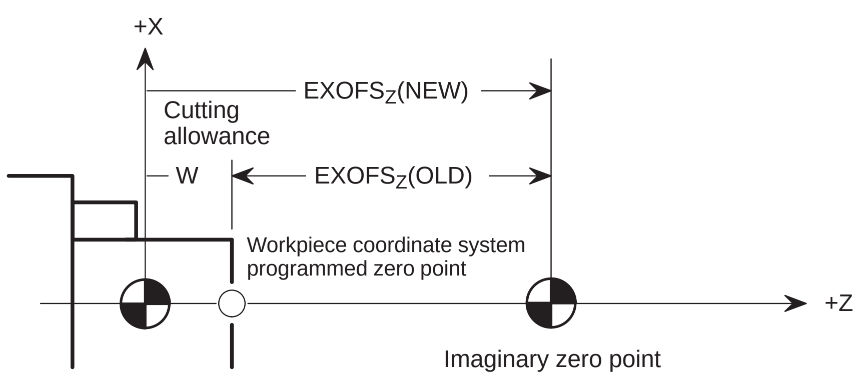
Using the above methods, the workpiece coordinate system is set with the workpiece end face (the contact point of the sensor) specified as the programmed zero point of the workpiece coordinate system of the Z-axis.



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 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 CRT display is automatically changed to the tool offset screen (geometry), and the ”OFST” indicator starts blinking in the status indication area in the bottom of the screen, which informs 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 by page key and cursor key.
Besides, the cursor can also be coincided with the tool offset number desired to set automatically by the tool offset number input signals (when parameter QNI(No.0024#6)=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 (+MIT1, –MIT1, +MIT2 or –MIT2) to input to CNC.
The tool compensation value writing signal is set to ”1”, and the :

- The axis is interlocked in this direction and its feeding is stopped.
- 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 value are set by the operations **5** and **6**.

8 Repeat operations **3** to **7** for 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 shifting amount is computed based on the machine coordinate system of the tool.

3 Set the workpiece coordinate system shifting amount writing mode select signal WOQSM to “1”.
The CRT display is automatically switches to the workpiece shifting screen, the ”WFST” indicator starts blinking at the status indicator area in the bottom of the screen, which inform that the workpiece coordinate system shifting 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.0122) in advance.
Besides the tool offset number can be set automatically by setting the tool offset number input signal (with parameter QNI(No.0024#6)=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 by manual handle feed.
When the tool edge contacts with the end face of the workpiece, input the workpiece coordinate system shift amount signal WOSET.
The workpiece coordinate system shifting 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 <G132#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 in 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 <X008#2, #4> –MIT1, –MIT2 <X008#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”, 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.

- 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 used as the manual feed interlock signal in each axis direction.

Tool offset number select signals OFN0 to OFN4 <G132#0 to #4>

[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 specififed by 7–bit binary number. Number 0 to 31 corresponds to the compensation number 1 to 32.

> **NOTE**
> This signal is available only when parameter QNI (No. 0024#6) =1.

Workpiece coordinate system shift value write mode select signal WOQSM <G132#6>

[Classification] Input signal

[Function] Selects the mode for writing the shift amount for the workpiece coordinate system.

[Operation] When this signal is turned ”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 on the CRT to the WORK SHIFT screen and blinks the OFST status display in 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 <G133#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 automatic calculation and setting of the shift amount for the workpiece coordinate system.

Tool compensation number automatic selection disable signal TNFS <G132#5>

[Classification] Input signal

[Function] Enables cursor movement during tool compensation number automatic selection.

[Operation] Even if the tool compensation number automatic selection parameter (bit 6 of parameter No. 024) is 1, setting TNFS to 1 enables cursor movement.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008			–MIT2	+MIT2	–MIT1	+MIT1		
G132	GOQSM	WOQSM	TNFS	OFN4	OFN3	OFN2	OFN1	OFN0
G133	WOSET							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0024	EDILK	QNI						

[Data type] Bit

EDILK Interlock for each axis direction
0 : Enabled
1 : Disabled

QNI In the function of input of offset value measured B
0 : Not automatically select the tool offset number
1 : Automatically selects a tool offset number

	#7	#6	#5	#4	#3	#2	#1	#0
0395							TLSCUR	

[Data type] Bit

TLSCUR 1 : When the offset screen is switched using a soft key, the previous cursor position for the tool compensation number is preserved.
0 : When the offset screen is switched using a soft key, the previous cursor position for the tool compensation number is not preserved.

0743	Distance (XP) betweeen reference position and X axis + contact surface
0744	Distance (XM) betweeen reference position and X axis – contact surface
0745	Distance (ZP) betweeen reference position and Z axis + contact surface
0746	Distance (ZM) betweeen reference position and Z axis – contact surface

[Data type] Two–word

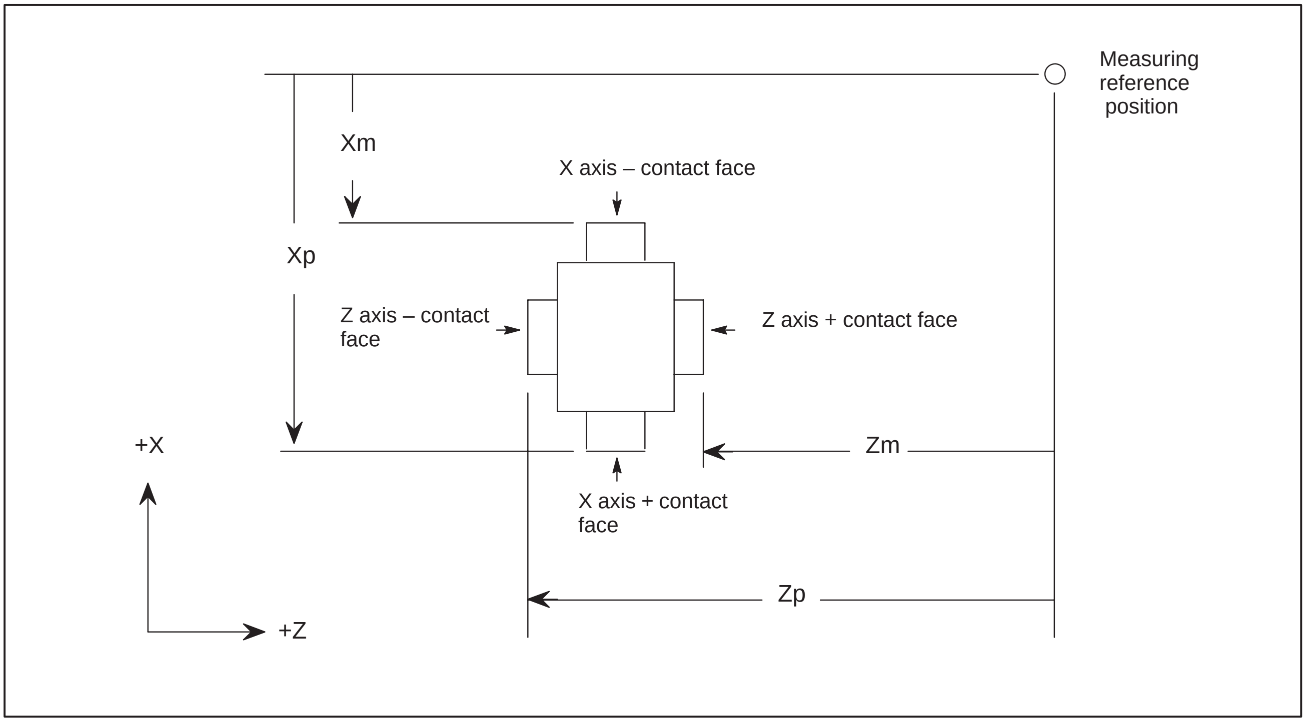
[Unit of data]

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 of input of 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 by a diameter value.



0122	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, #6 of parameter 0024, is zero).

Reference item

OPERATOR'S MANUAL (For Lathe) (B–61394E)	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 command value matches the actual measurement.

14.4.4 Grinding Wheel Diameter Automatic Compensation

Signal

Grinding wheel diameter automatic compensation signal (0–GSC) GWLF <F165#3>

[Classification] Output signal

[Function] Indicates when a specified compensation amount becomes less than the minimum allowable grinding wheel diameter (specified in a parameter).

[Operation] The signal becomes 1 when:

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F165					GWLF			

Parameter

0838	Minimum grinding wheel diameter used in a grinding wheel diameter check

[Data type] Two–word

[Unit of data] 0.001 (mm input), 0.0001 (inch input)

[Valid data range] 0 to 999999

[Description] The parameter specifies the minimum grinding wheel diameter to be used in a grinding wheel diameter check.

> **NOTE**
> When a specified compensation amount becomes less than the minimum allowable grinding wheel diameter, GWLF (bit 3 of F165) becomes 1.

15 PMC CONTROL FUNCTION

15.1 PMC AXIS CONTROL/PMC AXIS SPEED CONTROL FUNCTION

General

The PMC can directly control any given axis, independently 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 moving 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 moving distance specified

(2) Cutting feed – feed per minute, with moving distance specified

(3) Cutting feed – feed per revolution, with moving distance specified (*)

(4) Skip – feed per minute, with moving distance specified (*)

(5) Dwell

(6) Jog 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

(*)The operation cannot be used for the 5th and 6th axes.

Two specification types are applicable to PMC axis control: specification A and specification B.

When specification A is used with the M series, the X axis, Y axis, Z axis, or fourth axis is controlled using the DI/DO signals of a PMC command group.

With the T series, the third axis is controlled using the DI/DO signals of two PMC command groups.

When specification B is used with the M series or T series, any one axis selected from all axes including the 7th and 8th axes can be controlled, using the DI/DO signals of two PMC command groups. In other words, an axis can be selected from the available axes, then controlled using commands issued through two paths. From the two PMC command paths, the path to be used is selected for each axis by parameter setting. When multiple axes are simultaneously selected for one command path, the tool can be moved along these axes synchronously.

By issuing a command similar to the PMC command, an order–made macro can perform axis control.

When the increment system is set to 1/10, only 6.5 m/min can be specified as the maximum cutting speed for the PMC axis.

If this restriction proves inconvenient, set parameter No. 052 so that the least input increment for the PMC axis equals the standard specification (0.001 mm, 0.0001 inch, 0.001 deg).

Specification

Motion along a PMC controlled axis is independent of the CNC controlled axes (basic and additional axes) or other PMC controlled axes. Specification A or B is selected by setting bit 6 (EACSB) of parameter No. 0032.

Controlled axis <Specification A>

M series: One of the X axis, Y axis, Z axis, and fourth axis can be selected as a controlled axis by setting bits 0 and 1 (EAC0 and EAC1) of parameter No. 0030. The corresponding signal group is group A.

T series: The third axis and fourth axis can be controlled. The corresponding signals are of groups A and B respectively.

> **NOTE**
> If the 3rd axis is used as the Cf axis, the 3rd axis cannot be controlled.

PMC axis configuration						
	Master PC board				Multi axes	
	1st axis	2nd axis	3rd axis	4th axis	5th axis	6th axis
T series			○	○	○	○
M series				○	○	○
	○				○	○
		○			○	○
			○		○	○

EAC1	EAC0
0	0
0	1
1	0
1	1

<Specification B>

For both the M series and T series, any axis can be selected from the X–axis to 4th axis and the 7th and 8th axes, to perform simultaneous control.

> **NOTE**
> For controlling the fourth axis, expansion I/O C7 is required.

Controlled axis

(a) The controlled axis is not affected by the mode select signal.

(b) When the fourth axis is to be controlled, the fourth–axis interface option is required.

(c) By setting bit 5 (EAXOV) of parameter No. 0063, the same signals as those used in the CNC can be used to control rapid traverse override, cutting speed override, and dry run operations, in the same way as for the CNC.
By setting bit 7 (EAXOVE) of parameter No. 0078, the signals dedicated to PMC axis control can be used to perform override and dry run operations, independently of the CNC.

(d) The CNC–controlled feed hold, single block, reset, and interlock functions are not effective for the PMC controlled axis. However, control similar to such CNC control is enabled by manipulating the PMC signals.

(e) The CNC–controlled machine lock and emergency stop functions are enabled.
Machine lock, however, can be disabled by setting bit 4 (PNGMLK) of parameter No.0032.

(f) Where control of the same axis is switched between the CNC and PMC, when the axis is controlled by the PMC, the actual speed can be indicated without including motion along the PMC controlled axis by setting bit 6 (AXPCF) of parameter No.0062.

(g) In both the M series and T series, if the multi–axis printed circuit board and PMC–M are provided, the PMC can control the 5th and 6th axes.

Controlled axis in M series

(a) When the controlled axis is the fourth axis, the fourth axis ignore signal (X004.7 4NG) is enabled.

(b) An axial movement by the PMC during or after search for restarting a program has the same effect as the manual intervention for the axis. This operation, however, is invalid if the 4th axis is controlled and if no additional axis is provided. (The 4th–axis coordinate of the restart position is always displayed as 0.)

(c) When the controlled axis is the fourth axis, manual feed control cannot be performed by PMC axis control alone if the additional axis option for the fourth axis is not provided.

(d) In the same way as for machine lock, the Z–axis command cancel signal (G103.6 ZNG) is enabled.

(e) The setting parameter mirror image function (mirror X, mirror Y, and mirror 4) is enabled.

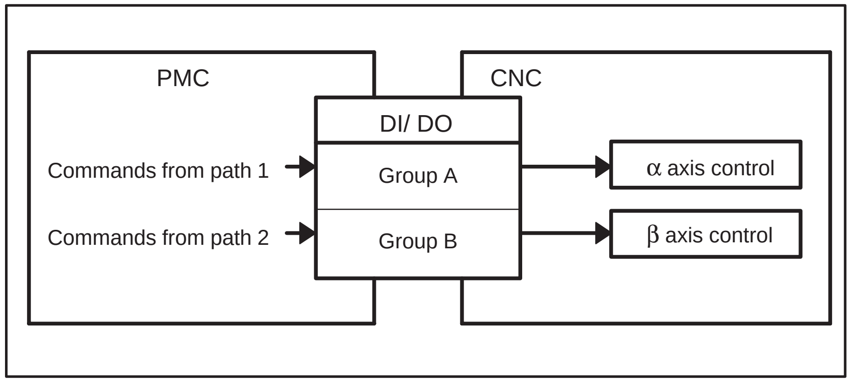
(f) If the setting of a workpiece coordinate system is specified by the CNC (G54 to G59) during tool movement along an axis under PMC control, the coordinate system cannot be set correctly.

Controlled axis in T series

(a) Radius specification is always used regardless of the settings of bit 2 (XRC) of parameter No. 0019.

(b) If the 3rd axis is used as the Cs axis, the 3rd axis cannot be controlled.

(c) When the controlled axis is the 3rd axis, manual feed control cannot be performed by PMC axis control alone. When the 3rd axis is used as the Cf axis, however, manual feed control can be performed.



In the following description, input/output signals from the four paths are called group A (path 1) and group B (path 2) 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 and EBUFB, which respectively correspond to signals of group A (path 1) and group B (path 2).

Axis selection at specification B

(1) In bits 0 to 5 of parameter No.0061, specify which DI/DO signal group (A or B) 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 EAX8), that corresponds to an axis to be controlled, to 1.

Basic procedure

(1) 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.

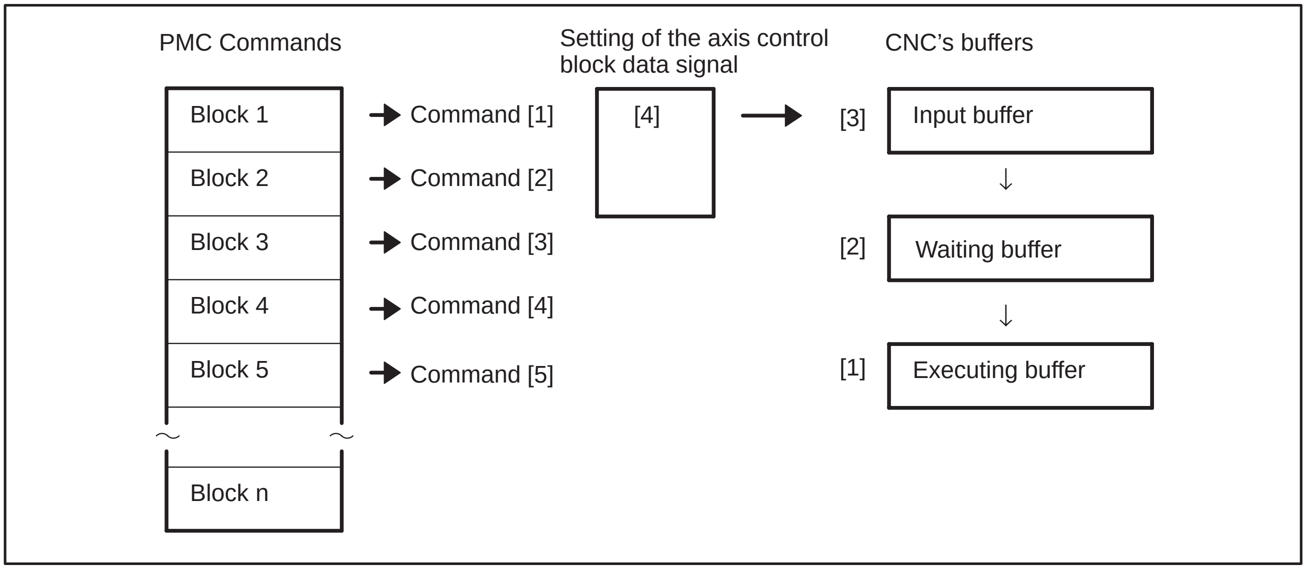
⊙ List of Signals Determining Data, Tantamount to One Block for PMC Axis Control

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 words

(2) When the data governing a complete operation (one block) is determined, reverse the logical state of 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:

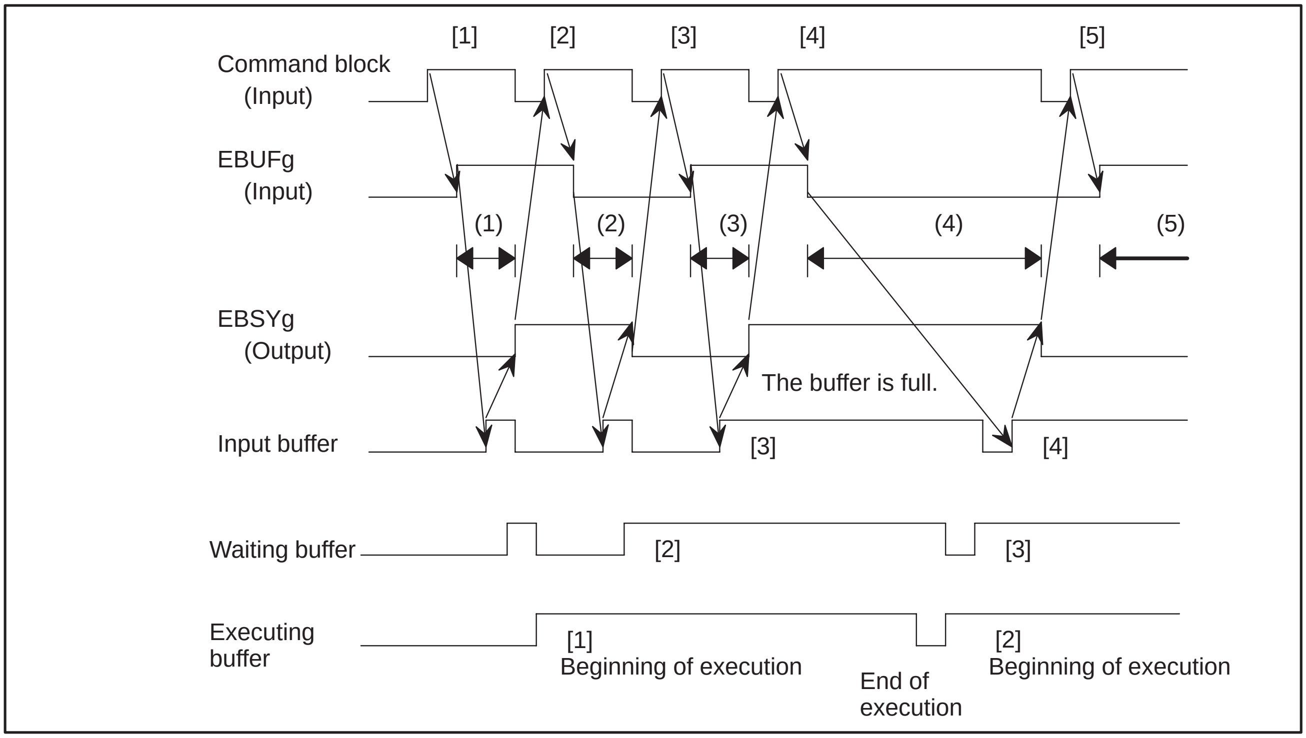
· command [2] is transferred from the waiting buffer to the executing buffer;

· 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 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.

(3) Repeat steps (1) and (2) until all the blocks have been issued.

When the final block has been issued, set control axis selection signals EAX1 to EAX8 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 EAX8 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.

(4) When control axis selection signals EAX1 to EAX8 are set to “0”, control is returned to the CNC.

Signal

Signal list

No.	Symbol	Signal name
1	EAX1 to EAX8	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 EM28g	Auxiliary function code signals
12	EMFg	Auxiliary function strobe signal
13	EFINg	Auxiliary function completion signal
14	ESOFg	Servo–off signal
15	*EAXSL	Control axis selection status signal
16	EINPg	In–position signal
17	ECKZg	Following zero checking signal
18	EIALg	Alarm signal
19	EGENg	Axis moving signal
20	EDENg	Auxiliary function executing signal
21	EOTNg	Negative–direction overtravel signal
22	EOTPg	Positive–direction overtravel signal
23	*OV1E to *OV8E	Feedrate override signals
24	OVCE	Override cancellation signal
25	ROV1E, ROV2E	Rapid traverse override signals
26	DRNE	Dry run signal
27	RTE	Manual rapid traverse selection signal
28	EOV0	Override 0% signal
29	ESKIP	Skip signal

Signal Detail

1 Control axis selection signals EAX1 to EAX8 (Specification B)

[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".

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 EAX8 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.
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.

Axis control command (hexadecimal code)	Operation
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 ZM, bits 0 to 3 of parameter No.0003, in rapid traverse mode, then performs the same operation as manual reference position return, done by the CNC.
06h	Jog 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.
0Eh	External pulse synchronization – 2nd manual handle
	Synchronizes with the second manual handle.

Axis control command (hexadecimal code)	Operation
0Fh	External pulse synchronization – 3rd manual handle
	Synchronizes with the 3rd manual handle.
10h	Speed command (linear acceleration/deceleration)
	Performs jog feed at the specified speed.
10h	Speed command (linear acceleration/deceleration)
	Performs jog feed at the specified speed.
12h	Auxiliary function
	Performs the same function as the miscellaneous function (M function), 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 or the PMC's axis interface feedrate signals EIF0g to EIF15g. This can be set with EFERPD, bit 7 of parameter No.0387.

Feed per minute

When the cutting feed – feed per minute command (EC0g to EC6g: 01h) is specified, the cutting feed time constant parameter (parameter Nos. 0651 to 0654; PEFDT to PEFDT4) and the lower cutting feedrate limit (FL) parameter (parameter Nos.0657 to 0660; PEAFLX to PEAFL4) can be set for each PMC axis, independently of the CNC.

Cutting feed – feed per revolution

When using the cutting feed – feed per revolution command (EC0g to EC6g: 02h).

M series : The optional function for threading in synchronous feed mode is necessary.

T series : The operation depends on spindle selection signal PC2SLCT <G146.7>.

The operation cannot be performed when ITPCNT, bits 4 to 6 of parameter No. 0055, specifies high–speed cycle machining.

Reference position return

When the function for returning to the reference position without dogs is enabled, and no reference position return has been performed since power–on operation, specifying a reference position return (EC0g to EC6g: 05h) command returns the tool to the reference position (positioning to the grid point nearest the current point) without having to issue a deceleration signal for reference position return. To do this, specify a jog feed (EC0g to EC6g: 06h) command to move the tool along each axis in the predetermined direction (positioning to a point near the reference position). Then, specify a reference position return (EC0g to EC6g: 05h).

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 128.

The direction of the grid relative to the proximate position depends on ZM*, bits 0 to 3 of parameter No.0003.

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 ZM*, bits 0 to 3 of parameter No.0003.

When reference position return is specified, a low feedrate (FL) can be set for the PMC axis, independently of the CNC, by using parameter No. 0672 (EZRNFL).

Reference position return without dogs

When the function for returning to the reference position without dogs is enabled, and the tool has not been returned to the reference position since the power–on operation, specifying the 1st reference position return (EC0g to EC6g: 07h) causes a P/S alarm (No.090).

1st to 4th reference position return

When using the 1st and 4th reference position return commands (EC0g to EC6g: 07h to 0Ah), the feedrate can be specified using EFERPD, bit 7 of parameter No.0387, 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 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 diameter programming is used, the amount of travel is doubled.

Display of remaining distance

When using the jog 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.

While position control is being executed for the jog 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 jog 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.

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 jog feed has been completed, always return the tool to the reference position before executing the move command.

Also, note that absolute pulse coder specification is not allowed.

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 EFEPRD, bit 7 of parameter No.0387, 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 EID0g 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 EFEPRD, bit 7 of parameter No.0387, 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	Jog feedrate EIF0g to EIF15g
Auxiliary function	12h	Auxiliary function code EID0g to EID15g

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 7 (EFERPD) of parameter 0387 is set to "1". For 1st reference position return, however, the rapid traverse rate specified with parameter 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 5 (EFML10) of parameter No.0049.

[Unit of data] When bit 5 (EFML10) of parameter No.0049 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 5 (EFML10) of parameter No.0049 is set to 1

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	10	1	mm/min
	Inch machine	0.1	0.001	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

WARNING
Cutting feedrate clamp is disabled.

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.

(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 (FRVF1) and 7 (FRVF2) of parameter No.0066, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FRVF2	FRVF1			
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 (FRVF1) and 7 (FRVF2) of parameter No. 0066, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FRVF2	FRVF1			
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. 0698 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 (FRVF1) and 7 (FRVF2) of parameter No. 0066 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 (parameter for the manual rapid traverse rate for each axis).

NOTE
When bit 2 (XRC) of parameter No.0019 is set to diameter programming, the tool moves with double pulse weight.

(14) Jog 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 jog feed.

Specify the feedrate with signals EIF0g to EIF15g, and invert the axis control command read signal EBUFg during jog 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 5 (EFML10) of parameter No.0049.

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

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.

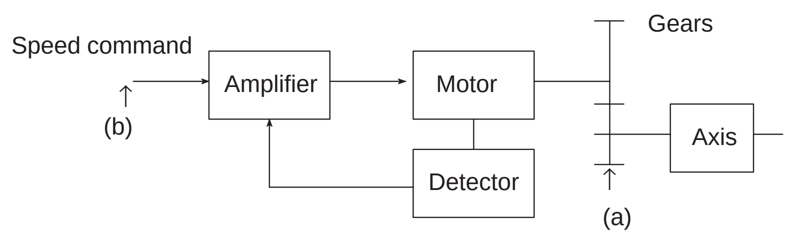
Specify a positive value for rotation in the forward direction. Specify a negative value (twos complement) for rotation in the reverse direction.

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	rpm

NOTE

1 The servo motor speed may contain a slight error, as follows:



(a) The speed command for PMC axis control requires 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:

$$\text{Fmin} = \frac{P \times 2}{15} \times \frac{1}{1000}$$

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 (rpm)

P : Number of detector pulses issued per rotation for velocity feedback

2 In speed command mode, the speed after acceleration/deceleration is specified to the servo control unit. The loop gain for position control is invalid.

4 Axis control data signals EID0g to EID31g

[Classification] Input signal

[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 used, the specified data is doubled for axial movement.

(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 diamete programming is used by bit 2 (XRC) of parameter No. 0019, dwell is executed double the specified time.

(6) Jog feed (EC0g to EC6g: 06h)

For this command, signal EID31g is used to specify the direction of jog feed, as follows:

0: Positive direction
1: Negative direction

Signals EID0g to EID30g are undefined.

(7) Auxiliary functions (EC0g to EC6g: 12h)

For this command, the signals are used to specify, in binary format, an auxiliary function code to be sent to the PMC.

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 jog 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 dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation when auxiliary function completion signal EFINg is input.

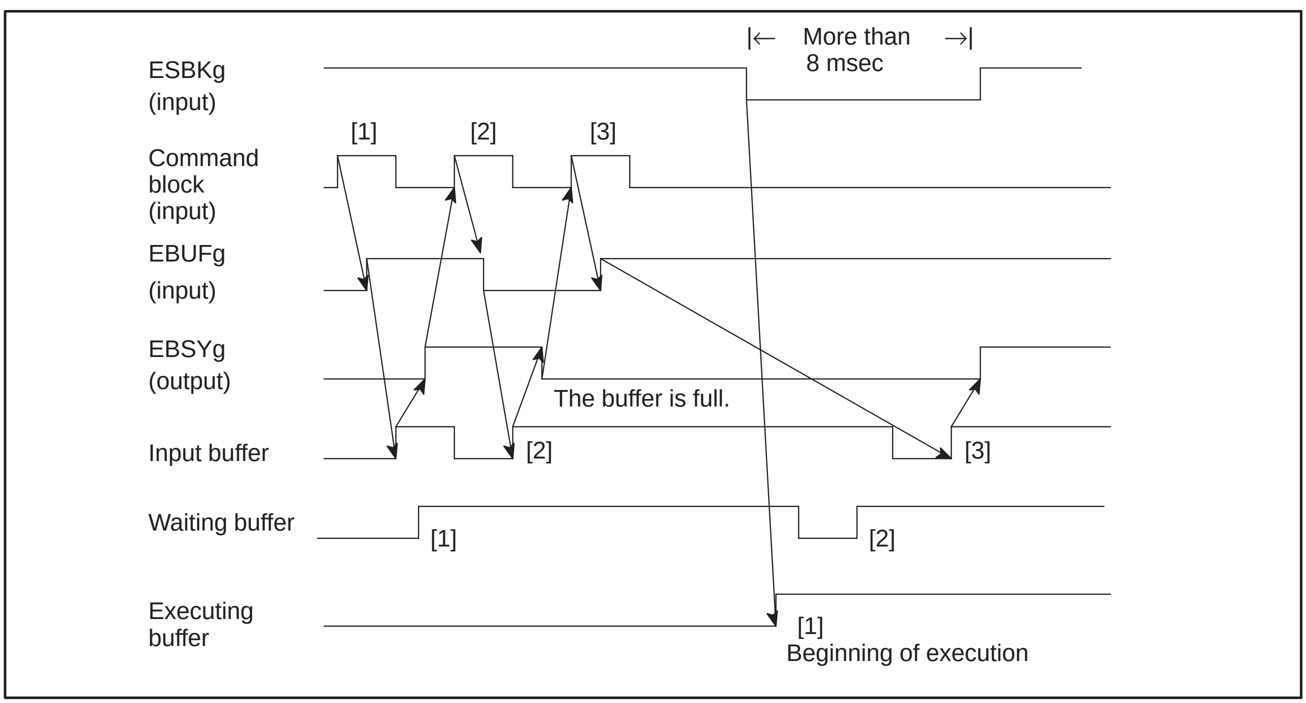
The stopped operation can be restarted by setting this signal to “0”.

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 EM28g

[Classification] Output signal

12 Auxiliary function strobe signal EMFg

[Classification] Output signal

13 Auxiliary function completion signal EFINg

[Classification] Input signal

[Function] When an auxiliary function command (EC0g to EC6g: 12h) is issued by the PMC, the CNC sends the auxiliary function code specified in signals EID0g to EID7g to auxiliary function code signals EM11g to EM28g and awaits 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.

14 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.

15 Control axis selection status signal *EAXSL (Specification B)

[Classification] Output signal

[Function] When this signal is set to "0", control axis selection signals EAX1 to EAX8 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 EAX8 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 two 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 the parameter EADSL (No.0389#4 (M)/No.0398#0 (T)) is set accordingly.

16 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.

17 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.

18 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 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) Jog 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.

19 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).

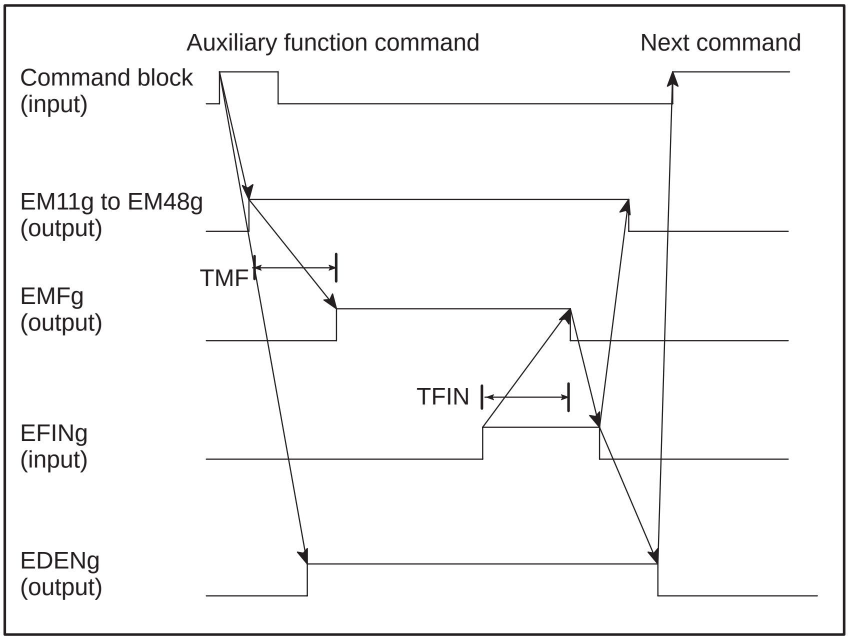
NOTE
This signal is set to "0" when distribution for the axis is completed (the signal is set to "0" during deceleration).

20 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 EID7g are sent to auxiliary function code signals EM11g to EM48g until auxiliary function completion signal EFINg is returned.

The timing chart for the command operation is shown below.



TMF and TFIN are set with parameter 0009.

21 Negative–direction overtravel signal EOTNg
22 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.

23 Feedrate override signals *OV1E to *OV8E

[Classification] Input signal

[Function] Like the CNC's feedrate override signals *OV1 to *OV8, these signals can be used to select the override for the cutting feedrate, in steps of 10% from 0 to 150%, independently of the CNC using bit 7 (EAXOVE) of parameter No.0078.

NOTE
OVRIE is set with bit 4 of parameter No. 0078.

Contact signal status on machine side				Override value	
*OV1E	*OV2E	*OV4E	*OV8E	OVRIE=1	OVRIE=0
0	0	0	0	0%	150%
1	0	0	0	10%	140%
0	1	0	0	20%	130%
1	1	0	0	30%	120%
0	0	1	0	40%	110%
1	0	1	0	50%	100%
0	1	1	0	60%	90%
1	1	1	0	70%	80%
0	0	0	1	80%	70%
1	0	0	1	90%	60%
0	1	0	1	100%	50%
1	1	0	1	110%	40%
0	0	1	1	120%	30%
1	0	1	1	130%	20%
0	1	1	1	140%	10%
1	1	1	1	150%	0%

24 Override cancellation signal OVCE

[Classification] Input signal

[Function] When override is enabled, independently of the CNC, by setting bit 7 (EAXOVC) of parameter No.0078, setting this signal to “1” fixes the cutting feed override to 100%. This signal does not affect the rapid traverse override.

25 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 7 (EAXOVC) of parameter No.0078.

Contact signal status on machine side		Override value	
ROV1E	ROV2E	OVRIE=1	OVRIE=0
1	1	100%	F0
0	1	50%	25%
1	0	25%	50%
0	0	F0	100%

F0 is the minimum feedrate specified with parameter No.0685.

26 Dry run signal DRNE
27 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 7 (EAXOVE) of parameter No.0078. When 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 multiplied by the specified override. Bit 6 (RDRNE) of parameter No.0078 can be used to specify whether to enable or disable dry run for rapid traverse.

When 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 dry run signal DRNE is set to “0”, the specified rapid traverse rate or cutting feedrate is restored.

Manual rapid traverse select signal	Command from PMC	
	Rapid traverse	Feed
1	Rapid traverse rate	Maximum jog feedrate
0	Jog feedrate(*)	Jog feedrate

* Can also be set to the rapid traverse rate with bit 6 (RDRNE) of parameter No.0078.

Contact signal status on machine side				Jog feed			
				OVRIE=1		OVRIE=0	
*OV1E	*OV2E	*OV4E	*OV8E	Metric system [mm/min]	Inch system [inch/min]	Metric system [mm/min]	Inch system [inch/min]
0	0	0	0	0	0	1260	50
1	0	0	0	2.0	0.08	790	30
0	1	0	0	3.2	0.12	500	20
1	1	0	0	5.0	0.2	320	12
0	0	1	0	7.9	0.3	200	8.0
1	0	1	0	12.6	0.5	126	5.0
0	1	1	0	20	0.8	79	3.0
1	1	1	0	32	1.2	50	2.0
0	0	0	1	50	2.0	32	1.2
1	0	0	1	79	3.0	20	0.8
0	1	0	1	126	5.0	12.6	0.5
1	1	0	1	200	8.0	7.9	0.3
0	0	1	1	320	12	5.0	0.2
1	0	1	1	500	20	3.2	0.12
0	1	1	1	790	30	2.0	0.08
1	1	1	1	1260	50	0	0

28 Override 0% signal EOV0

[Classification] Output signal

[Function] Signal EOV0 is turned “1” at 0% override value, when override is valid independently of the CNC by setting the parameter EAXOVE (bit 7 of No.0078).

29 Skip signal ESKIP

[Classification] Input signal

[Function] When this signal is set to “1”, the block being executed is immediately stopped and the next block is executed. Bit 3 (EPMSKP) of parameter No.0066 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.

Signal address

MT → CNC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
X008	SKIP	ESKIP						

PMC → CNC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
G144			EAX8	EAX7	EAX4	EAX3	EAX2	EAX1
G146							ROV2E	ROV1E
G147	DRNE	RTE	OVCE		*OV8E	*OV4E	*OV2E	*OV1E

For group A

	#7	#6	#5	#4	#3	#2	#1	#0
G210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

For group B

	#7	#6	#5	#4	#3	#2	#1	#0
G218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

CNC → PMC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F188	*EAXSL		EOV0					

	PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
For group A	F270	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
	F271								EMFA
	F272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A

	PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
For group B	F273	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
	F274								EMFB
	F275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B

Signals for the 7th and 8th axes

Signal name	Symbol
Reference position return deceleration signal	*DEC7 *DEC8
Servo off signal	SVF7 SVF8
Reference position return completion signal	ZP7 ZP8
2nd, 3rd, or 4th reference position return completion signal	ZP27, ZP37, ZP47 ZP28, ZP38, ZP48

MT → CNC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
X016	*EAXSL					*DEC7		
X017						*DEC8		

PMC → CNC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
G105			SVF8	SVF7				

CNC → PMC

PMC ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
F166	ZP28	ZP27	ZP8	ZP7	ZP48	ZP47	ZP38	ZP37

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0030							EAC1	EAC0

[Data type] Bit

EAC1, EAC0 One of the axes from the X axis to the fourth axis is selected as the controlled axis. (For M series with specification A only)

EAC1	EAC0	PMC controlled axis
0	0	Fourth axis
0	1	X axis
1	0	Y axis
1	1	Z axis

	#7	#6	#5	#4	#3	#2	#1	#0
0032		EACSB		PNGMLK				

[Data type] Bit

PNGMLK 1 : Machine lock signal MLK is invalid for the PMC controlled axis.
0 : Machine lock signal MLK is valid for the PMC controlled axis.

EACSB 1 : Specification B is used for the PMC axis control function.
0 : Specification A is used for the PMC axis control function.

	#7	#6	#5	#4	#3	#2	#1	#0
0049			EFML10					

[Data type] Bit

EFML10 1 : The cutting feedrate command for PMC axis control is multiplied by ten.
0 : The ordinary specification is valid.

	#7	#6	#5	#4	#3	#2	#1	#0
0052			NODIC6	NODIC5	NODIC4	NODIC3	NODIC2	NODIC1

[Data type] Bit

NODICn n=1 to 6
1 : The increment system 1/10 for the n–th axis is ignored.
0 : The increment system 1/10 for the n–th axis is not ignored.

	#7	#6	#5	#4	#3	#2	#1	#0	
0061			EBC8	EBC7	EBC4	EBCZ	EBCY	EBCX	(M series)
0061			EBC8	EBC7	EBC4	EBC3	EBCZ	EBCX	(T series)

[Data type] Bit

EBCx 1 : Group B is used for the DI/DO signals for the PMC axis control function.
0 : Group A is used for the DI/DO signals for the PMC axis control function.

	#7	#6	#5	#4	#3	#2	#1	#0
0062		AXPCF						

[Data type] Bit

AXPCF 1 : Movement along the PMC controlled axis is not included in the actual speed display.
0 : Movement along the PMC controlled axis is included in the actual speed display.

> **NOTE**
> This parameter is valid when control of an axis is switched between the CNC and PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
0063			EAXOV					

[Data type] Bit

EAXOV 1 : The same override and dry run operations as those for the CNC are valid for the PMC axis. (This setting is ignored when bit 7 (EAXOVE) of parameter No. 0078 is set to 1.)
0 : The same override and dry run operations as those for the CNC are invalid for the PMC axis.

	#7	#6	#5	#4	#3	#2	#1	#0
0066	ERVF2	ERVF1			EPMSKP	ENSU		

[Data type] Bit

ERVF1, ERVF2 Sets the multiplier for the feedrate when the cutting feed – feed per rotation command (EC0g to EC6g: 02h) is specified.

ERVF2	ERVF1	Multiply
0	0	$\times 1$
1	1	
0	1	$\times 10$
1	0	$\times 100$

ENSU 1 : Acceleration/deceleration is not performed for an axis which is controlled in sync with the external pulse signal.
0 : Acceleration/deceleration is performed for an axis which is controlled in sync with the external pulse signal.

EPMSKP 1 : A skip signal dedicated to the PMC is used.
0 : The same skip signal as that for the CNC is used.

	#7	#6	#5	#4	#3	#2	#1	#0
0078	EAXOVE	RDRNE		OVRIE				

[Data type] Bit

EAXOVE 1 : Override and dry run in PMC axis control independent of the CNC are valid.
0 : Override and dry run in PMC axis control independent of the CNC are not valid.

RDRNE 1 : Dry run for the rapid traverse command in PMC axis control independent of the CNC is also valid.
0 : Dry run for the rapid traverse command in PMC axis control independent of the CNC is not valid. (This setting is allowed only when EAXOVE = 1.)

OVRIE 1 : In PMC axis control, the polarity of the override signal is set independently of the CNC so that specifying “1” increases the feedrate.
0 : n PMC axis control, the polarity of the override signal is set independently of the CNC so that specifying “0” increases the feedrate. (This setting is allowed only when EAXOVE = 1.)

0350	PVRIDX

PVRIDX Axis number of an axis for which a velocity command is issued.

Setting value	T series	M sereis
1	X axis	X axis
2	Z axis	Y axis
3	3rd axis	Z axis
4	4th axis	4th axis
5	7th axis	7th axis
6	8th axis	8th axis

If a value that falls outside the above range is set, a velocity command cannot be issued for any axis. When this parameter has been set, the power must be turned off then back on before operation is continued.

	#7	#6	#5	#4	#3	#2	#1	#0
0387	EFERPD							

[Data type] Bit

EFERPD 1 : Any rapid traverse rate can be specified in PMC axis control independently of the CNC.
0 : The rapid traverse rate in PMC axis control depends on the parameter setting made for the CNC.

0651	EFDTX	
0652	EFDTY	(M series)
0652	EFDTZ	(T series)
0653	EFDTZ	(M series)
0653	EFDT3	(T series)
0654	EFDT4	

EFDTx Set the time constant of exponential acceleration/deceleration for cutting feed along each axis when the axis is used as the PMC axis.

[Valid data range] 1 to 4000

[Unit of data] msec

When 0 is set, the FEEDT value set in parameter No. 0529 is used.

0657	EAFLX	
0658	EAFLY	(M series)
0658	EAFLZ	(T series)
0659	EAFLZ	(M series)
0659	EAFL3	(T series)
0660	EAFL4	

EAFLx Set the lower feedrate limit (FL) for exponential acceleration/deceleration for cutting feed along each axis when the axis is used as the PMC axis.
When metric output mode is used, the specifiable range is 6 to 15000 (mm/min).
When inch output mode is used, the specifiable range is 6 to 5000 (inches/min).
When 0 is set, the FEDFL value set in parameter No. 0530 is used.

0672	EZRNFL

EZRNFL Sets a low feedrate (FL) at which reference position return is performed along the PMC axis.
When metric output mode is used, the specifiable range is 6 to 15000 (mm/min).
When inch output mode is used, the specifiable range is 6 to 6000 (inches/min).
When 0 is set, the ZRNFL value set in parameter No. 0534 is used.

0679	PVRTC1	(T series)
0462	PVRTC1	(M series)

PVRTC1 Time constant for linear acceleration/deceleration for a specified feedrate. Set the time required to increase/decrease the servo motor speed by 1000 rpm.

[Valid data range] 1 to +32767

[Unit of data] msec/1000rpm

When 0 is set, acceleration/deceleration control is not performed.

0685	RPDFLE

RPDFLE Specifies the lowest feedrate (Fo) for rapid traverse overrides performed in PMC axis control independently of the CNC.
When metric output mode is used, the specifiable range is 6 to 15000 (mm/min).
When inch output mode is used, the specifiable range is 6 to 6000 (0.1inches/min).

NOTE
When the increment system is 1/10, the same unit is used.

0698	PFEDMX

PFEDMX Sets the upper speed limit on feed per rotation used for the PMC axis.
When metric output mode is used, the specifiable range is 6 to 15000 (mm/min).
When inch output mode is used, the specifiable range is 6 to 6000 (0.1inches/min).

Parameters for 7th axis and 8th axis

	#7	#6	#5	#4	#3	#2	#1	#0
0021			APC8	APC7				

APC7,8 1 : When there is absolute pulse coder detector.
0 : When there is no absolute pulse coder detector.

	#7	#6	#5	#4	#3	#2	#1	#0
0022			ABS8	ABS7				

ABS7,8 1 : Reference point position is established by absolute pulse coder.
0 : Reference point position is not established by absolute pulse coder.

		#7	#6	#5	#4	#3	#2	#1	#0
0052		NODIC8	NODIC7						

NODIC7,8 1 : Current position display remains as standard specification without following decimal point position of setting units of 1/10.
0 : Current position display follows decimal point of setting units of 1/10.

		#7	#6	#5	#4	#3	#2	#1	#0
0061			DSP78						

DSP7,8 1 : 7th axis and 8th axis are displayed absolutely and relatively at current position.
0 : 7th axis and 8th axis are not displayed absolutely and relatively at current position.

		#7	#6	#5	#4	#3	#2	#1	#0
0062		C8NG		ADLN8	ADLN7				

ADLN7,8 1 : Sets linear axis.
0 : Sets rotation axis.

C8NG 1 : When removing 8th axis.
0 : When using 8th axis.

		#7	#6	#5	#4	#3	#2	#1	#0
0066				ZM8	ZM7				

ZM7,8 1 : Sets reference point return direction and backlash initial direction when turning on power as minis.
0 : Sets reference point return direction and backlash initial direction when turning on power as plus.

		#7	#6	#5	#4	#3	#2	#1	#0
0067			DMR7			GRD7			
0068			DMR8			GRD8			

DMR7,8 Setting for detection multiply.

GRD7,8 Setting for capacity of reference counter

0275	CMR7
0276	CMR8

CMR7,8 Setting of command multiply.

0285	DPAX7
0286	DPAX8

DPAX7,8 Setting of axis name (character code).
(When value is “0”, “55” and “56” are set as defaults.)

0632	LPGM7
0633	LPGM8

LPGM7,8 Loop gain for position control.
(Enabled when No.0517=0)

0637	INP7
0638	INP8

IPN7,8 Width of in–position.

0639	SERR7
0640	SERR8

SERR7,8 Position deviation amount limit value.

0641	GRDS7
0642	GRDS8

GRDS7,8 Setting of grid shift value.

0643	RPDF7
0644	RPDF8

RPDF7,8 Rapid traverse rate.
(Because no rapid traverse rate is specified for jog feed, these parameters are used also under the 1st reference position return command.)

0645	LINT7
0646	LINT8

LINT7,8 Time constant for linear type acceleration/deceleration. (For rapid traverse)

0647	BKL7
0648	BKL8

BKL7,8 Backlash value.

0649	STPE7
0650	STPE8

STPE7,8 Position deviation amount limit value during stop.

0655	EFDT7
0656	EFDT8

EFDT7,8 Time constants of exponential acceleration/deceleration of PMC axis cutting feed for each axis.

[Valid data range] 1 to 4000

[Unit of data] msec

When 0 is set, the data for FEEDT (PRM No.529) is used.

0661	EAFL7
0662	EAFL8

EAFL7,8 Lower limit speeds (FL) at exponential acceleration/deceleration of PMC axis cutting feed for each axis.

6 to 15000 Unit : mm/min (metric output)

6 to 6000 Unit : 0.1 inch/min (inch output)

When 0 is not set, the cutting FL speed for FEDFL (parameter No.530) is used.

0821	LT171
0822	LT181
0823	LT172
0824	LT182

LT171,LT181 Stored stroke limit.

LT172,LT182

0825	PRS7
0826	PRS8

PRS7,8 Coordinate value of reference position when performing automatic coordinate system setting.

0827	ABS7P
0828	ABS8P

ABS7P,ABS8P When using absolute pulse corder, value of counter data at reference position return position. (unnecessary to set)

0866	REF27
0867	REF28

REF27,8 Distance between 1st reference position and 2nd reference position.

0868	REF37
0869	REF38

REF37,8 Distance between 1st reference position and 3rd reference position.

0870	REF47
0871	REF48

REF47,8 Distance between 1st reference position and 4th reference position.

7th axis and 8th axis

Control of 7th axis and 8th axis is enabled when the 4th axis interface and 7th /8th axis printed board are installed.
Control of 7th axis and 8th axis also becomes disabled at the same time as control of 4th axis is disabled by the 4th axis ignore signal (No.X004 4NG).
When control of 7th axis and 8th axis is enabled, control of 8th axis only can be disabled if the parameter C8NG (bit 7 of No.0062) is set to on.

- Display of current position
  When control of 7th axis and 8th axis is enabled, if the parameter DSP78 (bit 6 of No.0061) is set on, position of 7th axis and 8th axis are displayed on the current position absolute/relative coordinate screen.
- Presetting the relative coordinates
  The relative coordinate of the 7th or 8th axis can be preset as specified in ALLPRE (bit 1 of parameter No. 0064) while it is displayed by pressing numeric key 7 or 8 of the MDI panel.
- Reading position information by custom macro (A specification and B specification in common)
  Position information can be obtained by reading the values of system variables.

System variable	Position information	Reading during shift
#5027 #5028	7th axis Current position (ABSMT) 8th axis Current position (ABSMT)	Not possible
#5047 #5048	7th axis Current position (ABSOT) 8th axis Current position (ABSOT)	Not possible
#5067 #5068	7th axis Skip signal position (ABSKP) 8th axis Skip signal position (ABSKP)	Possible

Alarm list

1) Absolute pulse coder (APC) alarm

Number	Contents and remedy
370	Manual reference position return is required for the 7th–axis.
371	7th–axis APC communication error.
372	7th–axis APC overtime error.
373	7th–axis APC framing error.
374	7th–axis APC parity error.
375	7th–axis APC pulse error alarm.
376	7th–axis APC battery voltage has decreased to a low level so that the data cannot be held.
377	7th–axis axis APC battery voltage reaches a level where the battery must be renewed.
378	7th–axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF).
380	Manual reference position return is required for the 8th–axis.
381	8th–axis APC communication error.
382	8th–axis APC overtime error.
383	8th–axis APC framing error.
384	8th–axis APC parity error.
385	8th–axis APC pulse error alarm.
386	8th–axis APC battery voltage has decreased to a low level so that the data cannot be held.
387	8th–axis axis APC battery voltage reaches a level where the battery must be renewed.
388	8th–axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF).

2) Servo alarms

Number	Contents and actions
406	7–axis, 8–axis overload signal is on. Or, READY signal (DRDY) for velocity control went off.
470	The position deviation value when the 7–th axis stops is larger than the set value.
471	The position deviation value when the 7–th axis moves is larger than the set value.
473	The value of position error of the 7th axis exceeds the limit of $\pm$32767. Alternatively, the value of velocity command for the D/A converter exceeds the range of –8192 to +8191. This error usually occurs as the result of an improperly set parameters.
474	7–th axis digital servo system fault. Refer to diagnosis display No. 726 for details.

Number	Contents and actions
475	A speed higher than 511875 units/s was attempted to be set in the 7–th axis. This error occurs as the result of improperly set CMR.
476	Position detection system fault in the 7–th axis pulse coder (disconnection alarm).
477	This alarm occurs when the n–th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8720 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No. 8722 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8723 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8724 (number of position feedback pulses per motor revolution).
480	The position deviation value when the 8–th axis stops is larger than the set value.
481	The position deviation value when the 8–th axis moves is larger than the set value.
483	The value of position error of the 8th axis exceeds the limit of $\pm$32767. Alternatively, the value of velocity command for the D/A converter exceeds the range of –8192 to +8191. This error usually occurs as the result of an improperly set parameters.
484	8–th axis digital servo system fault. Refer to diagnosis display No. 727 for details.
485	A speed higher than 511875 units/s was attempted to be set in the 8–th axis. This error occurs as the result of improperly set CMR.
486	Position detection system fault in the 8–th axis pulse coder (disconnection alarm).
487	This alarm occurs when the n–th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8820 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No. 8822 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8823 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8824 (number of position feedback pulses per motor revolution).

3) Over travel alarms

Number	Contents and remedy
570	Exceeded the 7–th axis + side stored stroke limit.
571	Exceeded the 7–th axis – side stored stroke limit.
580	Exceeded the 7–th axis + side stored stroke limit.
581	Exceeded the 7–th axis – side stored stroke limit.

Diagnosis display

	#7	#6	#5	#4	#3	#2	#1	#0	
0726	OVL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL	7th axis
0727	OVL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL	8th axis

Indicate the details of servo alarm 474 for the 7th axis and of servo alarm 484 for the 8th axis respectively.

OFAL An overflow alarm is being generated.

FBAL A disconnection alarm is being generated.

DCAL A regenerative discharge circuit alarm is being generated.

HVAL An overvoltage alarm is being generated.

HCAL An abnormal current alarm is being generated.

OVC A overcurrent alarm is being generated.

LV A low voltage alarm is being generated.

OVL An overload alarm is being generated.

0806	SVERR7
0807	SVERR8

SVERR7,SVERR8 Indicate the values of position errors of the 7th axis and the 8th axis respectively.

0826	ABSMT7
0827	ABSMT8

ABSMT7,ABSMT8 Indicate the machine positions of the 7th axis and 8th axis respectively.

Order–made macro (OMM) interface

An order–made macro can also control the PMC–controlled axis, using an interface similar to the PMC interface.

Axis control command read signal EBUF is checked from both the PMC signals (G210 and G218) and order–made macro data. The axis control command of the signal or data that has changed is taken to perform axis control. Both the PMC signals and order–made macro data are valid also for reset signal ECLR, axis control temporary stop signal ESTP, servo–off signal ESOF, and block stop signal ESBK.

Auxiliary function code signals EM11 to EM28 and auxiliary function strobe signal EMF are always output to the PMC. These signals are not output to the order–made macro.

Auxiliary function completion signal EFIN is input from the PMC. The order–made macro cannot set the signal. (If the order–made macro sets the signal, the setting is ignored.)

The following show the addresses of the command data of the order-made macro corresponding to the signals from PMC.

DI/DO signal A		DI/DO signal B	
PMC address	Order-made macro address	PMC adress	Order–made macro address
G210	0DBA0 H	G218	0DBA8 H
G211	0DBA1 H	G219	0DBA9 H
G212	0DBA2 H	G220	0DBAA H
G213	0DBA3 H	G221	0DBAB H
G214	0DBA4 H	G222	0DBAC H
G215	0DBA5 H	G223	0DBAD H
G216	0DBA6 H	G224	0DBAE H
G217	0DBA7 H	G225	0DBAF H
F270	0DBB0 H	F273	0DBB3 H

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 applying the normal procedure, alarm signal EIALg being 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 the 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 (G112), a P/S alarm No. 130 occurs.

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 PMC controlled axis was again selected. 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 PMC.

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 PNGMLK bit (bit 4 of parameter No.0032) is specified accordingly. 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. In other than the above blocks, a new block starts its operation after the following zero of the servo acceleration/deceleration is confirmed.
3 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.

Note

NOTE

1 The actual speed excluding the effect of the movement along a PMC–controlled axis can be displayed if the AXPCF bit (bit 6 of parameter No.0062) is specified.
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 added, the PMC cannot control the fourth axis.

Control for 5th axis and 6th axis

General

When a multi–axis printed circuit board is provided, this function specified by the PMC enables axis control independent of the CNC axis control. The order–made macro can also perform axis control by issuing a command similar to the PMC command. The axial movement command can be specified in incremental mode only. Radius programming must always be performed.

Specification

Motion along the controlled axis is independent of the motion along the basic CNC controlled axes or other PMC controlled axes.

Controlled axis

5th axis and 6th axis.

Notes for controlled axis

a) Feed hold, single block, reset, and interlock controlled by the CNC are not effective for the PMC–controlled axes. Similar control can be performed by manipulating the PMC signals.

b) Machine lock and emergency stop controlled by the CNC are effective.

c) Axis control by the PMC is not affected by the mode selection signal.

Axis name

The axis names of the 5th and 6th axes can be selected by DSPSUB1, parameter No. 7130, and DSPSUB2, parameter No. 7131. (The default names are 53 and 54.) DSPSUB, parameter No. 0029, determines whether the axis names are displayed on the current position display screen.

Command

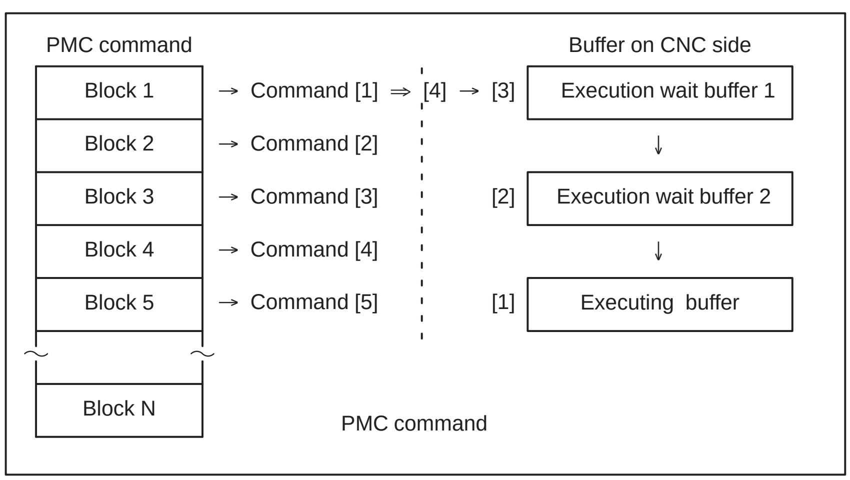
Axis control command signals EC0C to EC6C and EC0D to EC6D from the PMC specify the type of operation as indicated below:

Axis control command signal	Operation
00H	Rapid traverse (linear acceleration/deceleration) Performs the same operation as G00, used by the CNC.
01H	Cutting feed (exponential acceleration/deceleration) Performs the same operation as G98 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 ZM5 and ZM6, parameter No. 7003, by means of rapid traverse, then performs the same operation as manual reference position return, done by the CNC.
06H	Continuous feed (exponential acceleration/deceleration) Moves the tool along the controlled axis in a single direction by means of continuous feed.
12H	Auxiliary function Performs the same operation as the miscellaneous function, used by the CNC.

a) Axis control command method by PMC
When the axis control command end signal EBUF is inverted (ON → OFF/OFF → ON) after inputting axis control command signals EC0C to EC6C and EC0D to EC6D and the command data on the C side, the CNC accepts those command block and executes the command.

Movement	Axis control command signal	Command data
Rapid traverse	00H	Total move distance (EID0C to EID31C ,EID0D to EID31D)
Cutting feed	01H	Total move distance (EID0C to EID31C , EID0D to EID31D) Cutting feedrate (EID0C to EID31C , EID0D to EID31D)
Dwell	04H	Dwell time (EID0C to EID31C , EID0D to EID31D)
Reference position return	05H	
Continuous feed	06H	Feedrate (EIF0C to EIF15C , EIF0D to EIF15D) Continuous feed direction (EID31C, EID31D)
Auxiliary function	12H	Auxiliary function code (EID0C to EID7C , EID0D to EID7D)

b) Commanding two or more operations continuously
With the axis control function by PMC, the command blocks of two or more operations are buffered on the CNC side so that they are made continuously by PMC.
Namely even if one command is being executed, the CNC accepts the next command as long as there is a vacancy in the buffer on the CNC side.



When command (1) (block 1) has been executed, the following transfers are made:

Command(2) → Executing buffer

Command(3) → Executing wait buffer2

Command(4) → Executing wait buffer1

After command (2) (block 2) starts to execute, command (5) (block 5) can be sent to the CNC side.

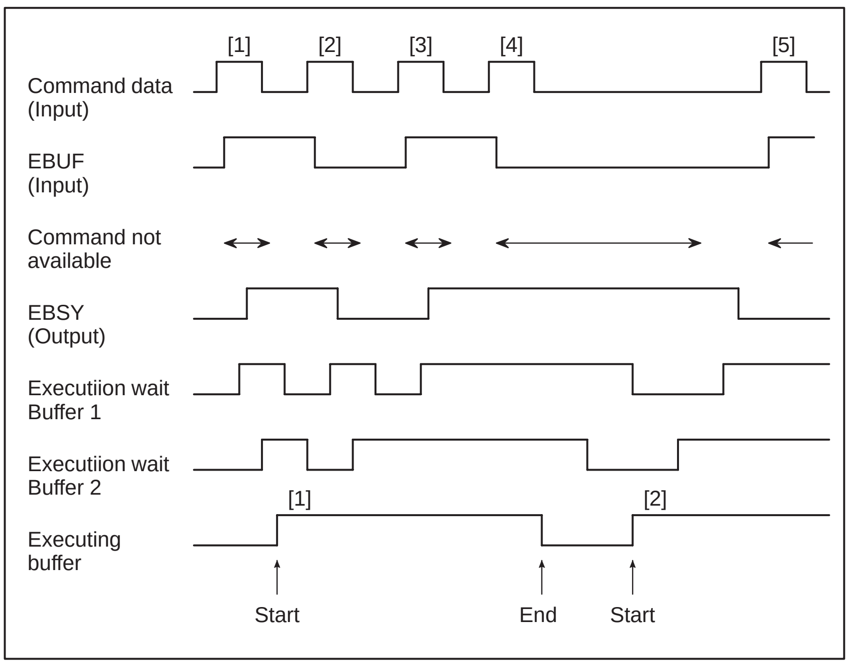
c) Buffer status on CNC side

It is possible to judge the buffer status on the CNC side by obtaining exclusive OR of the axis control command end signal EBUF on the PMC side and the axis control command read signal EBSY on the CNC side.

EBUF = EBSY	Exclusive (XOR)	Operation
0 =1 1= 1	0	It is possible to accept the command from the PMC side because the buffer is in the “vacant” status.
0 = 1 1= 0	1	It is impossible to accept the command from the PMC side because the buffer is in the “full” status. When the command currently being executed has ended and a vacancy has been created in the buffer, the command is accepted and the axis control command read signal EBSY is inverted.

d) Time chart

The command operation time chart is shown below.



Setting unit

	Metric input system	Inch input system
Least input increment	0.001 mm	0.0001 inch
Least command increment	0.001 mm	0.0001 inch

Feedrate

	Metric input system	Inch input system
Rapid traverse rate	1 to 24000 mm/min	1 to 960.00 inch/min
Cutting feedrate	1 to 15000 mm/min	1 to 600.00 inch/min

Override

EAXOVS, parameter No. 7063, enables or disables the override function.

Cutting feed override

The feedrate specified by the PMC can be overridden by the value ranging from 0% to 150% specified by the override signals (in steps of 10%).

Rapid traverse override

The rapid traverse rate can be overridden by any of the values indicated below, as specified by the rapid traverse override signals.

Fo, 25%, 50%, 100%

Fo: Certain rate specified by RPDFLS, parameter No. 7533

NOTE

If EAXOVS, parameter No. 7063, is OFF, override of 50% can be applied when the rapid traverse override signal (ROVS) is ON or override of 100% when the ROVS signal is OFF.

Dryrun

The dry run function is enabled by EAXOVS, parameter No. 7063. The feedrate specified by the PMC is ignored, and the feedrate indicated in the table below can be selected instead.

Rapid traverse button ON/OFF	Command from PMC	
	Rapid traverse	Cutting feed
ON	Rapid traverse rate	Maximum jog feedrate
OFF	Jog feedrate(*)	Jog feedrate

NOTE

* The rapid traverse rate can be selected by specifying RDRNS, parameter No. 7001, accordingly.

Time constant of exponential acceleration/deceleration of cutting feed

The time constant of cutting feed for the 5th and 6th axes only and the lower limit of cutting feedrate (FL) can be set for each axis.

Pitch error compensation

The pitch error of ball screw can be corrected in the least command increment for each axis. This function is enabled after the reference position return is performed.

Reading position information by custom macro (specifications A and B)

The position information can be obtained by reading the values of system variables.

System variable	Position information	Read during movement
#5025 #5026	5th axis Machine position (ABSMT) 6th axis Machine position (ABSMT)	Disabled
#5045 #5046	5th axis Current position (ABSOT) 6th axis Current position (ABSOT)	Disabled

PMC window function

The function specifies the data to be transferred from the PMC to the CNC and receives the data. The following control data is necessary to transfer the data.

PMC ADDRESS	DGN NO.	
G1402	3402	Data type
G1403	3403	Number of data words
G1404	3404	Axis number
G1405	3405	0

1) Data type: Specify 01 for the current position or 02 for the machine position.
2) Number of data words: Specify 02 for one axis only or 04 for all axes.
3) Axis number: To obtain the data of a single axis, specify 00 for the 5th axis or 04 for the 6th axis. To obtain the data of all axes, specify 00 so that the whole data can be obtained starting from that for the 5th axis.

After setting the data shown above, invert the transfer request bit (bit 0) of F1450 (DGN No. 3450). The data is transferred to F1452 to F1467 (DGN No. 3452 to 3467).

CAUTION
1 Always set G1405 to 0.
2 The PMC window function cannot be used in a system with the 7th and 8th axes.

Signal

Signal list

Item	Signal name	Symbol
(1)	Axis control command signal	EC0C to EC6C EC0D to EC6D
(2)	Axis control command end signal	EBUFC EBUFD
	Axis control command read signal	EBSYC EBSYD
(3)	Reset signal	ECLRC ECLRD
(4)	Axis control stop signal	ESTPC ESTPD
(5)	Servo–off signal	ESOFC ESOFD
(6)	Block stop signal	ESBKC ESBKD
	Block stop inhibit signal	EMSBKC EMSBKD
(7)	Auxiliary function BCD code signal	EM11C to EM28C EM11D to EM28D
	Auxiliary function read signal	EMFC EMFD
	Auxiliary function completion signal	EFINC EFIND
(8)	Cutting feedrate signal Continuous feedrate signal	EIF0C to EIF15C EIF0D to EIF15D
(9)	Total move distance Dwell time Auxiliary function code	EID0C to EID31C EID0D to EID31D
	Continous feed direction signal	EID31C EID31D
(10)	In-positioning signal	EINPC EINPD
(11)	Error zero checking signal	ECKZC ECKZD
(12)	Alarm signal	EIALC EIALD
(13)	Deceleration signal for reference position return	*DEC5 *DEC6
	Reference position return completion signal	ZP5 ZP6

Item	Signal name	Symbol
(14)	Axis moving signal	EGENC EGEND
(15)	Auxiliary function executing signal	EDENC EDEND
(16)	Overtravel signal in negative direction	EOTNC EOTND
	Overtravel signal in positive direction	EOTPC EOTPD
(17)	SUB ready signal	MAS
(18)	SUB servo ready signal	SAS
(19)	SUB reset signal	RSTS
(20)	SUB alarm signal	ALS
(21)	Sub–spindle enable signal	ENBS
	Sub–spindle stop signal	*SSTPS
(22)	Sub–spindle analog 12–bit command signal	RO1IS to R12IS
(23)	Sub–spindle polarity switch signal	SGNS
(24)	Override signal	*OV1S *OV2S *OV4S *OV8S
(25)	Override cancel signal	OVCS
(26)	Rapid traverse override signal	ROVS ROV1S ROV2S
(27)	Dry run signal	DRNS
	Manual rapid traverse select signal	RTS
(28)	V–READY check signal	IGNVRDY

NOTE

×××C Signals for 5th axis
×××D Signals for 6th axis

Address map

DI signal

MT→CNC

PMC ADDRESS	DGN NO.	#7	#6	#5	#4	#3	#2	#1	#0
X016	016					*DEC5			
X016	017					*DEC6			

PMC→CNC

	PMC ADDRESS	DGN NO.	#7	#6	#5	#4	#3	#2	#1	#0
5th axis	G1410	3410	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC			EFINC
	G1411	3411	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
	G1412	3412	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
	G1413	3413	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
	G1414	3414	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
	G1415	3415	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
	G1416	3416	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
	G1417	3417	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

			#7	#6	#5	#4	#3	#2	#1	#0
6th axis	G1418	3418	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD			EFIND
	G1419	3419	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
	G1420	3420	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
	G1421	3421	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
	G1422	3422	EID7D	DID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
	G1423	3423	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
	G1424	3424	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
	G1425	3425	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

PMC ADDRESS	DGN NO.	#7	#6	#5	#4	#3	#2	#1	#0
G1316	3316	ROV1S							
G1317	3317	ROV2S							
G1318	3318	DRNS							
G1320	3320	ROVS							
G1321	3321		RTS			*OV8S	*OV4S	*OV2S	*OV1S
G1323	3323	*SSTPS							IGNVRYS
G1324	3324	R08IS	R07IS	R06IS	R05IS	R04IS	R03IS	R02IS	R01IS
G1325	3325			SGNS		R12IS	R11IS	R10IS	R09IS
G1326	3326				OVCS				

DO signal

CNC→PMC

	PMC ADDRESS	DGN NO.	#7	#6	#5	#4	#3	#2	#1	#0
5th axis	G1470	3470	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
5th axis	G1471	3471								EMFC
5th axis	G1472	3472	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C

			#7	#6	#5	#4	#3	#2	#1	#0
6th axis	G1473	3473	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
6th axis	G1474	3474								EMFD
6th axis	G1475	3475	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
	F1348	3348		SAS					ZP6	ZP5
	F1349	3349	MAS			ENBS			RSTS	ALS

Signal details

NOTE
1 In the following description, the names of the signals for the 5th axis end with C (xxxC). The names of the signals for the 6th axis end with D (xxxD). (This rule does not apply to the names of the reference position return signals and section titles.)
2 The 5th–axis signal does not affect the operation of the 6th axis and vice versa.

(1) Axis control command signal <EC0C to EC6C/EC0D to EC6D>

These signals command following operations to the axis contrlled by PMC.

Axis control command (Hexadecimal code)	Operation
00H	Rapid traverse (Linear acceleration/deceleration)
01H	Cutting feed (Exponential acceleration/deceleration)
04H	Dwell
05H	Reference position returm
06H	Continuous feed (Exponential acceleration/deceleration)
12H	Auxiliary function

NOTE
The hexadecimal code bit 7 is used for another signal.

(2) Axis control command end signal (input) <EBUFC/EBUFD> Axis control command read signal (output) <EBSYC/EBSYD>

When the axis control command end signal EBUF is inverted (ON→OFF/OFF→ON), the CNC accepts those command and executes the command.

(3) Reset signal <ECLRC/ECLRD>

The signal resets the axis controlled by PMC.
When it is turned ON:

- When an axis is being moved
  Decelerates and stops the axis
- When dwell is being executed
  Stops the execution
- When an auxiliary function is being executed
  Stops the execution

And, at the same time, all the buffered commands are invalidated. Also all the commands and controls made while this signal is ON are invalidated.

(4) Axis control stop signal <ESTPC/ESTPD>

- When and axis is being moved
  Decelerates and stops the axis
- When dwell is being executed
  Stops the execution
- When and auxiliary function is being executed
  Stops the execution at the input of the auxiliary function finish signal EFIN.

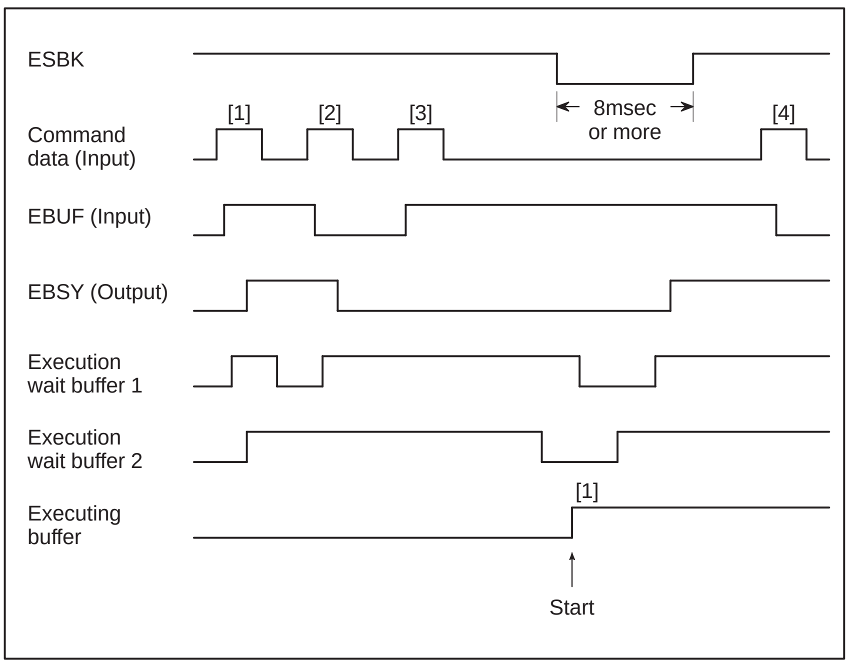
The operation of the next block is not executed, however. The stopped operation is restarted when this signal is turned OFF.

(5) Servo–off signal <ESOFC/ESOFD>

When this signal is turned ON, excitation of the axis controlled by PMC is at off and the servo–off status occurs.

(6) Block stop signal <ESBKC/ESBKD> Block stop inhibit signal <EMSBKC/EMSBKD>

When the block stop signal ESBK is turned ON while a command from PMC is being executed, the operation is being executed, the operation is stopped when the block currently being executed has ended.
The command operation time chart is shown below.



(7) Auxiliary function BCD code signal <EM11C to EM28C/EM11D to EM28D> Auxiliary function read signal <EMFC/EMFD> Auxiliary function completion signal <EFINC/EFIND>

If the command from PMC is an auxiliary function, CNC sends and auxiliary function code EID0 – EID7 to and auxiliary function BCD code EM11 – EM28 and waits for the auxiliary function finish signal EFIN. The next command block is proceeded to when the auxiliary function completion signal EFIN has been returned. The auxiliary function completion signal receiving timing are the same as with the auxiliary function of the CNC control (M function). (See item (15).)

(8) Cutting feedrate signal <EIF0C to EIF15C /EIF0D to EIF15D> Continuous feedrate signal <FIF0C to FIF15C /EIF0D to EIF15D>

a) Cutting feedrate
If the command from PMC is cutting feed, the axis feedrate shall be set in a binary code.

	Metric input	Inch input
Setting value	1 to 15000 [mm/min]	1 to 60000 [0.01 inch/min]

CAUTION
The feedrate is not clamped.

NOTE
The following parameters are used for the rapid traverse rate or FL feedrate at reference position return:
Rapid traverse rate
5th axis: Parameter No. 7518, RPDF5
6th axis: Parameter No. 7519, RPDF6
FL feedrate
5th axis: Parameter No. 7534, ZRNFLS
6th axis: Parameter No. 7534, ZRNFLS

b) Continuous feedrate
If continuous feed is specified by the PMC, the axis feedrate should be specified by a binary code. The setting is the same as the cutting feedrate for ordinary control by the PMC.

CAUTION
1 The feedrate can be changed during continuous feed. Specify a new feedrate (EIF0 to EIF15), then invert EBUF. The new continuous feedrate becomes valid. This command is not held in the buffer. EBSY need not be checked when the command is specified.
2 ECLR can cancel the continuous feed and can stop the movement.
3 The temporary stop signal (ESTP) is enabled.

NOTE
Specify these signals in the same way as for the move command for ordinary PMC control.

(9) Total move distance, dwell time, auxiliary function code <EID0C to EID31C/EID0D to EIF31D> continuous feed direction signal <EID31C/EID31D>

a) At rapid traverse or cutting feed

Set the move distance (incremental value) from the current position of the axis in a binary code.

	Metric input	Inch input
Setting value	0 to ±3FFFFFFF 0.001 mm/min	0 to ±3FFFFFFF 0.0001 inch/min

CAUTION
The value specified with the minus sign is also considered.

b) At dwell

Set the dwell time in a binary code.

Setting value	1 to 9999999 msec

c) Continuous feed direction

Set the feed direction along the axis in EID31:

0: The traveling direction is positive (+).
1: The traveling direction is negative (–).

CAUTION
Do not care about EID0 to EID30.

d) Auxiliary function

Set the auxiliary function code the send to PMC in EID0 – EID7 in a 2–digit BCD code.

(10) In–positioning signal <EINPC/EINPD>

The signal is ON while the axis controlled by PMC is in the in–position status.

(11) Error zero checking signal <ECKZC/ECKZD>

This signal is ON while the axis controlled by PMC is performing an error zero check or an in–position check.

(12) Alarm signal <EIALC/EIALD>

This signal turns ON when servo alarm or overtravel alarm is generated in the axis controlled by PMC.
Alarm is reset by the following operation and this signal is turned OFF.

- Servo alarm
  Remove the cause of the alarm and reset NC.
- Over travel alarm
  Reset NC after moving the tool within the stored stroke limit by a rapid traverse (00H) or cutting feed (01H) command (EC0 – EC6)

CAUTION
In the above "reset NC", the reset signal ECLR is not available. So the reset button or the external reset signal ERS or the emergency stop signal *ESP should be used.

(13) Deceleration signal for reference position return <*DEC5/*DEC6> Reference position return completion signal <ZP5/ZP6>

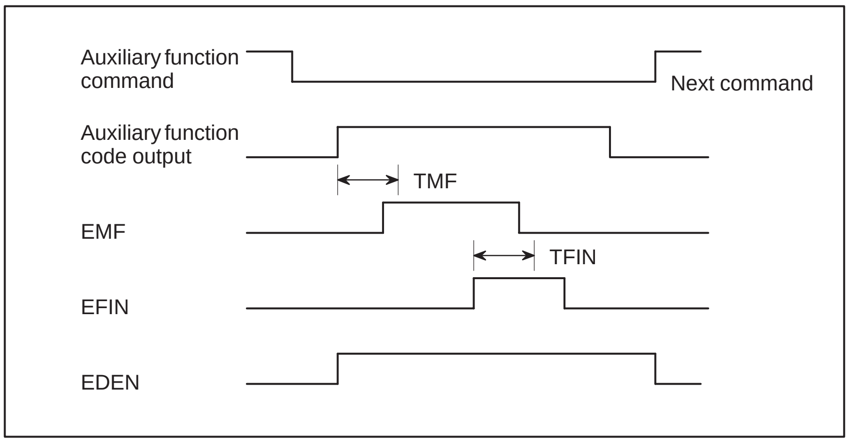
When the reference position return is specified by the PMC, the tool is moved in the direction of reference position return specified in ZM5 and ZM6, parameter No. 7003, by means of rapid traverse. Then, the operation of reference position return is performed in the same way as for the X–axis and Z–axis.

(14) Axis moving signal <EGENC/EGEND>

This signal is ON while the axis is being moved by a command (EC0 – EC6) from PMC of rapid traverse (00H), cutting feed (01H) or reference position return (05H).

(15) Auxiliary function executing signal <EDENC/EDEND>

This signal is ON while the auxiliary function finish signal EFIN has not been returned after an auxiliary function code EID0 – EID7 was sent to an auxiliary function BCD code EM11 – EM28 when the command from PMC is an auxiliary function.
The command operation time chart is shown below.



TMF and TFIN are set by a parameter

(16) Over travel signal in negative direction <EOTNC/EOTND> Overtravel signal in positive direction <EOTPC/EOTPD>

When overtravel alarm is detected, the negative direction signal EOTN is turned ON if the tool has exceeded the stroke limit on the negative side and the positive direction signal EOTP is turned ON if the tool has exceeded the stroke limit on the positive side and also the alarm signal EIAL is turned ON to inform the PMC side.
These signals are tuned OFF when the reset signal ECLR has been turned ON after resetting the overtravel alarm.

(17) SUB ready signal <MAS>

When the control unit is turned on, the ready signal (MAS) goes on.

(18) SUB servo ready signal <SAS>

When the servo system is ready for normal operation, SAS goes on.

(19) SUB reset signal <RSTS>

The reset signal (RSTS) goes on when:

1) The reset button of the control unit setting panel is pressed.
2) The external reset signal is connected.
3) The emergency stop signal is left open.

(20) SUB alarm signal <ALS>

When a servo alarm or overtravel alarm is issued, the alarm signal (EIAL) and ALS go on.

(21) Sub–spindle enable signal <ENBS> Sub–spindle stop signal <*SSTPS>

When the sub–spindle stop signal (*SSTPS) is open, the enable signal (ENBS) goes off. When the *SSTPS signal is closed, the ENBS signal goes on.

(22) Sub–spindle analog 12–bit command signal <R01IS to R12IS>

These signals control the analog voltage for the 5th and 6th axes. When all these signals are set to 0, the resultant voltage is 0 V. When all the signal are set to 1, the resultant voltage is 10 V.

(23) Sub–spindle polarity switch signal <SGNS>

The signal determines the polarity of the output voltage.

0: A positive (+) voltage is output.
1: A negative (–) voltage is output.

(24) Override signal <*OV1S, *OV2S, *OV4S, *OV8S>

The combination of ON/OFF statuses of these signals determines the value of override to be applied within the range of 0% to 150% (in steps of 10%).

Contact condition on machine side				Override value	
*OV1S	*OV2S	*OV4S	*OV8S	OVRIS=1	OVRIS=0
0	0	0	0	0%	150%
1	0	0	0	10%	140%
0	1	0	0	20%	130%
1	1	0	0	30%	120%
0	0	1	0	40%	110%
1	0	1	0	50%	100%
0	1	0	0	60%	90%
1	1	1	0	70%	80%
0	0	0	1	80%	70%
1	0	0	1	90%	60%
0	1	0	1	100%	50%
1	1	0	1	110%	40%
0	0	1	1	120%	30%
1	0	1	1	130%	20%
0	1	1	1	140%	10%
1	1	1	1	150%	0%

NOTE
OVRIS is the bit 4 of parameter No. 7003.

(25) Override cancel signal <OVCS>

When the signal is set on, the cutting feed override is set to 100%. The signal does not affect the rapid traverse override.

(26) Rapid traverse override signal <ROVS> <ROV1S, ROV2S>

The override signal is switched by setting EAXOVS, parameter No. 7063, on or off.

When EAXOVS, parameter No. 7063, is set off (0)

The override value is selected by setting the ROVS signal on or off. When the signal is on (1), the override value is 50%. When the signal is off (0), the override value is 100%.

When EAXOVS, parameter No. 7063, is set on (1)

The ON/OFF statuses of the ROV1S and ROV2S signals and of OVRIS, parameter No. 7003, determine the override value, as indicated below:

Contact condition on machine side		Override value	
ROV1S	**ROV2S**	**OVRIS=0**	**OVRIS=1**
0	0	100%	Fo
1	0	50%	25%
0	1	25%	50%
1	1	Fo	100%

NOTE
The feedrate is specified by the Fo parameter, or parameter No. 7533.

(27) Dry run signal <DRNS> Manual rapid traverse <RIS>

When the DRNS signal is set on (1), the specified rapid traverse rate and cutting feedrate are ignored. The overridden speed becomes effective.

By specifying RDRNS, parameter No. 7001, whether the dry run function is enabled for rapid traverse can be selected.

If the RTS signal is set on (1) during dry run, the tool is moved at the rapid traverse rate in rapid traverse or at the maximum jog feedrate in cutting feed. When the RTS signal is set off (0), the tool is moved at the same speed as in manual continuous feed.

When the DRNS signal is set off (0), the specified rapid traverse rate and cutting feedrate become effective again.

Contact condition on machine side				Jog feed			
				OVRIS=0		OVRIS=1	
*OV1S	*OV2S	*OV4S	*OV8S	Metric system	Inch system	Metric system	Inch system
0	0	0	0	0	0	1260	50
1	0	0	0	2.0	0.08	790	30
0	1	0	0	3.2	0.12	500	20
1	1	0	0	5.0	0.2	320	12
0	0	1	0	7.9	0.3	200	8.0
1	0	1	0	12.6	0.5	126	5.0
0	1	1	0	20	0.8	79	3.0
1	1	1	0	32	1.2	50	2.0
0	0	0	1	50	2.0	32	1.2
1	0	0	1	79	3.0	20	0.8
0	1	0	1	126	5.0	12.6	0.5
1	1	0	1	200	8.0	7.9	0.3
0	0	1	1	320	12	5.0	0.2
1	0	1	1	500	20	3.2	0.12
0	1	1	1	790	30	2.0	0.08
1	1	1	1	1260	50	0	0

(28) V–READY check signal <IGNVRDY>

While the signal is held to 1, servo alarm No. 491, V ready off, is not issued even when the 5th/6th–axis servo amplifier ready signal (DRDY) is set to 0.

Parameter

The PMC axis control function uses the following parameters:

> **NOTE**
> 1 In the description of parameters, the 5th and 6th axes correspond to the 1st and 2nd axes of the SUB side respectively.
> 2 This function is applied only to the digital servo system.
> 3 This function is enabled only with the PMC–M (or by the order–made macro).

	#7	#6	#5	#4	#3	#2	#1	#0
0029		DSPSUB						

DSPSUB 1 : The data is displayed on the sub (5th/6th–axis) current position screen. (Absolute coordinates and relative coordinates)
0 : The data is not displayed on the sub (5th/6th–axis) current position screen.

	#7	#6	#5	#4	#3	#2	#1	#0
0066								ADP56

ADP56 1 : Position of 5/6th axis is displayed in over–all position screen.
0 : Position of 5/6 axis is not displayed in over–all position screen.

> **NOTE**
> 1 This parameter is valid only when parameter DSPSUB (bit 6 of No.0029) is set to “1”.
> 2 Run hour and parts count are not displayed when setting this parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
7001		RDRNS	DECIS					SCWS

RDRNS 1 : The dry run function is enabled also for the rapid traverse command.
0 : The dry run function is disabled for the rapid traverse command.

DECIS 1 : Machine decelerates when deceleration signal is 1 in reference position return.
0 : Machine decelerates when deceleration signal is 0 in reference position return.

SCWS 1 : Minimum command increment is the inch system (Machine inch system).
0 : Minimum command increment is the metric system (Machine metric system).

	#7	#6	#5	#4	#3	#2	#1	#0
7002							PPDS	

PPDS 1 : Relative coordinate is also preset by coordinate setting.
0 : Relative coordinate is not preset by coordinate setting.

	#7	#6	#5	#4	#3	#2	#1	#0
7003				OVRIS			ZM6	ZM5

OVRIS 1 : Each override signal (*OV1 to *OV8, ROV1, ROV2) increases the speed when it is set to 1.
0 : Each override signal (*OV1 to *OV8, ROV1, ROV2) increases the speed when it is set to 0.

ZM5, ZM6 5th/6th axes reference position return direction and backlash initial direction at power ON, respectively.
1 : Minus direction
0 : Plus direction

	#7	#6	#5	#4	#3	#2	#1	#0
7004		DMR5			GRD5			
7005		DMR6			GRD6			

DMR5, DMR6 Detection multiplier for 5th and 6th axes, respectively.

GRD5, GRD6 Reference counter capacity for 5th and 6th axes, respectively.

	#7	#6	#5	#4	#3	#2	#1	#0
7009	TMF				TFIN			

TMF Time period from when the M code is sent until MF is sent (16–ms steps)

TFIN Time width in which the FIN signal is accepted (16–ms steps)

	#7	#6	#5	#4	#3	#2	#1	#0
7010	APRSS					OFFVYS		

APRSS 1 : Sets automatic coordinate system at reference position return.
0 : Does not set automatic coordinate system at reference position return.

OFFVYS 1 : Servo alarm does not occur even when VDRI is ON before outputting PRDY.
0 : Servo alarm occurs when VRDY is ON before outputting PRDY.

	#7	#6	#5	#4	#3	#2	#1	#0
7011							PML2S	PML1S

PML1S, PML2S Pitch error compensation multiplier for pitch error compensation (common to 5th axis and 6th axis)

PML1S	PML2S	Multiplier
0	0	×1
0	1	×2
1	0	×4
1	1	×8

	#7	#6	#5	#4	#3	#2	#1	#0
7021		SUB1					APC6	APC5

APC5, APC6 1 : When the absolute pulse coder detector is mounted for 5th/6th axes.
0 : When the absolute pulse coder detector is not mounted for 5th/6th axes.

SUB1 1 : PMC axis control is the 5th axis.
0 : PMC axis control is the 5th/6th axis.

	#7	#6	#5	#4	#3	#2	#1	#0
7022							ABS6	ABS5

ABS5, ABS6 1 : Reference position in absolute pulse coder is defined for 5th/6th axes.
0 : Reference position in absolute corder is not defined for 5th/6th axes.
(the signal becomes “1” automatically if the reference position return is performed by PMC axis control. Do not change setting by the time the detector is replaced.)
When installing the CNC or replacing the position detector, always set these parameter to 0, turn off and on the power then perform manual reference position return.

	#7	#6	#5	#4	#3	#2	#1	#0
7032	ROT10S			PNGMLS			ROT6	ROT5

ROT10S 1 : The unit of jog feedrate, maximum cutting feedrate, rapid traverse F0 speed, and FL feedrate at reference position return in inch output for the 5th or 6th axis is 1 degree per minute.
0 : The unit of the parameters is 0.1 degree per minute.

PNGMLS 1 : MLK signal is invalid for the 5th/6th axes.
0 : MLK signal is valid for the 5th/6th axes.

ROT5, 6 1 : The 5th/6th axes is a rotary axis.
0 : The 5th/6th axes is a linear axis.

		#7	#6	#5	#4	#3	#2	#1	#0
7035		ACMRS							

ACMRS 1 : The optional CMR function is used.
0 : The optional CMR function is not used.

> **NOTE**
> When the arbitrary command multiply function is used, either of the following settings is selected:
> 1) When the command multiplier ranges from 1/2 to 1/27
>
> $$(\text{Setting}) = \frac{1}{(\text{command multiply})} + 100$$
>
> 2) When the command multiplier ranges from 2 to 48
> (Setting) = 2 x (command multiply)
> In 2) above, set such a value that the value of command multiply becomes an integer.

		#7	#6	#5	#4	#3	#2	#1	#0
7037		PLC01S						STPT6	STPT5

PLCO1S 1 : 0.1µ pulse–coder is used.
0 : 0.1µ pulse–coder is not used.

SPTP5, 6 Type of position detector for 5th/6th axes, respectively.
1 : The separate type pulse coder is used.
0 : The separate type pulse coder is not used.

		#7	#6	#5	#4	#3	#2	#1	#0
7063				EAXOVS					

EAXOVS 1 : The dry run function is enabled. Override is applied to the axes in the same way as by the CNC.
0 : The dry run function is disabled. Override of 50% or 100% is applied only in rapid traverse.

7100	CMR5
7101	CMR6

CMR5, CMR6 Command multiplier for 5th/6th axes, respectively.

Stting code	Multiplier
1	0.5
2	1
4	2
10	5
20	10

When the arbitrary command multiply function is used (when ACMRS, parameter No. 7035, is set to 1), either of the following setting methods is selected:

(1) When the command multiplier ranges from 1/2 to 1/27

$$(\text{Setting}) = \frac{1}{(\text{command multiply})} + 100$$

(2) When the command multiplier ranges from 2 to 48
(Setting) = 2 x (command multiply)

NOTE
1 In (2) above, set such a value that the value of command multiply becomes an integer.
2 When using the arbitrary command multiply function, use the detection unit for setting backlash compensation or pitch error compensation.

7130	DSPSUB1
7131	DSPSUB2

DSPSUB1, DSPSUB2 Set the axis names of the 5th/6th axes in sequence.
The setting code shall be in accordance with the general switch code on the operator's panel Usable characters:
(X, Y, Z, U, W, A, B, C, H, 0 – 9, O, N, D, –, .)

NOTE
If '0' is set, '53' (for the fifth axis) and '54' (for the sixth axis) are assumed by default.

7500	INP5
7501	INP6

INP5, INP6 In–position width for 5th/6th axes, respectively.

[Valid data range] 0 to 32767

[Unit of data] Detection unit

7517	LPGINS

LPGINS Setting of position control loop gain (5th/6th axes in common).
(Usually set 3000)

7518	RPDF5
7519	RPDF6

RPDF5, RPDF6 Rapid traverse rate for 5th/6th axes, respectively.

[Valid data range] 30 to 24000 (Metric output)
30 to 9600 (Inch output)

[Unit of data] mm/min (Metric output)
0.1inch/min (Inch output)

> **CAUTION**
> Even when setting unit is 1/10 the unit is the same.

7522	LINT5
7523	LINT6

LINT5, LINT6 Time constant of liner acceleration and deceleration for 5th/6th axes respectively. (for rapid traverse)

[Valid data range] 8 to 4000

[Unit of data] msec

7529	FEEDTS

FEEDTS Time constant of cutting feed exponential acceleration/deceleration (5th/6th axes in common.)

[Valid data range] 0 to 4000

[Unit of data] msec
Specify 0 when the exponential acceleration/deceleration is not used.

7530	FEDFLS

FEDFLS Lower limit speed of cutting feed exponential acceleration and deceleration (FL (5th/6th axes in common).

[Valid data range] 6 to 15000 (Metric output)
6 to 6000 (Inch output)

[Unit of data] mm/min (Metric output)
0.1inch/min (Inch output)
Usually set this parameter to 0.

> **CAUTION**
> Even when setting unit is 1/10 the unit is the same.

7533	RPDFLS

RPDFLS Sets the lowest feedrate (F_0) for the rapid traverse override (Common to 5th axis and 6th axis)

[Valid data range] 6 to 15000 (Metric output)
6 to 6000 (Inch output)

[Unit of data] mm/min (Metric output)
0.1inch/min (Inch output)

> **CAUTION**
> Even when setting unit is 1/10 the unit is the same.

7534	ZRNFLS

ZRNFLS Low feedrate at reference point return (FL) (5th/6th axes in common).

[Valid data range] 6 to 15000 (Metric output)
6 to 6000 (Inch output)

[Unit of data] mm/min (Metric output)
0.1inch/min (Inch output)

> **CAUTION**
> Even when setting unit is 1/10 the unit is the same.

7535	BKL5
7536	BKL6

BKL5, BKL6 Backlash amount for 5th/6th axes, respectively.

[Valid data range] 0 to 2550 (Metric output)
0 to 2550 (Inch output)

[Unit of data] 0.001mm/min (Metric output)
0.0001inch/min (Inch output)

> **CAUTION**
> 1 Unit becomes 1/10 in increment system 1/10.
> 2 The detection unit is used when the optional CMR function is used.

7539	SPDLCS

SPDLCS Set the subspindle speed offset compensated value; the zero offset compensation value of subspindle speed command voltage.
(With constant surface speed control)

[Valid data range] 0 to ±8191

[Unit of data] VELO

7593	STPE5
7594	STPE6

STPE5, STPE6 Limit value of positional deviation for 5th and 6th axes at stop, respectively.

[Valid data range] 0 to 32767

[Unit of data] Detection unit

7700	LT151
7701	LT161
7704	LT152
7705	LT162

LT151, LT161 Stored stroke limit of 5th and 6th axes.

LT152, LT162

[Valid data range] 0 to ±99999999

[Unit of data] 0.001mm (Metric output)
0.0001inch (Inch output)

Specify a distance from the reference position. Any value beyond the limit specified in the parameter is inhibited. Usually, the maximum stroke of the machine is specified. When the limit is exceeded, an overtravel alarm is issued.
The stroke should leave leeway for the variations in the detection operation. As a guide, make an allowance of rapid traverse rate multiplied by 1/5 (mm) for a metric machine.
When the power is turned on or after emergency stop or servo alarm is released, the actual machine position is a bit deviated from the stored position. Before starting operation, carry out the reference position return. Otherwise, the operation is performed with the overtravel detection position deviated likewise.
The stroke becomes infinite when the following is set:

LT1□1 LT1□2

Represents an axis.

> **CAUTION**
> 1 The parameters cannot be used for the rotation axis.
> 2 Unit becomes 1/10 in increment system 1/10.

7708	PRS5
7709	PRS6

PRS5, PRS6 Coordinate value of 5th and 6th axes reference point. (Set in unit of input system)

[Valid data range] 0 to ±99999999

[Unit of data] 0.001mm (Metric output)
0.0001inch (Inch output)

> **CAUTION**
> Unit becomes 1/10 in increment system 1/10.

7713	PECINT5
7714	PECINT6

PECINT5, PECINT6 Compensation intervals at pitch error compensation of 5th/6th axes.

[Valid data range] 8000 to 99999999 (Metric output)
4000 to 99999999 (Inch output)

[Unit of data] 0.001mm (Metric output)
0.0001inch (Inch output)

> **CAUTION**
> 1 When 0 is set to this parameter, the compensation is not performed.
> 2 Unit becomes 1/10 in increment system 1/10.

7717	PROUND

PROUND If the 5th or 6th axis is a rotation axis, the value specified in this parameter becomes the reference value for machine coordinate rounding. When 0 is specified, the reference value of coordinate rounding is 360 degrees as usual.

[Unit of data] 0.001deg

8500	to	8565	Parameters related to degital servo for 5th axis
8600	to	8665	Parameters related to degital servo for 6th axis

5000	PECORG5
6000	PECORG6

PECORG5, PECORG6, Set the position of the zero point on the pitch error compensation table for the 5th axis and 6th axis respectively.
Set an appropriate value for the machine, ranging from 0 to 127, for each axis.

5001	to	5128	Pitch error compensation for the 5th axis
6001	to	6128	Pitch error compensation for the 6th axis

A compensation value ranging from 0 to ±7 can be specified. Any other value is ignored.

Alarm

The servo alarm or overtravel alarm issued in axis control by the PMC is detected for the 5th and 6th axes.

If an alarm is issued, the CNC sets alarm signal EIAL on to inform the PMC of the alarm, as well as performing processing required for ordinary alarms.

This signal goes off when the alarm status is released.

1) Absolute pulse coder (APC) alarm

Number	Contents and remedy
350	Manual reference position return (by PMC axis control)) is required for the 5th–axis.
351	5th–axis APC communication error.
352	5th–axis APC overtime error.
353	5th–axis APC framing error.
354	5th–axis APC parity error.
355	5th–axis APC pulse error alarm.
356	5th–axis APC battery voltage has decreased to a low level so that the data cannot be held.
357	5th–axis axis APC battery voltage reaches a level where the battery must be renewed.
358	5th–axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF).
360	Manual reference position return (by PMC axis control)) is required for the 6th–axis.
361	6th–axis APC communication error.
362	6th–axis APC overtime error.
363	6th–axis APC framing error.
364	6th–axis APC parity error.
365	6th–axis APC pulse error alarm.
366	6th–axis APC battery voltage has decreased to a low level so that the data cannot be held.
367	6th–axis axis APC battery voltage reaches a level where the battery must be renewed.
368	5th–axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF).

2) Servo alarms

Number	Contents and actions
450	The position deviation value when the 5–th axis stops is larger than the set value.
451	The position deviation value when the 5–th axis moves is larger than the set value.
452	The 5th–axis drift is too high. (500VELO is exceeded.)
453	The contents of the 5th–axis error register exceed the limit of ±32767. Otherwise, the value of the velocity command for the D/A converter is beyond the range of –8192 to +8191. This error usually occurs as the result of an improperly set parameters.
455	A speed higher than 511875 units/s was attempted to be set in the 5–th axis. This error occurs as the result of improperly set CMR.
456	Position detection system fault in the 5–th axis pulse coder (disconnection alarm).
460	The position deviation value when the 6–th axis stops is larger than the set value.
461	The position deviation value when the 6–th axis moves is larger than the set value.
462	The 6th–axis drift is too high. (500VELO is exceeded.)
463	The contents of the 6th–axis error register exceed the limit of ±32767. Otherwise, the value of the velocity command for the D/A converter is beyond the range of –8192 to +8191. This error usually occurs as the result of an improperly set parameters.
465	A speed higher than 511875 units/s was attempted to be set in the 6–th axis. This error occurs as the result of improperly set CMR.
466	Position detection system fault in the 6–th axis pulse coder (disconnection alarm).
490	The sub overload signal is high.
491	The sub velocity control ready signal (VRDY) went off.
494	The sub position control ready signal (RRDY) is off, but the velocity control ready signal (VRDY) does not go off. Alternatively, when the power is applied, the DRDY is on, but the MCON is not. Ensure that the axis card and servo amplifier are connected.
495	An error occurs in the sub position control system. It is likely that a return to the reference position failed because of an error in the NC or the servo system. Retry a return to the reference position.

3) Over travel alarms

Number	Contents and remedy
550	Exceeded the 5–th axis + side stored stroke limit.
551	Exceeded the 5–th axis – side stored stroke limit.
560	Exceeded the 6–th axis + side stored stroke limit.
561	Exceeded the 6–th axis – side stored stroke limit.

4) Digital servo alarms

Number	Contents and actions
454	5–th axis digital servo system fault. Refer to diagnosis display No. 724 for details.
457	This alarm occurs when the 5–th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8520 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No. 8522 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8523 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8524 (number of position feedback pulses per motor revolution).
464	6–th axis digital servo system fault. Refer to diagnosis display No. 725 for details.
467	This alarm occurs when the 6–th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8620 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No. 8622 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8623 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8624 (number of position feedback pulses per motor revolution).

Diagnosis display

The details of digital servo alarm 4□4 for the 5th axis and 6th axis are indicated in diagnostic data No. 724 and 725 respectively.

	#7	#6	#5	#4	#3	#2	#1	#0	
0724	OVL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL	5th axis
0725	OVL	LV	OVC	HCAL	HVAL	DCAL	FBAL	OFAL	6th axis

OFAL An overflow alarm is being generated.

FBAL A disconnection alarm is being generated.

DCAL A regenerative discharge circuit alarm is being generated.

HVAL An overvoltage alarm is being generated.

HCAL An abnormal current alarm is being generated.

OVC A overcurrent alarm is being generated.

LV A low voltage alarm is being generated.

OVL An overload alarm is being generated.

Position error display

0804	SVERR5
0805	SVERR6

SVERR5, SVERR6 Indicate the position errors of the 5th axis and 6th axis respectively.

Display of machine position viewed from the reference position

0824	ABSMT5
0825	ABSMT6

ABSMT5, ABSMT6 Indicate the machine positions on the 5th axis and 6th axis respectively.

Order–made macro (OMM) interface

An order–made macro can also control the PMC–controlled axis, using an interface similar to the PMC interface.

Axis control command read signal EBUF is checked from both the PMC signals (G1410 and G1418) and order–made macro data. The axis control command of the signal or data that has changed is taken to perform axis control. Both the PMC signals and order–made macro data are valid also for reset signal ECLR, axis control temporary stop signal ESTP, servo–off signal ESOF, and block stop signal ESBK.

Auxiliary function code signals EM11 to EM28 and auxiliary function strobe signal EMF are always output to the PMC. These signals are not output to the order–made macro.

Auxiliary function completion signal EFIN is input from the PMC. The order–made macro cannot set the signal. (If the order–made macro sets the signal, the setting is ignored.)

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 address signals EA0 to EA6 that indicate the data type and data signals ED0 to ED15.

(2) The PMC sets read signal ESTB to 1.

(3) When the ESTB signal is set to 1, the control unit reads the address.

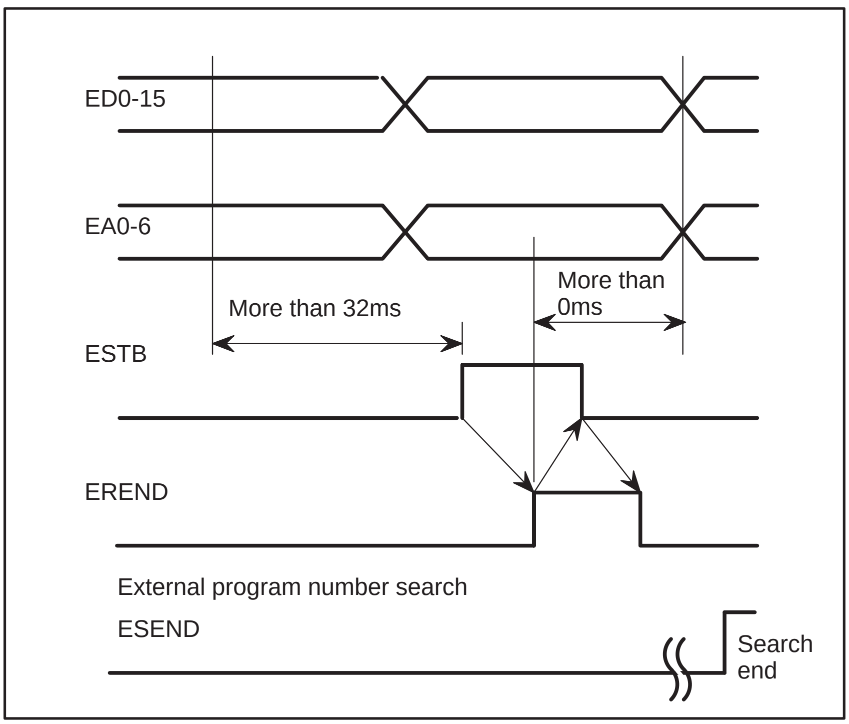
(4) After reading the address, the control unit sets read completion signal EREND to 1.

(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:



Kind of data accessed by external data input

<table>
<tr><th rowspan="2">No.</th><th rowspan="2">Item</th><th rowspan="2">E
S E E E
T A A A
B 6 5 4</th><th rowspan="2">E E E E
A A A A
3 2 1 0</th><th colspan="4">ED15 to ED0</th></tr>
<tr><th>15141312</th><th>1110 9 8</th><th>7 6 5 4</th><th>3 2 1 0</th></tr>
<tr><td rowspan="2">1</td><td rowspan="2">External program number search</td><td rowspan="2">1 0 0 0</td><td rowspan="2">× × × ×</td><td colspan="4">Program number(BCD4 digits)</td></tr>
<tr><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td></tr>
<tr><td rowspan="2">2</td><td rowspan="2">External tool compensa-tion</td><td rowspan="2">1 0 0 1</td><td rowspan="2">× × × ×</td><td colspan="4">Offset value(BCD 4 digits with sign)</td></tr>
<tr><td>±0 to 7</td><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td></tr>
<tr><td rowspan="2">3</td><td rowspan="2">External workpiece coordi-nate
system shift</td><td rowspan="2">1 0 1 0</td><td rowspan="2">axis code</td><td colspan="4">Shift value(BCD 4 digits with sign)</td></tr>
<tr><td>±0 to 7</td><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td></tr>
<tr><td>4</td><td>External machine coordi-nate system shift</td><td>1 0 1 1</td><td>axis code</td><td colspan="4">Machine coordinate system shift value(binary)
±0 to 9999</td></tr>
<tr><td rowspan="9">5</td><td rowspan="2">Alarm set</td><td rowspan="2">1 1 0 0</td><td rowspan="2">0 0 0 0</td><td colspan="4">Alarm No.(binary)</td></tr>
<tr><td></td><td colspan="3">0 to 999</td></tr>
<tr><td rowspan="2">Alarm clear</td><td rowspan="2">1 1 0 0</td><td rowspan="2">0 0 0 1</td><td colspan="4">Alarm No.(binary)</td></tr>
<tr><td></td><td colspan="3">0 to 999</td></tr>
<tr><td rowspan="2">Oper to tor message list</td><td rowspan="2">1 1 0 0</td><td rowspan="2">0 1 0 0</td><td colspan="4">Message No.(binary)</td></tr>
<tr><td></td><td colspan="3">0 to 999</td></tr>
<tr><td rowspan="2">Operator message clear</td><td rowspan="2">1 1 0 0</td><td rowspan="2">0 1 0 1</td><td colspan="4">Message No.(binary)</td></tr>
<tr><td></td><td colspan="3">0 to 999</td></tr>
<tr><td>Message</td><td>1 1 0 0</td><td>0 × 1 1</td><td colspan="4">character(Character code)</td></tr>
<tr><td rowspan="4">6</td><td rowspan="2">Substitute No. of parts required</td><td rowspan="2">1 1 1 0</td><td rowspan="2">0 0 0 0</td><td colspan="4">No. of parts required(BDC4 digits)</td></tr>
<tr><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td></tr>
<tr><td rowspan="2">Substitute No. of parts machined</td><td rowspan="2">1 1 1 0</td><td rowspan="2">0 0 0 1</td><td colspan="4">No. of parts machined(BDC4 digits)</td></tr>
<tr><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td><td>0 to 9</td></tr>
</table>

WARNING

Though bits EA4 to EA6 distinguish one set of data from another, the machine side must be interlocked in order to prevent other function data being fed during a process for which they are invalid.

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 the outside and is selected in the CNC memory.
For machines that can load several kinds of workpieces, this function can automatically select for execution the program corresponding to a specific workpiece.
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.

The ESEND signal switches from “0” to “1” at the end of the 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 make a cycle start signal after the search.

NOTE

1 The external program number search is valid when parameter EXTS (No.0028#4) is 1.
2 The reset state is when the automatic operation lamp is 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 not a program stored in memory corresponding to the set program number, alarm No.59 will be activated.
4 Program search is not made if the program number is set to “0”. When the start button is pushed, execution starts from the position indicated by the pointer, instead.

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 amount is 0 to $\pm$7.999mm or $\pm$0.7999inch at a time.

Data specification method in 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 ← Offset amount → LSB														

sign: 0 : + ; 1 : –

Offset amount: Specified by BCD 4-digit code (0000 to 7999)

EA2–EA0: Specify “000”

EA3:
0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

EA6–EA4: Specify “001” for external tool offset

Data specification method in 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 ← Offset amount → LSB														

sign: 0 : + ; 1 : –

Offset amount: Specified by BCD 4-digit code (0000 to 7999)

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

EA2:
0 : Specification of tool wear offset amount
1 : Specification of tool geometry offset amount

EA3:
0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

EA6–EA4: 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 PMC. Each axis (parameter Nos.0751 to 0754) 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 makes a new shift amount. The amount that can be input is 0 to ±7.999mm or ±7.999inch.

4) External machine coordinate system

The machine coordinate system can be shifted by inputting 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 the sensor.
The specification method for the axis to be shifted is the same as that for the external workpiece coordinate system shift.
The compensation value is specified for the signals ED0 to ED15 by a binary code ranging from 0 to ±9.999. This compensation value should be absolute and the amount which the machine actually moves on input is the difference from the previously stored value. When a large amount of compensation is applied at a time, an alarm such as ”excessive error on stop” may occur. In this case, input the compensation amount several times.

5) External message

❑ External alarm message
The external alarm message holds the CNC under an alarm condition by sending an alarm number from the external unit, 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, and the CNC displays it with 1000 added. The message for one alarm number can be up to 15 characters long. The alarm condition is reset by external data.

❑ 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 127 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.

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 Table below).
> ED15 - ED8 .... Character code in 1st character.
> ED7 - ED0 .... Character code in 2nd character.
> If 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	\\
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 method 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 Value of aprt nos.		LSB

BCD 4–digit code
(0000 to 9999)

0 : No. of parts required
1 : No. of parts machined

Set to 110000.

Signals

Data signals for external data input ED0 to ED15 <G100, G101>

[Classification] Input signal

[Function] The 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<G102>

[Classification] Input signal

[Function] The signals indicate the type of the entered data.

Read signal for external data input ESTB<G102#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<F160#0>

[Classification] Output signal

[Function] The 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<F160#1>

[Classification] Output signal

[Function] The signal reports that program number search, specified by external data input, has been completed.

[Output condition] The 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0

	#7	#6	#5	#4	#3	#2	#1	#0
F160							ESEND	EREND

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0028				ESR				

[Data type] Bit

ESR External program number search
0 : Disabled
1 : Enabled

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 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

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 <G122#4 to #7>

[Classification] Input signal

[Function] Select the number of a workpiece to be machined in the memory mode. Four code signals are provided. These signals are set as binary code to designate a workpiece number as follows:

Workpiece number search signal				Workpiece number
PN8	PN4	PN2	PN1	
0	0	0	0	00
0	0	0	1	01
0	0	1	0	02
0	0	1	1	03
0	1	0	0	04
0	1	0	1	05
0	1	1	0	06
0	1	1	1	07
1	0	0	0	08
1	0	0	1	09
1	0	1	0	10
1	0	1	1	11
1	1	0	0	12
1	1	0	1	13
1	1	1	0	14
1	1	1	1	15

Workpiece number 00 is used for special designation “no search”. Thus, a workpiece number ranges from 01 to 15.

NOTE
These signals are also used to specify a file number for file search during external program input. See Section 13.3, "External Program Input."

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G122	PN8	PN4	PN2	PN1				

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.

Note

NOTE

1 This function can be used only in memory operation. It cannot be used during manual data input operation.
2 Select the program number from O001 to O015.
3 Program numbers from O001 to O015 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 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 which is searched by 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 PN8 “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 by this function, perform the usual program number search operation (CRT/MDI panel operation), turn all the PN1 to PN8 to “0” and press the start button

15.4 SPINDLE OUTPUT CONTROL BY THE PMC

General

The PMC can control the speed and polarity of each spindle motor, connected by the optional spindle serial output/spindle analog output function.

In the serial output, the first and second spindles 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 the multispindle control function (T series) is added, the CNC can also control the second and 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 an SINDx signal

· Setting the spindle motor speed data, calculated by the PMC, in spindle control signal 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 setting (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 <G145, #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 be obtained easily.

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 require 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 <F185 to F187> and SF signal <F150#2> can be output only when many conditions, determined by the CNC spindle control, are satisfied. In some cases, the signals cannot be used under standard conditions.

Twelve code signals corresponding to the S value (output)

Twelve code signals corresponding to S value R01O to R12O <F172 to F173> 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.0542 and 0543, 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 those 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.0543 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*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 <G110#0 to G111#3> and perform the orientation.

Signal

PMC spindle control signals

For the first spindle: **SIND, SSIN, SGN <G125#7, #6, #5>**
R01I to R12I <G124#0 to G125#3>

For the second spindle: **SIND2, SSIN2, SGN2 <G107#7, #6, #5>**
R01I2 to R12I2 <G106#0 to G107#3>

For the third spindle: **SIND3, SSIN3, SGN3 <G109#7, #6, #5>**
R01I3 to R12I3 <G108#0 to G109#3> **(for the multi–spindle function is T series)**

[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.

In the serial output, even if the multispindle control function is not provided, these signals allow the second spindle to be controlled.

When the multispindle control function and type A are being used (if the MSPDB bit, bit 3 of parameter No. 0070, 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.

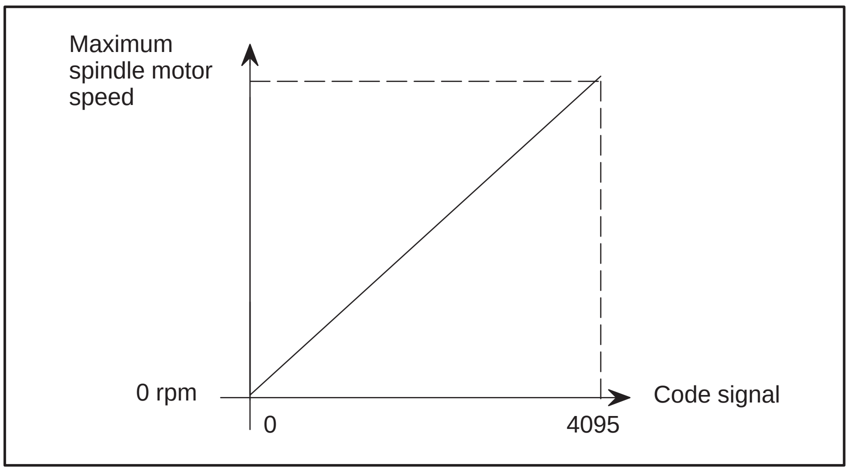
0: The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- 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.0013) 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.

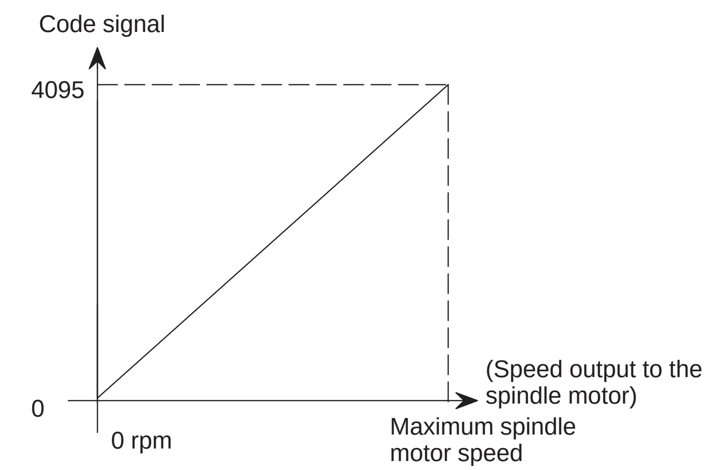
Twelve code signals corresponding to the S value R01O to R12O <F172#0 to F173#3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output to the spindle motor that is required to control the connected spindle. The converted value is sent to the PMC with twelve code signals, in proportion 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.)



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G106	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2	
G107	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2	
G108	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3	(T Series)
G109	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3	(T series)
G124	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I	
G125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I	

CAUTION
Note that signals G106 to G109 are also used for external key input.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0	
0020	SFOUT								(M series)

[Data type] Bit

SFOUT 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	
0029				SFOB					(M series)

[Data type] Bit

SFOB When an S code command is issued in constant surface–speed control,
0 : SF is output.
1 : SF is not output.

	#7	#6	#5	#4	#3	#2	#1	#0	
0049								EVSF	(T series)

[Data type] Bit

EVSF 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.

	#7	#6	#5	#4	#3	#2	#1	#0	
0070					MSPDB				(T series)

[Data type] Bit

MSPDB 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.)

	#7	#6	#5	#4	#3	#2	#1	#0
0071				SRL2SP				

[Data type] Bit

SRL2SP The number of connections in serial spindle control
0 : 1
1 : 2

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 MSPDB bit, bit 3 of parameter No. 0070, is set to 0) when the multispindle control function is used.

   → The second or third spindle 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 <G145, #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.0013, 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 or third spindle, see Sections 9.2 and 9.10.

4 If the multispindle control function is not being used, 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”.

15.5 EXTERNAL KEY INPUT

General

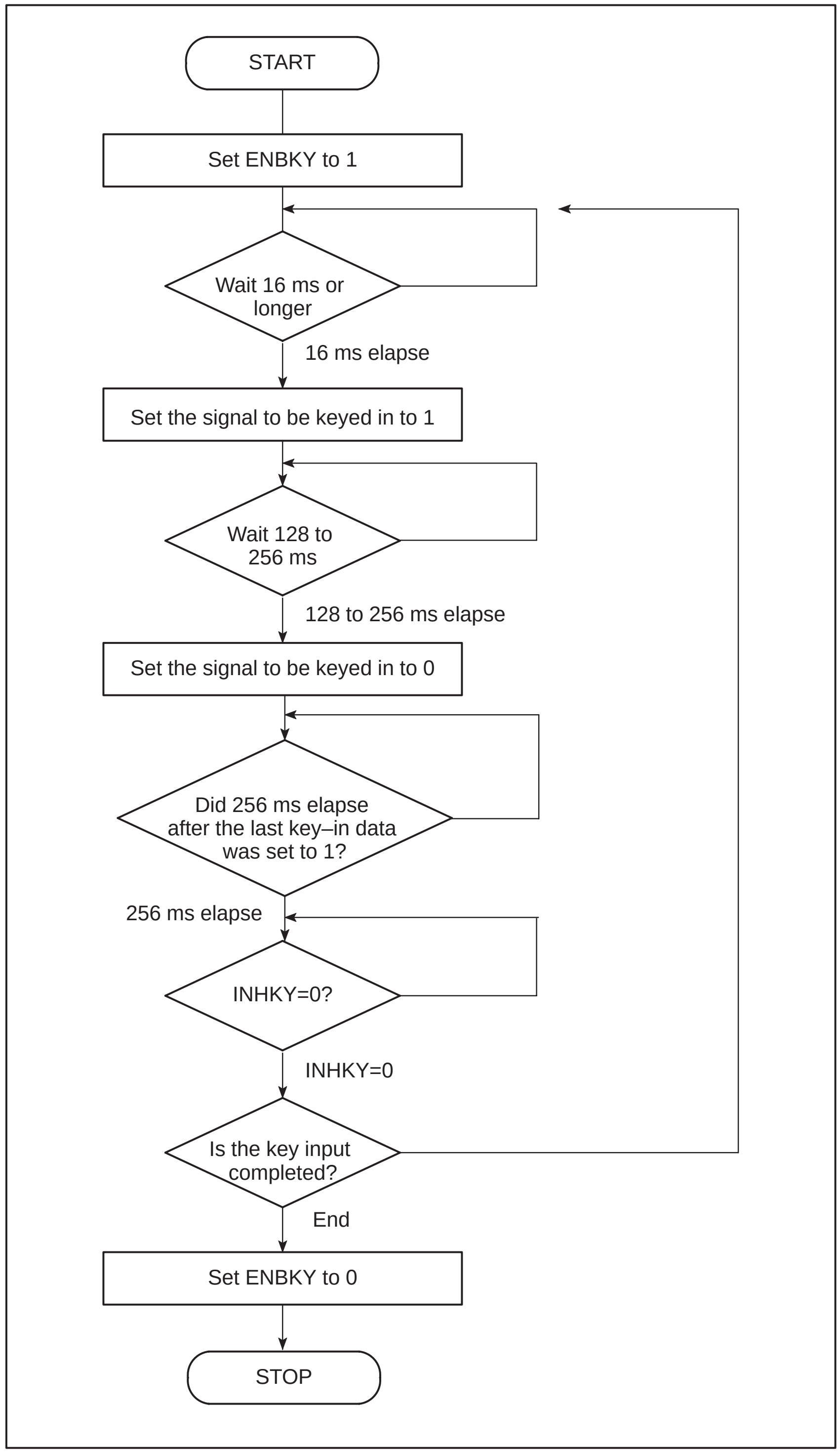
MDI key codes can be sent from the PMC to CNC by means of interface signals. This allows the CNC to be controlled in the same way as when the operator performs MDI key operation.

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)	
Key input disable signal (output)	INHKY
Program screen display mode signal (output)	PRGDPL

The processing flow in the PMC is shown below.

Flowchart of PMC program processing at key input



Signal

External key input mode selection signal ENBKY <G134#0>

[Classification] Input signal

[Function] While the signal is held to 1, the external key input control function is enabled. Meanwhile, the CRT/MDI key operation is ignored.

Key signal <G106 to G114>

[Classification] Input signal

[Function] Sets the input key.

Key input disable signal INHKY <F180#0>

[Classification] Output signal

[Function] While the signal is held to 1, any key code of external key input is not accepted.

Program screen display signal PRGDPL <F180#1>

[Classification] Output signal

[Function] The signal is set to 1 while the CNC is displaying the program screen.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0	
G106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S	(T series)
G107	KEOB,	KH,1,K	KA,C	KP,Q	KT,.	KM,–	K9,G	K8,N	(T series)
G108				KSTART	KINOUT	KDELET	KINSRT	KALTER	(T series)
G109			KALARM	KDGNOS	KPARAM	KOFSET	KPRGRM	KPOS	(T series)
G110	KCAN			KXZ	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓	(T series)
G111	KRESET								(T series)
G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR		(T series)
G113	KFWH	KFUV	KFZJ	KFXY	KFRC	KFGB	KFN)	KFO(	(T series)
G114	KFL+	KFT*	KFS=	KFM#	KFF	KFE	KFK@	KFI,	(T series)
G142	KXZ								(T series)
G106	K7,0	K6,Z	K5,Y	K4,X	K3,R	K2,F	K1,H	K0,X	(M series)
G107	KEOB,/,#	KH,J,K	KB,K4,D	KP,Q,L	KT,.	KM,–	K9,G	K8,N	(M series)
G108				KSTART	KINPUT	KDELET	KINSRT	KALTER	(M series)
G109			KALARM	KDGNOS	KPARAM	KOFSET	KPRGRM	KPOS	(M series)
G110	KCAN	KYZ	KZX	KXY	KPAGE↑	KPAGE↓	KCUS↑	KCUS↓	(M series)
G111	KRESET								(M series)
G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR		(M series)
G113	KF4TH	KFZW	KFYV	KFXU	KFRC	KFGE	KFN)	KFO(	(M series)
F114	KFL+	KFt*	KFS=	KFM#	KFF	KFF@	KFJA	KFI,	(M series)
G134								ENBKY	
F180							PRGDPL	INHKY	

Correspondence between the MDI keys and signals

Standard CRT/MDI key switch

Signal name	Key		Signal name	Key
K0.S	0 S		KI.H.K	K I H (0T)	K J I (0M)
K1.U	1 U (0T)	1 U (0M)	KEOB./	/ # EOB
K2.W	↓ 2 W (0T)	↓ 2 F (0M)	KALTER	ALTER
K3.R	3 R (0T)	3 R (0M)	KINSRT	INSRT
K4.X	← 4 X		KDELET	DELET
K5.Z	5 Z (0T)	5 (0M)	KINPUT	INPUT
K6.F	→ 6 F (0T)	→ 6 Z (0M)	KSTART	OUTPT START
K7.O	7 O (0T)	7 O (0M)	KPOS	POS
K8.N	↑ 8 N		KPRGRM	PRGRM
K9.G	9 G (0T)	9 G (0M)	KOFSET	MENU OFSET
KM.–	— M		KPARAM	DGNOS PARAM
KT.&	. T		KGRAPH	AUX GRAPH
KP.Q	No. J Q P (0T)	No. L Q P (0M)	KALARM	OPR ALARM
KA.C	← A C (0T)	← 4th B (0M)	KCUS ↓	↓ Cursor key

Standard CRT/MDI key switch

Signal name	Key		Signal name	Key
KCUS ↑	⬆	Cursor key	KCAN	CAN
KPAGE ↓	⬇	Page key	KRESET	RESET
KPAGE ↑	⬆	Page key	KXZ	Signal for entering addresses X and Z simultaneously (can be used in playback)

When a key signal is output from the PMC to the CNC, the following three control signals are used. The signals are descried below. The flowchart of key input by PMC program is shown below.

ENBK Key input enable signal
(PMC⇒CNC)

INHKY Key input disable signal
(CNC⇒PMC)

PRGDPL Signal indicating that the program screen is displayed on the CRT screen
(CNC⇒PMC)
(The signal is held to 1 while the program screen is being selected.)

CAUTION
When ENBKY is set to 1, the signal of the CRT/MDI key switch is ignored.

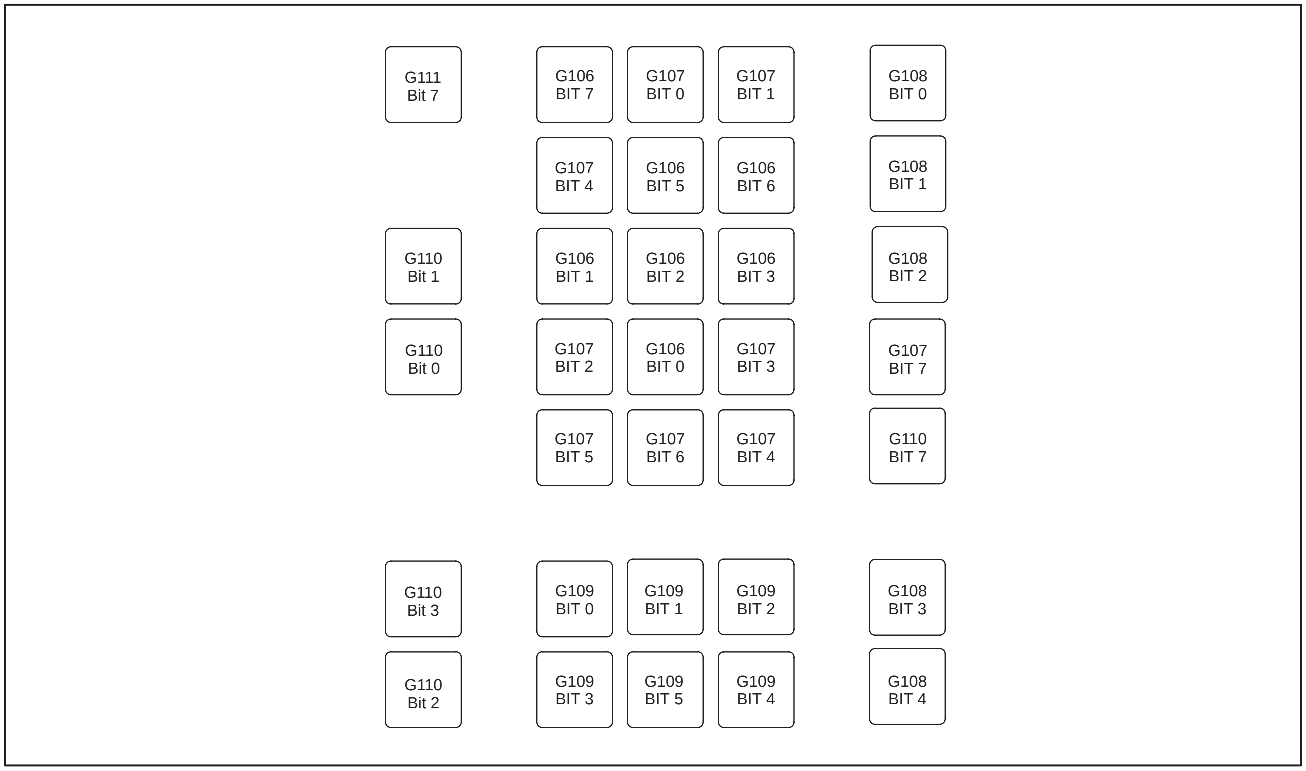


Fig.15.5 (a) Correspondence between Keys and Key Addresses (Standard Keyboard)

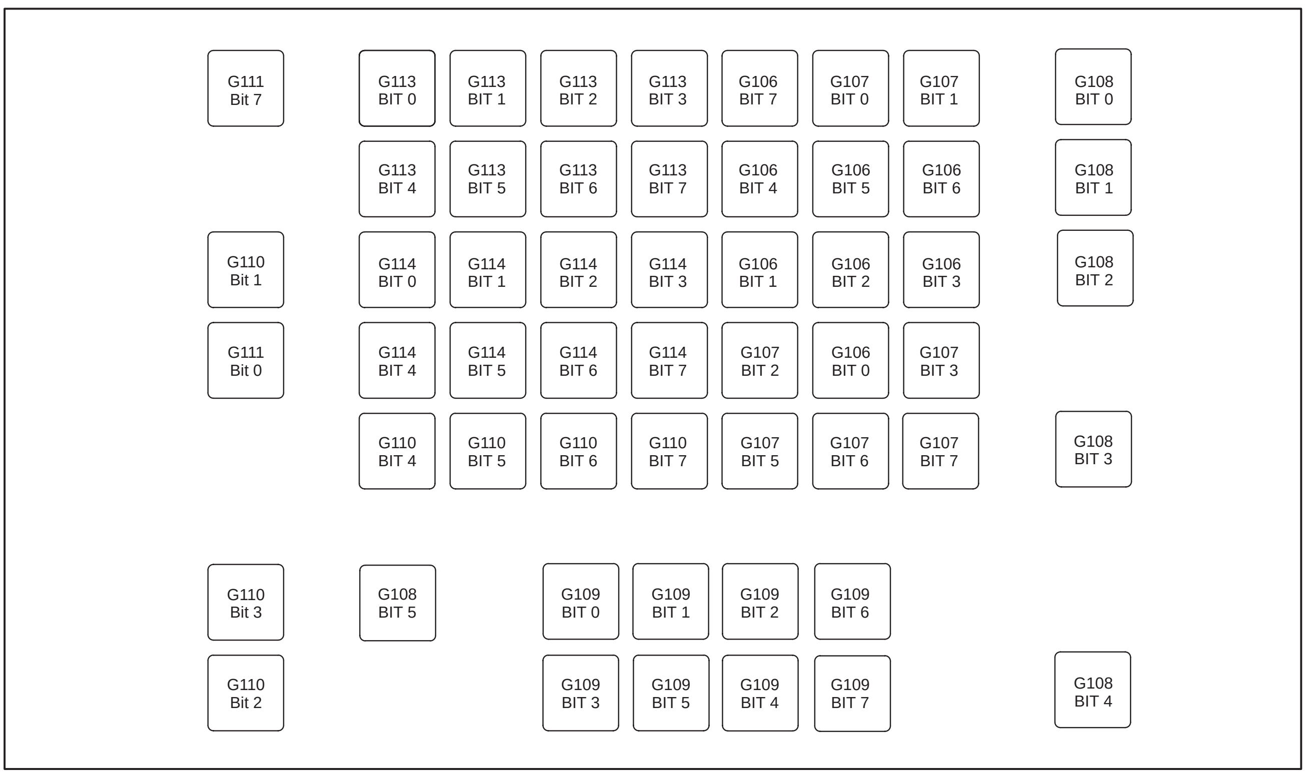


Fig.15.5(b) Correspondence between Keys and Key Addresses (Full Keyboard)

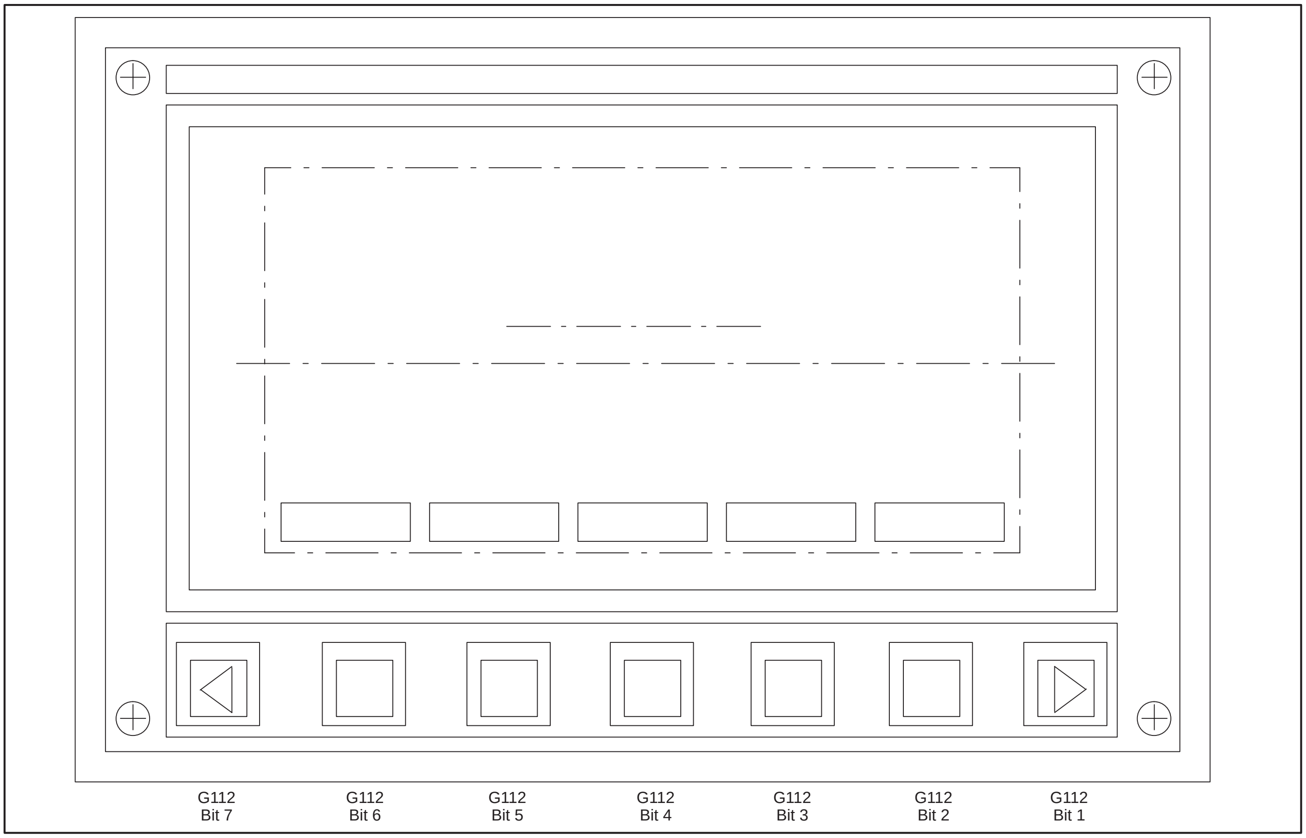


Fig.15.5(c) Correspondence between Keys and Addresses (Soft Keys)

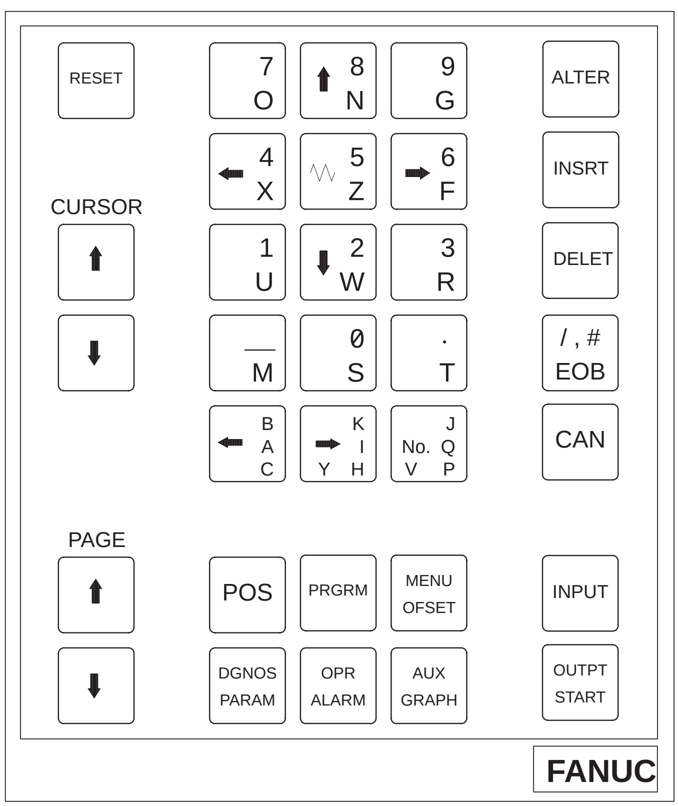


Fig.15.5(d) MDI Keys of the Series 0–TC (Standard Keyboard)

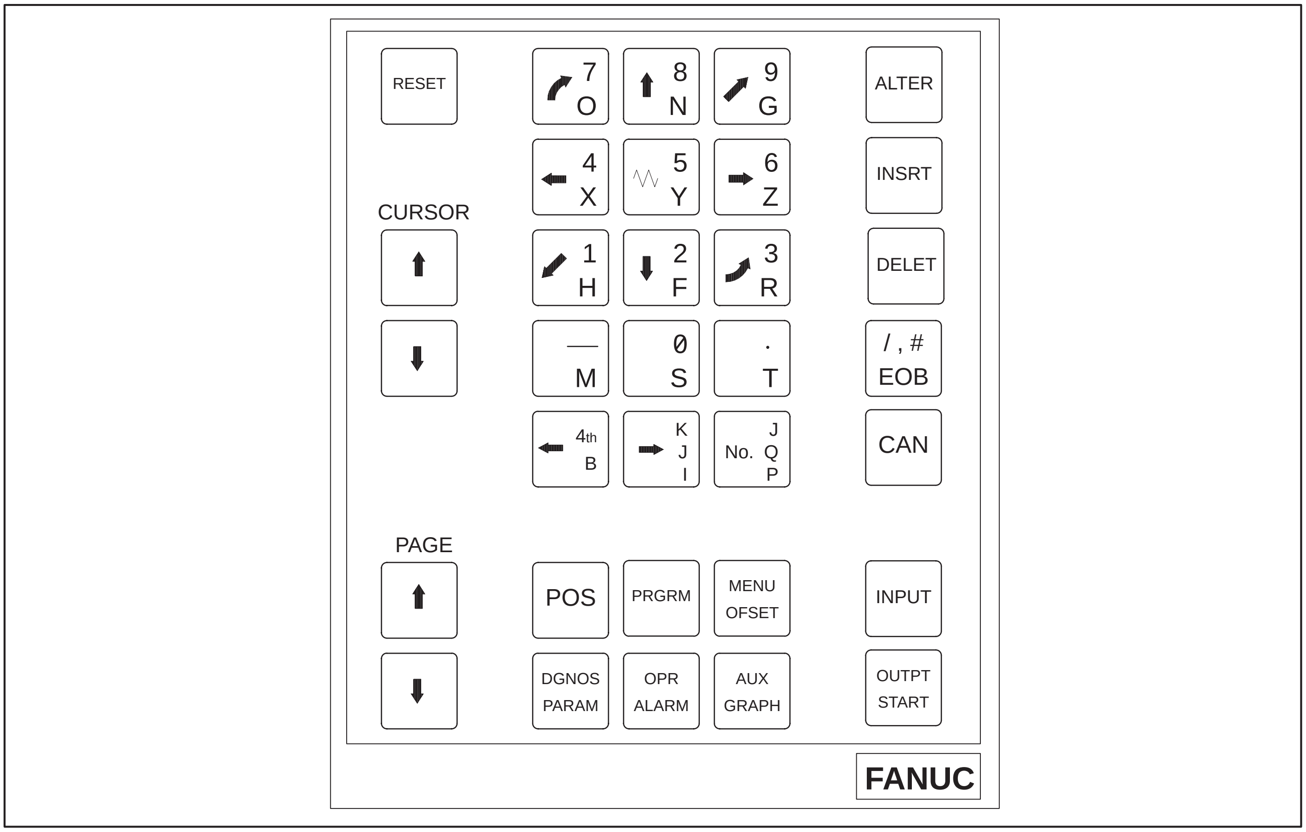


Fig.15.5(e) MDI Keys of the Series 0–MC (Standard Keyboard)

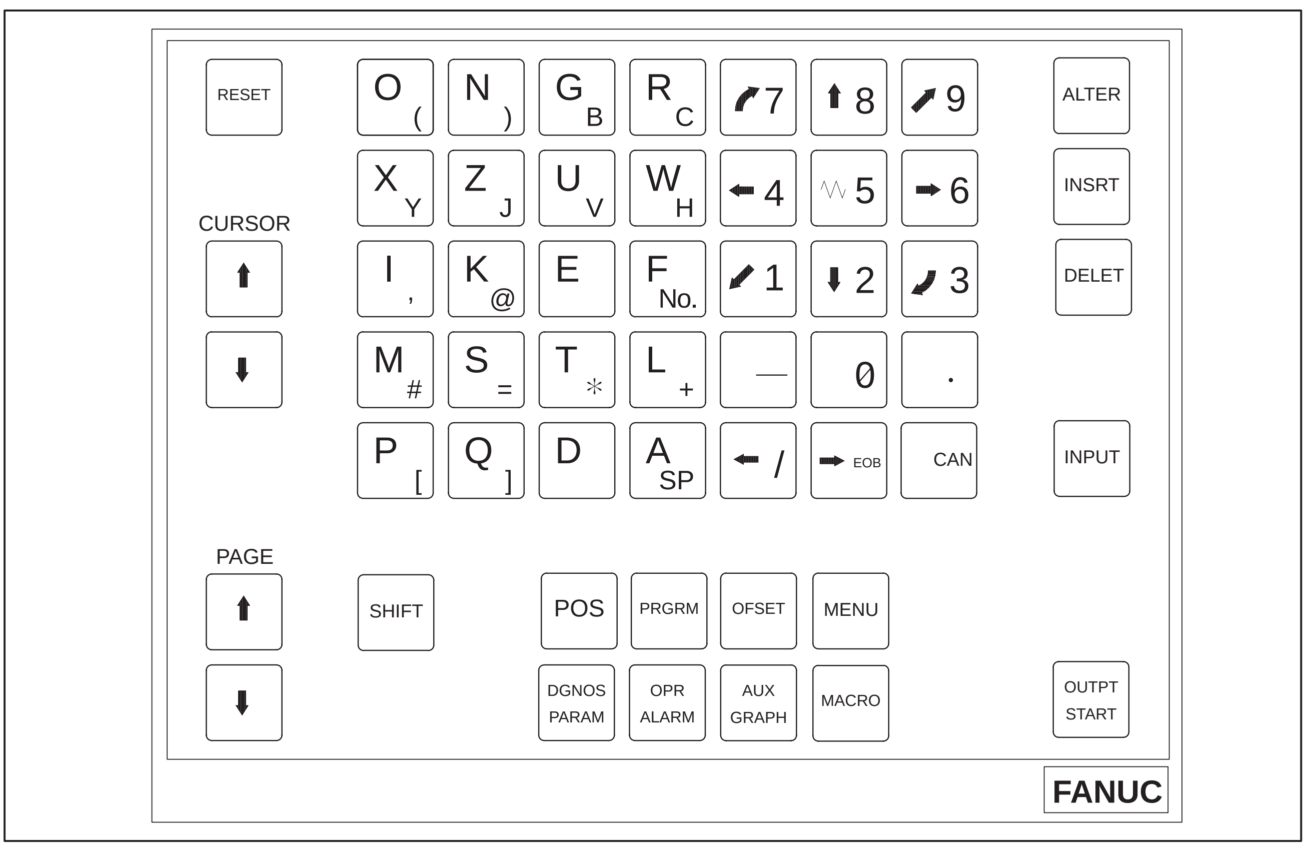


Fig.15.5(f))MDI Keys of the Series 0–TC (Full Keyboard)

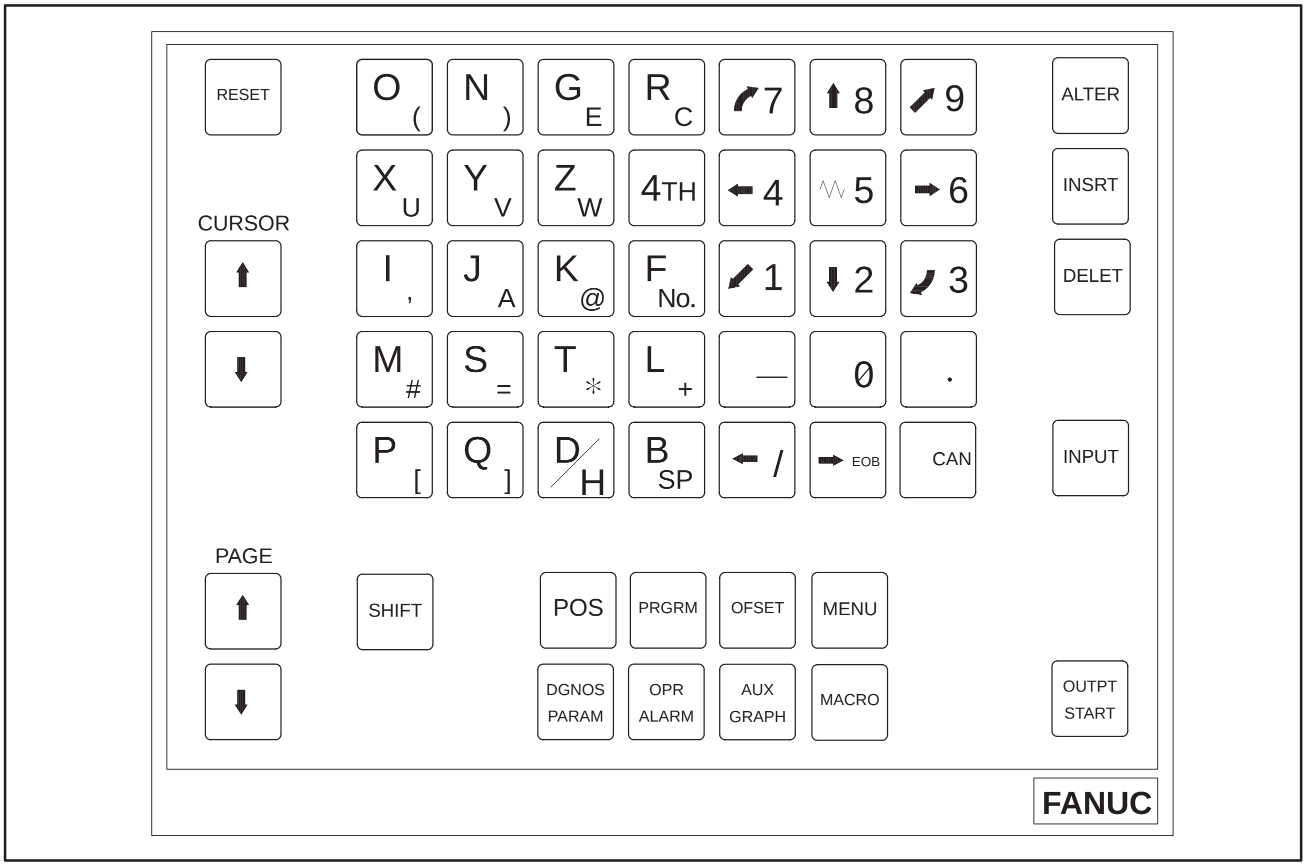


Fig.15.5(g) MDI Keys of the Series 0–MC (Full Keyboard)

15.6 DIRECT OPERATION BY MMC (T SERIES)

General

Activating memory operation in memory operation mode (AUTO) with the direct operation select signal set to 1 enables machining (direct operation) while reading a program stored in the MMC.

Signal

Direct operation select signal DMMC <G128#7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program stored in the MMC.

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

· When memory operation mode (AUTO) is not selected, the control unit ignores this signal.

· When memory operation mode (AUTO) is selected, the control unit selects direct operation mode and enables direct operation.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G128	DMMC							

15.7 PMC WINDOW FUNCTION

General

This function allows the PMC to read and rewrite CNC data.

Signal

Window request signal WNRQ<G200#0>

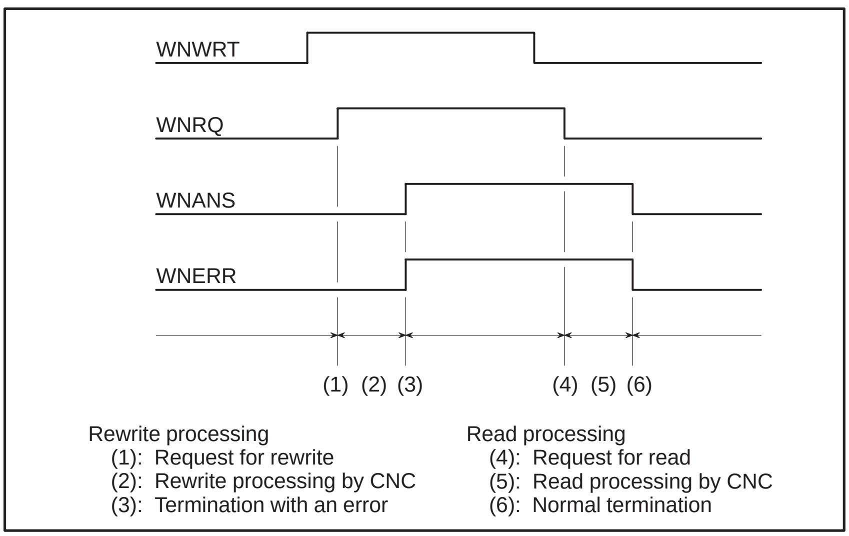
[Classification] Input signal

[Function] See the description of window completion signal WNANS, below.

Window completion signal WNANS<F250#0>

[Classification] Output signal

[Function] Window request signal WNRQ is sent, by the PMC, to request the CNC to read/write data. A request is made by placing the signal in a state other than the signal state of window completion signal WNANS. By request, the CNC reads or writes data, then posts the completion of the requested processing by placing WNANS in the same signal state as that of WNRQ.



Window rewrite signal WNWRT<G200#1>

[Classification] Input signal

[Function] When data in the CNC is read (from CNC to PMC; output), this signal is set to 0. The read data is output to F252 to F267.
When data in the CNC is rewritten (from PMC to CNC; input), this signal is set to 1. The data to be written is set in G206 to G209.

Window error signal WNERR<F250#1>

[Classification] Output signal

[Function] When data passed to the window contains an error, completion notification is made using WNANS, and WNERR is set to 1. When the data contains no errors, WNERR is set to 0. Therefore, the PMC must check to ensure that WNERR is 0 after checking that completion signal WNANS has been inverted.

List of functions

Function No.	Data name	Function No.	Data name
0		20	Distribution information
1	Absolute coordinate system	21	A/D conversion data
2	Machine coordinate system	22	Output of axis movement signal
3	Skip coordinate system	23	In–position information
4	Servo delay amount	24	Distribution information
5		25	Load information of serial spindle
6		26	Speed information of serial spindle
7	Distribution amount	27	Alarm information of serial spindle
8		28	
9		29	
10	Alarm information	30	
11		31	D/A conversion function
12		32	Torque limit
13		33	Output of modal data
14	Program number of the current selection	34	
15	Output of the servo load current value	35	
16	Output of the number of required parts	36	
17	Output of the number of machined parts	37	
18	Output of date	38	
19	Output of time	39	

* Those function numbers for which the data name fields are blank must not be used.

Function details

- **Functional Nos.01,02,03 : Current Position**

(1) Function
Enables output of the current position.
The following three factors can be obtained with three functional numbers:
1 : absolute coordinates
2 : machine coordinates
3 : skip position

(2) Data content
Type : binary
Data length : 2 words (for one axis)

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	1 to 3
G203	Word number of data	2 to 8
G204	Axis selection code	0 to 20
G205	00	0

Example)
To obtain the data of the 2nd axis, set the word number of data to “2” and the axis selection code to “4”. To obtain the data for several axes at a time, set the word number of data to “number of axis × 2” and the axis selection code to the value for the first axis.

Axis Selection Code)

		M series	T series
0	:	X axis	X axis
4	:	Y axis	Z axis
8	:	Z axis	3rd axis
12	:	4th axis	4th axis
16	:	5th axis	5th axis
20	:	6th axis	6th axis

- **Functional No.04 : Servo Delay Amount**

(1) Function
Enables reading of the current servo delay amount.

(2) Data content
Type : binary
Data length : 2 words (for one axis)

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	4
G203	Word number of data	2 to 8
G204	Axis selection code	0 to 20
G205	00	0

Example)

To obtain the data of the 2nd axis, set the word number of data to “2” and the axis selection code to “4”. To obtain the data for several axes at a time, set the word number of data to “number of axis × 2” and the axis selection code to the value for the first axis.

Axis Selection Code)

		M series	T series
0	:	X axis	X axis
4	:	Y axis	Z axis
8	:	Z axis	3rd axis
12	:	4th axis	4th axis
16	:	5th axis	5th axis
20	:	6th axis	6th axis

- **Functional No.07 : Distribution Amount**

(1) Function

Enables reading of the distribution amount of the current axis at every interpolation cycle.
Window function is useful to discriminate the direction and the speed of the movement in constant moving.
Please note that a time delay inevitably occurs when reading is performed by PMC.

(2) Data content

Type : binary
Data length : 2 words (for one axis)

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	7
G203	Word number of data	2 to 8
G204	Axis selection code	0 to 20
G205	00	0

Example)

To obtain the data of the 2nd axis, set the word number of data to “2” and the axis selection code to “4”. To obtain the data for several axes at a time, set the word number of data to “number of axis × 2” and the axis selection code to the value for the first axis.

Axis Selection Code)

		M series	T series
0	:	X axis	X axis
4	:	Y axis	Z axis
8	:	Z axis	3rd axis
12	:	4th axis	4th axis
16	:	5th axis	5th axis
20	:	6th axis	6th axis

- **Functional No.10 : Alarm Information**

(1) Function
Enables reading of the current alarm information.

(2) Data content
Type : bit
Data length : 1 byte

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	10
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
Data can be obtained in a bit format.

	#7	#6	#5	#4	#3	#2	#1	#0
F252		SV	OH	OTS	PS	PS3	PS2	PS1

PS1 P/S alarm 100 (PWE ON)
PS2 P/S alarm 000 (power off demand)
PS3 P/S alarm 101 (program area deletion demand)
PS P/S alarm
OTS Over travel alarm
OH Over heat alarm
SV Servo alarm

- **Functional No.14 : Program Number of the Current Selection**

(1) Function
Enables reading of the program number that is currently selected.

(2) Data content
Given by 2–byte size.
The range of value is 0 through 9999.

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	14
G203	Word number of data	1
G204	00	0
G205	00	0

● Functional No.15 : Output of the Servo Load Current Value

(1) Function

Enables reading of the output of load current value with sign of servo.

(2) Data content

Given by a binary format specified by 2–byte size for one axis.

Conversion of the load current value is given by the following formula:

$$\text{Load current value [Apeak]} = D \times \frac{5}{256} \times \frac{\lambda}{128}$$

Where, D : read value in the PMC window

λ : motor model,

40 [Apeak] for 0, 5, 10, 20

80 [Apeak] for 20, 30

(3) Usage

1) Classification

Output

2) Control data

G202	Functional binary number	15
G203	Word number of data	1
G204	Axis selection code	2 to 74
G205	00	0

Example)

To obtain the data of the 2nd axis, set the word number of data to “1” and the axis selection code to “26”. The data for several axes can not be obtained at the same time with this window.

Axis Selection Code)

	M series	T series
2 :	X axis	X axis
26 :	Y axis	Z axis
50 :	Z axis	3rd axis
74 :	4th axis	4th axis

● Functional No.16 : Output of Number of Required Parts

(1) Function

Enables reading of the number of part required in the display of the machining parts number of CNC.

(2) Data content

Type : binary

Data length : 1 word

Data range : 0 through 32767

(3) Usage

1) Classification

Output

2) Control data

G202	Functional binary number	16
G203	Word number of data	1
G204	00	0
G205	00	0

- **Functional No.17 : Output of Number of Machined Parts**

(1) Function
Enables reading of the machined parts number in the display of the machined parts number of CNC.

(2) Data content
Type : binary
Data length : 2 words
Data range : 0 through 99999999

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	17
G203	Word number of data	2
G204	00	0
G205	00	0

- **Functional No. 18 : Output of data**

(1) Function
Enables reading the date of the timer built into the CNC.

(2) Data content
Type : binary
Data length : 2 words
Data format : 19970330 for 97/03/30
(March 30th., 1997)

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	18
G203	Word number of data	2
G204	00	0
G205	00	0

- **Functional No. 19: Output of time**

(1) Function
Enables reading the time of the timer built into the CNC.

(2) Date content
Type : binary
Date length : 2 words
Data format : 93045 for 09:30:45

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	19
G203	Word number of data	2
G204	00	0
G205	00	0

- **Functional No.20 : Distribution Information**

(1) Function
Enables reading of the following distribution information.

(2) Data content
Type : bit
Data length : 1 byte

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	20
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
The data can be obtained in a bit format.

	#7	#6	#5	#4	#3	#2	#1	#0
F252	MTN	DWL	AF	CYL				ORGN

ORGN 1 : block for setting coordinate system.
0 : block not for setting coordinate system.

CYL 1 : in a cycle operation.
0 : out of a cycle operation.

AF 1 : with auxiliary function
0 : without auxiliary function

DWL 1 : with dwell
0 : without dwell

MTN 1 : with distribution
0 : without distribution

- **Functional No.21: A/D Conversion Data**

(1) Function
Enables reading of four channels of analog signals (voltages) in the PMC window with an A/D converter.

(2) Data content
Input range : –10V to + 10V
(input unit: 5mV)
Type : binary
Date length : 1 word
Data range : –1999 throuth +2000

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	21
G203	Word number of data	1 to 4
G204	Channel selection code	0 : 1 channel 2 : 2 channels 4 : 3 channels 6 : 4 channels
G205	00	0

Example)

To obtain the data of the 2nd channel, set the word number of data to “1” and the channel selection code to “1”. To obtain all the channel selection codes, set the word number of data to “4” and the channel selection code to “0”.

Remarks)

A/I board is necessary for the A/D conversion.

- **Functional No.22 : Output of Axis Movement Signal**

(1) Function
Enables reading if the current axis is moving.

(2) Data content
Type : bit
Data length : 1 byte

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	22
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)

The data can be obtained in a bit format.

	#7	#6	#5	#4	#3	#2	#1	#0	
F252					MV4	MV3	MVZ	MVX	(T series)
					MV4	MVZ	MVY	MVX	(M series)

MVx Represents the movement signal of X, Y (Z), Z (3) and 4th axis successively. Each axis is moving when each bit is 1.

- **Functional No.23 : In–position Information**

(1) Function
Enables reading if an axis is in the in–position.

(2) Data content
Type : bit
Data length : 1 byte

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	23
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
The data can be obtained in a bit format.

	#7	#6	#5	#4	#3	#2	#1	#0	
F252					INP4	INP3	INPZ	INPX	(T series)
					INP4	INPZ	INPY	INPX	(M series)

INPx Represents the in–position of X, Y (Z), Z (3) and 4th axis successively. Each axis is in the in–position when each bit is 1.

- **Functional No.24 : Distribution Information**

(1) Function
Enables reading of the information related to the distribution of CNC.

(2) Data content
Type : bit
Data length : 1 byte

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	24
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
The data can be obtained in a bit form.

	#7	#6	#5	#4	#3	#2	#1	#0
F252	AUTO	@POS	HND			ZRNM	G30	G27
			CIR	G03	SCRW			

AUTO 0 : pulse distribution in a manual mode (jog, step, etc.)
1 : pulse distribution in an automatic mode (AUTO, MDI, etc.)

@POS 0 : pulse distribution in a position format.
1 : pulse distribution in an interpolation format.

The content of the information of bit 0 through 5 varies according to the state of @POS as follows:

In the case of @POS = 0 ;

HND 0 : pulse distribution other than a handle.
1 : pulse distribution of a handle.

ZRNM 0 : non reference position return
1 : reference position return

G30 0 : out of 2nd reference position returning
1 : in 2nd reference position returning

G27 0 : non reference position return check
1 : reference position return check

In the case of @POS = 1;

CIR 0 : linear interpolation
1 : circular interpolation

G03 0 : G02 (significant only when CIR = 1)
1 : G03

SCRW 0 : out of thread cutting
1 : in thread cutting

• Functional No.25 : Load Information of Serial Spindles

(1) Function
Enables reading of the load information of serial spindles.

(2) Data content
Type : binary
Data length : 1 word
Data range : 0 through 32767

(3) Usage

1) Classification
Output

2) Control data

G202	Functional binary number	25
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
For reference, the formula to normalize the load information is shown as follows:

$$\text{Load (\%)} = \frac{L}{32767} \times \lambda$$

where , L is the value read from the window,
λ is the ratio of maximum output to the continuous rated output of a motor (when the continuous rated output is regarded as 100%, λ for 180% of the maximum output becomes 180).

• Functional No.26 : Speed Information of Serial Spindles

(1) Function
Enables reading of the speed information of serial spindles.

(2) Data content
Type : binary
Data length : 1 word
Data range : 0 to ±16383

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	26
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
For reference, the formula to obtain the rotation number from the speed information is shown as follows:
Rotation Number (rpm)

$$= \frac{|V|}{16383} \times \text{(Maximum rotation number of the motor)}$$

where, V is the value read from the window,
$|V|$ is the absolute value of V.

• Functional No.27 : Alarm Information of Serial Spindles

(1) Function
Enables reading of the alarm information of serial spindles.

(2) Data content
Type : binary
Data length : 1 word

(3) Usage
1) Classification
Output
2) Control data

G202	Functional binary number	27
G203	Word number of data	1
G204	00	0
G205	00	0

Remarks)
The alarm code and the content of alarms are shown as follows:

Code	Content of alarm
0	Normal operation
1	Motor over heat
2	Velocity error excess
4	Phase lack alarm
7	Over speed

Code	Content of alarm
9	Over load
10	Low voltage in control power supply
11	Over voltage in DC linking part
12	Over current in DC linking part
13	Internal memory abnormality of CPU
14	ROM alarm
18	Sumcheck alarm of program memory (ROM)
19	Excessive offset in U phase current detection circuit
20	Excessive offset in V phase current detection circuit
24	Serial data transmission alarm
25	Serial LSI watchdog
26	Burn–out alarm of 9000p speed detector
27	Burn–out alarm of position coder
28	Burn–out alarm of 9000p position detector
29	Over load
30	Over current in input circuit
31	Motor binding alarm
32	Internal RAM abnormality in serial LSI
33	Undercharge alarm at DC linking part
34	Parameter data malsetting
35	Disagreement in gear ratio setting
36	Error counter overflow
37	HCRDY handshake alarm

- **Functional No. 31: D/A Conversion Function**

(1) Function
Enables conversion of the data input from the PMC window to the analog signal (voltage) to 2 channels with a D/A converter.

(2) Data content
Type : binary
Data length : 1 word
Input range : –7168 to +7168
Output range : –11.0V to +11.0V

(3) Usage

1) Classification
Input

2) Control data

G202	Functional binary number	32
G203	Word number of data	1
G204	Channel selection code	0 to 1
	00	0
G205 to G206	D/A conversion data	–7168 to 7168

Channel Selection Data)

Set value : 0 : connector M121
1 : connector M121 and M122

D/A CONVERSION DATA)

10V, 0V and –10V are generated by the set value of 6553, 0 and –6553 respectively.

Example)

To output the data of the 2nd channel, set channel selection code to “1”.

Remarks)

A/I board is necessary for the use of D/A conversion.
Data for only 1 channel can be input at the same time.
The channel to use the D/A conversion can not used for multiples spindles.

- **Reference parameters:**

	#7	#6	#5	#4	#3	#2	#1	#0
0070							DAC2	DAC1

DAC1 1 : Channel 1 is used for the D/A
0 : Channel 1 is not used for the D/A conversion.

DAC2 1 : Channel 2 is used for the D/A conversion.
0 : Channel 2 is not used for the D/A conversion.

0613	PSANG2	(T series)
0673	PSANG2	(M series)

PSANG2 Sets data for the gain adjustment in an analog output.
(channel 1)
Setting range : 700 to 1250
Standart set value : 1000

0614	SPDLC2	(T series)
0674	SPDLC2	(M series)

SPDLC2 Sets the offset compensation value, i.e. the zero offset compensation value of output voltage.
(channel 1)
Setting range : 0 to ±1023
Standard set value : 0

0617	PSANG3	(T series)
0677	PSANG3	(M series)

PSANG3 Sets data for the gain adjustment in an analog output.
(Channel 2)
Setting range : 700 to 1250
Standard set value : 1000

0618	SPDLC3	(T series)
0678	SPDLC3	(M series)

SPDLC3 Sets the offset compensation value, i.e. the zero offset compensation value of output valtage.
(Channel 2)
Setting range : 0 to ±1023
Standard set value : 0

- **Functional No.32 : Torque Limit**

(1) Function
Enables setting of the torque limit to a servo motor from the PMC window.

(2) Data content
Torque limit data
Type : binary
Data length : 1 word
Input range : 0 to 255
0 for 0%.
255 for 100%.

(3) Usage
1) Classification
Input
2) Control data

G202	Functional binary number	32
G203	Word number of data	1
G204	Axis selection code	0 to 5
G205	00	0
G206 to G209	Torque limit data	0 to 255

Axis Selection Code)
Set value : 0 to 5

		M series	T series
0	:	X axis	X axis
1	:	Y axis	Z axis
2	:	Z axis	3rd axis
3	:	4th axis	4th axis
4	:	5th axis	5th axis
5	:	6th axis	6th axis

Example)
To set 50% of the torque limit to 2nd axis, set the axis selection code to “1” and the torque limit data to “128”. To cancel the torque limit, set the torque limit data to “255”.

Remarks)
When the number exceeding the maximum controlled axis number is set, an error occurs.

- **Functional No.33 : Output of Modal Data**

(1) Function

Enables reading of the modal information of CNC from PMC window.

(2) Data content

The modal data of G function:

The read modal data of G function is set in F252 by a binary code in the following format:

	#7	#6	#5	#4	#3	#2	#1	#0
F252								

(#7) 1 : represents that the read modal data is commanded in a current block of the part program.

0 : represents that the read modal data is not commanded in a current block of the part program.

(#6 to #0) The modal data is shown by code.

Modal data other than G function:

The eleven kinds of data, the addresses D, H, L, M, N, O, S, T, F, of the NC part program and 2nd auxiliary function, can be read as modal data other than G function.

The read data is set in the area of 5 byte F252 to F256 as follows:

	#7	#6	#5	#4	#3	#2	#1	#0
F252								
F253								
F254								
F255								

The read data is set in a binary format.

	#7	#6	#5	#4	#3	#2	#1	#0
F256			±					

(#3 to #0) No. of digit

(#5) 1 : Negative command value

0 : Positive command value

(#7) 1 : represents that the read modal data is commanded in a current block of the part program.

0 : represents that the read modal data is not commanded in a current block of the part program.

(3) Usage

1) Classification

Output

2) Control data

G202	Functional binary number	33
G204 to G205	Modal Data Group number	100 to 116, 200 to 216 124 to 133, 224 to 233

Remarks)

The correspondence between the numbers to be specified to G204 to G205 and the modal data is shown as follows:
Either number “1” (current) or “2” (next) is specified to the blank space of each 3rd digit according to the necessary modal data.
The G code in a lathe system is represented on the basis of B in the G code system.
For example, G32 in the G code system A corresponds to G33 in the G code system B and the code read to F252 becomes 4.

Number Correspondence Table of G Code Group

Group number	G code in the machining system	Code	G code in the lathe system	Code
□00	G00 G01 G02 G03 G33	0 1 2 3 4	G00 G01 G02 G03 G33 G34 G77 G78 G79	0 1 2 3 4 8 5 6 7
□01	G17 G18 G19	0 8 4	G96 G97	1 0
□02	G90 G91	0 1	G90 G91	0 1
□03	G22 G23	1 0	G68 G89	1 0
□04	G94 G95	0 1	G94 G95	0 1
□05	G20 G21	0 1	G20 G21	0 1
□06	G40 G41 G42	0 1 2	G40 G41 G42	0 1 2
□07	G43 G44 G49	1 2 0	G25 G26	0 1
□08	G73 G74 G76 G80 G81 G82 G83 G84 G85 G86 G87 G88 G89	10 11 12 0 1 2 3 4 5 6 7 8 9	G22 G23	0 1

Group number	G code in the machining system	Code	G code in the lathe system	Code
□09	G98 G99	0 1	G80 G83 G84 G86 G87 G88 G89	0 1 2 3 5 6 7
□10	G50 G51	0 1	G98 G99	0 1
□11	G66 G67	1 0	G66 G67	1 0
□12	G96 G97	1 0		
□13	G54 G55 G56 G57 G58 G59	0 1 2 3 4 5		
□14	G61 G62 G63 G64	1 2 3 0		
□15	G68 G69	1 0	G17 G18 G19	0 4 8
□16	G15 G16	0 1		

Number Correspondence Table of Modal Data Group Other than G Code

Modal data other than the G function	Machine system	Lathe system	Area where the data is read
D	□24	–	F252 to F255
H	□25	–	
L	□26	□24	
M	□27	□25	
N	□28	□26	
O	□29	□27	
S	□30	□28	
T	□31	□29	
F	□32	□30	
B	□33	□31	

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G200							WNWRT	WNRQ

G202	DI for PMC window (Function number)
G203	DI for PMC window (Number of word of data)
G204	DI for PMC window
G205	DI for PMC window
G206	DI for PMC window
to	to
G209	DI for PMC window

	#7	#6	#5	#4	#3	#2	#1	#0
F250							WNERR	WNANS

F252	DO for PMC window
to	to
F267	DO for PMC window

APPENDIX

A INTERFACE BETWEEN CNC AND PMC

A.1 LIST OF ADDRESSES

List of address (M series)

MT → PMC

	#7	#6	#5	#4	#3	#2	#1	#0
X004	4NG							
X008	SKIP	ESKIP	*RILK			ZAE	YAE	XAE
X016			*DECX		*DEC5S	*DEC7		
X017			*DECY		*DEC6S	*DEC8		
X018			*DECZ					
X019			*DEC4					
X020			*–LZ	*–LY	*–LX	*+LZ	*+LY	*+LX
X021				*ESP				

PMC → CNC

	#7	#6	#5	#4	#3	#2	#1	#0
G100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G103	AFL	ZNG	SPC	SPB	SPA			SRN
G104		RRW	*FLWU	ESRSYC	JOV8	JOV4	JOV2	JOV1
G105			SVF8	SVF7	SVF4	SVFZ	SVFY	SVFX
G106	K7, 0	K6, Z	K5, Z	K4, X	K3, R	K2, F	K1, H	K0, S
	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G107	KEOB,/,#	KH,J,K	KB,K4,D	KP,Q,L	KT,.	KM, –	K9, G	K8, N
	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G108				KSTART	KINPUT	KDELET	KINSRT	KALTER
G109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
G110	KCAN	KYZ	KZX	KXY	KPAGE↑	KPAGE↓	KCUS ↑	KCUS ↓
	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G111	KRESET							
					SHA11	SHA10	SHA09	SHA08
G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G113	KF4TH	KFXW	KFYV	KFXU	KFRC	KFGE	KFN)	KFO(
					SHB11	SHB10	SHB09	SHB08
G114	KFL+	KFT*	KFS=	KFM#	KFF	KFF@	KFJA	KFI,
G115	BFIN1	BFIN2			TFIN	SFIN	EFIN	MFIN
G116	HX/ROV1	AOV64	AOV32	AOV16	–X	+X	SBK	BDT
G117	HY/ROV2	AOV128			–Y	+Y	MLK	*ILK
G118	HZ/DRN				–Z	+Z		
G119	H4				–4	+4		*LDSP
G120	ZRN	*SSTP	SOR	SAR	FIN	ST	MP2	MP1/MINP
G121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
G122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
G123	CON	RTNT	MSPC	RTRCT	GR2	GR1	RGTPN	IGNVRY

	#7	#6	#5	#4	#3	#2	#1	#0
G124	RO8I	RO7I	RO6I	RO5I	RO4I	RO3I	RO2I	RO1I
G125	SIND	SSIN	SGN		R12I	R11I	R10I	RO9I
G126				OVC	HI4	HIZ	HIY	HIX
G127	MIR4	DLK	DNCI			*ABSM	MIRY	MIRX
G128	DMMC				*IT4	*ITZ	*ITY	*ITX
G129	RLSOT	EXLM2	–LMZ	–LMY	–LMX	+LMZ	+LMY	+LMX
G130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G132				BGEN	BGIALM	BGION		IOLACK
G133		SYNCJ					SLHZ1	SLHZ0
G134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135								RGTAP
G136								
G137								
G138	PKESS2	PKESS1	*–EDCZ	*–EDCY	*–EDCX	*+EDCZ	*+EDCY	*+EDCX
G139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G140	F1D	STWD	STRD	TLRSTI	UNIT	TL256	TL128	TLSKP
G141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	*–MIT4	*–MITZ	*–MITY	*–MITX	*+MIT4	*+MITZ	*+MITY	*+MITX
G143	*BECLP	*BEUCP						
G144			EAX8	EAX7	EAX4	EAX3	EAX2	EAX1
G145								
G146					SPPHS	SPSYC	ROV1E	ROV2E
G147	DRNE	RTE	OVCE		*OV8E	*OV4E	*OV2E	*OV1E

CNC → PMC

	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP	SA	STL	SPL	ZP4	ZPZ/EF	ZPY	ZPX
F149	MA		TAP	ENB	DEN	BAL	RST	AL
F150	BF1	BF2	DST		TF	SF	EF	MF
F151	M28	M24	M22	M21	M18	M14	M12	M11
F152	S28	S24	S22	S21	S18	S14/GR3O	S12/GR2O	S11/GR1O
F153	T28	T24	T22	T21	T18	T14	T12	T11
F154	M00	M01	M02	M30	B38	B34	B32	B31
F155	B28	B24	B22	B21	B18	B14	B12	B11
F156	T48	T44	T42	T41	T38	T34	T32	T31
F157			MF3	MF2	M38	M34	M32	M31
F158					MMI4		MMI2	MMI1
F159			BAL6	BAL5	BAL4	BAL3	BAL2	BAL1
F160							ESEND	EREND
F161				FXST	ZP24	ZP2Z	ZP2Y	ZP2X
F162	U07	U06	U05	U04	U03	U02	U01	U00
F163	U15	U14	U13	U12	U11	U10	U09	U08
F164	PRTSF	RWD	CKGRP					
F165	HOBSYN		MSPCF	RTRCTF	GWLF		RGSPM	RGSPP
F166	ZP28	ZP27	ZP8	ZP7	ZP48	ZP47	ZP38	ZP37
F167								
F168	TORQL				ZRF4	ZRFZ	ZRFY	ZRFX
F169	ZP44	ZP4Z	ZP4Y	ZP4X	ZP34	ZP3Z	ZP3Y	ZP3X
F170								
F171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173					R12O	R11O	R10O	R09O
F174	H4O	HZO	HYO	HXO	ZRNO	MD4O	MD2O	MD1O
F175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O

	#7	#6	#5	#4	#3	#2	#1	#0
F176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F177	–4O	+4O	–ZO	+ZO	–YO	+YO	–XO	+XO
F178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F179	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F180	PECK2			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F181								
F182								
F183								
F184	MV4	MVZ	MVY	MVX	INP4	INPZ	INPY	INPX
F185	S28	S24	S22	S21	S18	S14	S12	S11
F186	S48	S44	S42	S41	S38	S34	S32	S31
F187					S58	S54	S52	S51
F188	*EAXSL	CUT	EOV0	SRNMV	BCLP	BUCLP	TLNW	TLCH
F189					SPDS4	SPDS3	SPDS2	SPDS1
F190								
F191								
F192	SYNAL	RTPT				TLCHB	TLCHE	TLCHI
F193	M228	M224	M222	M221	M218	M214	M212	M211
F194	M318	M314	M312	M311	M238	M234	M232	M231
F195	M338	M334	M332	M331	M328	M324	M322	M321
F196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

PMC → CNC

	#7	#6	#5	#4	#3	#2	#1	#0
G200			PMC window DI					
to			to					
G209			PMC window DI					
G210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226								
G227								
G228								
G229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231						NRROA	ROTAA	INDXA
G232								
G233	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB

	#7	#6	#5	#4	#3	#2	#1	#0
G235						NRROB	ROTAB	INDXB
G236								
G237		INFD		SYNC4				
G238								
G239	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G240	Operator's panel DI							
to	to							
G249	Operator's panel DI							

PMC → CNC

	#7	#6	#5	#4	#3	#2	#1	#0
F250	PMC window DO							
to	to							
F269	PMC window DO							
F270	EBSYA	EOTNA	EOTPA	EGENA	EFENA	EIALA	ECKZA	EINPA
F271								EMFA
F272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F273	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F274								EMFB
F275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276								
F277								
F278								
F279								
F280								
F281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282					PCFNA	PCHPA	CFINA	CHPA
F283								
F284								
F285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286					PCFNB	PCHPB	CFINB	CHOB
F287								
F288								
F289								
F290	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0
F291	Operator's panel DO							
to	to							
F299	Operator's panel DO							

List of addresses (T series)

MT → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0
X008	SKIP	ESKIP	–MIT2	+MIT2	–MIT1	+MIT1	ZAE	XAE
	SKIP	ESKIP		SKIP4	SKIP3	SKIP2	ZAE	XAE

* 0–GCC/GCD for lower stage

Address	#7	#6	#5	#4	#3	#2	#1	#0
X016	*DEC3		*DECX		*DEC5S	*DEC7		

* By parameter setting

Address	#7	#6	#5	#4	#3	#2	#1	#0
X017	*DEC4		*DECZ		*DEC6S	*DEC8		

* By parameter setting

Address	#7	#6	#5	#4	#3	#2	#1	#0
X018	*ESPS		*+LZ					
X019	*DEC3		*DEC4					

* By parameter setting

Address	#7	#6	#5	#4	#3	#2	#1	#0
X020								
X021			*ESP					

PMC → CNC

	#7	#6	#5	#4	#3	#2	#1	#0
G100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G103	AFL	PRC	SPC	SPB	SPA	SPD		SRN
G104		RRW	*FLWU	ESRSYC				
G105	PLCRVON	KILPLUS	SVF8	SVF7	SVF4	SVF3	SVFZ	SVFX
G106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
	M2R08I	M2R07I	M2R06I	M2R05I	M2R04I	M2R03I	M2R02I	M2R01I
G107	KEOB	KH, I, K	KA, C	KP, Q	KT,.	KM, –	K9, G	K8, N
	M2SIND		M2SGN		M2R12I	M2R11I	M2R10I	M2R09I
G108				KSTART	KINPUT	KDELET	KINSRT	KALTER
	M3R08I	M3R07I	M3R06I	M3R05I	M3R04I	M3R03I	M3R02I	M3R01I
G109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
	M3SIND	M3SSIN	M3SGN		M3R12I	M3R11I	M3R10I	M3R09I
G110	KCAN				KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G111	KRESET							
					SHA11	SHA10	SHA09	SHA08
G112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G113	KFWH	KFUV	KFZJ	KFXY	KFRC	KFGB	KFN)	KFO(
					SHB11	SHB10	SHB09	SHB08
G114	KFL+	KFT*	KFS=	KFM#	KFF	KFFE	KFK@	KFI,
G115	BFIN				TFIN	SFIN		MFIN
G116	HX/ROV1	ROV3D	ROV2D	ROV1D	–X	+X	SBK	BDT
G117	HZ/ROV2				–Z	+Z	MLK	MP1/MINP
G118	DRN				GR2	GR1		MP2
	H3				–3	+3		MP2

* Lower stage by parameter setting

	#7	#6	#5	#4	#3	#2	#1	#0
G119	H4				–4	+4		*LDSP
G120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
G121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1

	#7	#6	#5	#4	#3	#2	#1	#0
G122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
G123		SPSTP	*SCPF	*SUCPF			RGTPN	COFF
	DRN	SPSTP	*SCPF	*SUCPF	GR2	GR1	RGTPN	COFF
	* Lower setting by parameter setting							
G124	RO8I	RO7I	RO6I	RO5I	RO4I	RO3I	RO2I	RO1I
G125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G126	CDZ	SMZ	*OV16	OVC	HI4	HI3	HIZ	HIX
G127		DLK	DNCI			*ABSM	MIZ	IGNVRY
G128	DMMC				IT4	IT3	ITZ	ITX
G129		EXLM2						
G130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G132	GOQSM	WOQSM	TNFS	OFN4	OFN3	OFN2	OFN1	OFN0
G133	WOSET	NOZAGC						
G134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135								
G136								
G137								
G138	PKESS2	PKESS1		*–EDCZ	*–EDCX		*+EDCZ	*+EDCX
G139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G140	AOVR128	AOVR64	AOVR32	AOVR16	UNIT			TLSKP
G141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	KXZ			BGEN	BGIALM	BGION		IOLACK
G143								
G144			EAX8	EAX7	EAX4	EAX3	EAX2	EAX1
G145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G146	PC2SLC	SBRT			SPPHS	SPSYC	ROV2E	ROV1E
G147	DRNE	RTE	OVCE	*OV16E	*OV8E	*OV4E	*OV2E	*OV1E

CNC → PMC

	#7	#6	#5	#4	#3	#2	#1	#0
F148	OP	SA	STL	SPL	ZP4	ZP3	ZPZ	ZPX
F149	MA	DEN2	TAP	ENB	DEN	BAL	RST	AL
F150	BF		DST		TF	SF		MF
F151	M28	M24	M22	M21	M18	M14	M12	M11
F152	S28	S24	S22	S21	S18	S14	S12	S11
F153	T28	T24	T22	T21	T18	T14	T12	T11
F154								SPAL
F155							MMI2	MMI1
F156			BAL6	BAL5	BAL4	BAL3	BAL2	BAL1
F157			MF3	MF2	M38	M34	M32	M31
F158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F159	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
F160	PSYN						ESEND	EREND
F161					ZP24	ZP23	ZP2Z	ZP2X
F162	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
F163	UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
F164	PRTSF	RWD	CKGRP		ENB3	ENB2	SUCLP	SCLP
F165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F166	ZP28	ZP27	ZP8	ZP7	ZP48	ZP47	ZP38	ZP37
F167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F168					ZRF4	ZRF3	ZRFZ	ZRFX
F169	ZP44	ZP43	ZP4Z	ZP4X	ZP34	ZP33	ZP3Z	ZP3X
F170			TORQ8	TORQ7	TORQ4	TORQ3	TORQ2	TORQ1
F171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173					R12O	R11O	R10O	R09O
F174	H4O	H3O	HZO	HXO	ZRNO	MD4O	MD2O	MD1O
F175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O

	#7	#6	#5	#4	#3	#2	#1	#0
F176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F177	–4O	+4O	–3O	+3O	–ZO	+ZO	–XO	+XO
F178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F179	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F180				BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F181								
F182								
F183								
F184	MV4	MV3	MVZ	MVX	INP4	INP3	INPZ	INPX
F185	S28	S24	S22	S21	S18	S14	S12	S11
F186	S48	S44	S42	S41	S38	S34	S32	S31
F187					S58	S54	S52	S51
F188	*EAXSL	CUT	EOV0	SRNMV	THRD		TLNW	TLCH
F189								
F190								
F191								
F192								
F193	M228	M224	M222	M221	M218	M214	M212	M211
F194	M318	M314	M312	M311	M238	M234	M232	M231
F195	M338	M334	M332	M331	M328	M324	M322	M321
F196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

PMC → CMC

	#7	#6	#5	#4	#3	#2	#1	#0
G200			PMC window DI					
to				to				
G209			PMC window DI					
G210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226								
G227								
G228								
G229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231								
G232								
G233	MRDYB	ORCMB	SFRB	ERVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	PCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB

	#7	#6	#5	#4	#3	#2	#1	#0
G235								
G236								
G237								
G238								
G239	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G240	Operator's panel DI							
to	to							
G249	Operator's panel DI							

CNC → PMC

	#7	#6	#5	#4	#3	#2	#1	#0
F250	PMC window DO							
to	to							
F269	PMC window DO							
F270	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F271								EMFA
F272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F273	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECZKB	EINPB
F274								EMFB
F275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276	B7	B6	B5	B4	B3	B2	B1	B0
F277	B15	B14	B13	B12	B11	B10	B9	B8
F278	B23	B22	B21	B20	B19	B18	B17	B16
F279	B31	B30	B29	B28	B27	B26	B25	B24
F280								
F281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282					PCFNA	PCHPA	CFINA	CHPA
F283								
F284								
F285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286					PCFNB	PCHPB	CFINB	CHPB
F287								
F288								
F289	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0
F290	Operator’s panel DO							
to	to							
F299	Operator’s panel DO							

List of addresses (0–TTC)

- **Interface between CNC and PMC or MT**

Diagnostic number and PMC address of 0–TTC machine interface signal

Diagnostic No.	PMC address	Direction of signal	Selection of tool post
000 to 040	X000 to X040	MT → PMC	Tool post 1 and tool post 2
048 to 086	Y048 to Y086	PMC → MT	Tool post 1 and tool post 2
100 to 147	G100 to G147	PMC → CNC	Tool post 1
148 to 199	F148 to F199	CNC → PMC	Tool post 1
200 to 249	G200 to G249	PMC → CNC	Tool post 1
250 to 299	F250 to F299	CNC → PMC	Tool post 1
100 to 147	G1300 to G1347	PMC → CNC	Tool post 2
148 to 187	F1348 to F1387	CNC → PMC	Tool post 2
200 to 209	G1400 to G1409	PMC → CNC	Tool post 2
250 to 279	F1450 to F1479	CNC → PMC	Tool post 2

Machine interface signal (MT → CNC) (for tool post 1 and 2)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18–36	M18–21	M18–5	M18–35	M18–20	M18–34	M18–19	M18–33
X002	002								
		M18–24	M18–8	M18–38	M18–23	M18–7	M18–37	M18–22	M18–6
X004	004								
		M18–11	M18–41	M18–26	M18–10	M18–40	M18–25	M18–9	M18–39
X006	006								
		M18–45	M18–14	M18–44	M18–13	M18–43	M18–12	M18–42	M18–27
X008	008	SKIPM		–MIT2M	+MIT2M	–MIT1M	+MIT1M	ZAEM	XAEM
		M18–49	M18–18	M18–48	M18–17	M18–47	M18–16	M18–46	M18–15
X010	010					/	/	/	/
		M20–11	M20–41	M20–26	M20–10				
X012	012								
		M20–45	M20–14	M20–44	M20–13	M20–43	M20–12	M20–42	M20–27
X014	014								
		M20–49	M20–18	M20–48	M20–17	M20–47	M20–16	M20–46	M20–15
X016	016	*DEC3	/	*DECXM	/	*DECXS			
		M1–6		M1–38		M1–20	M1–21	M1–11	M1–12
		* By parameter setting							
X017	017	*DEC4	/	*DECZM	/	*DECZS			
		M1–7		M1–39		M1–22	M1–23	M1–9	M1–10
		* By parameter setting							
X018	018	*ESPS	/	*+LZM	/	TRT2		/	/
		M1–8		M1–40		M1–24	M1–25		
X019	019	*DEC3	/	*DEC4	/			/	/
		M20–40		M20–25		M20–9	M20–39		
X020	020								
		M1–13	M1–37	M1–5	M1–14	M1–15	M1–16	M1–17	M1–18
X021	021				*ESPM				
		M1–41	M1–26	M1–27	M1–19	M1–33	M1–34	M1–35	M1–36
X022	022								
		M1–42	M1–43	M1–44	M1–45	M1–46	M1–47	M1–48	M1–49
X040	040	SKIPS	*+LZS	–MIT2S	+MIT2S	–MIT1S	+MIT1S	ZAES	XAES
		M28–11	M28–17	M28–10	M28–16	M28–9	M28–15	M28–8	M28–14

NOTE

An alphabet at the tail of a signal name indicates ad follows:

☐ . . ☐ M ; Signal for tool post 1

☐ . . ☐ S ; Signal for tool post 2

When the signals are descriminated per page between tool post 1 and tool post 2, the alphabet at the tail is omitted.

Machine interface signal (PMC → MT) (for tool post 1 and 2)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048					/			
		M2–5	M2–6	M2–7	M2–8		M2–27	M2–26	M2–25
Y049	049		/	/			/		
		M2–9			M2–41	M2–22		M2–23	M2–24
Y050	050	/	/		/			/	
				M2–10		M2–20	M2–19		M2–21
Y051	051								
		M2–33	M2–34	M2–35	M2–36	M2–37	M2–38	M2–39	M2–40
Y052	052								
		M2–11	M2–12	M2–13	M2–14	M2–15	M2–16	M2–17	M2–18
Y053	053								
		M2–42	M2–43	M2–44	M2–45	M2–46	M2–47	M2–48	M2–49
Y080	080								
		M19–8	M19–7	M19–6	M19–5	M19–4	M19–3	M19–2	M19–1
Y082	082								
		M19–16	M19–15	M19–14	M19–13	M19–12	M19–11	M19–10	M19–9
Y084	084								
		M20–36	M20–21	M20–5	M20–35	M20–20	M20–34	M20–19	M20–33
Y086	086								
		M20–24	M20–8	M20–38	M20–23	M20–7	M20–37	M20–22	M20–6

Machine interface signal (PMC → CNC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
G100	100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G101	101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G102	102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G103	103	AFL	PRC	SPC	SPB	SPA	SPD		SRN
G104	104		RRW	*FLWU	ESRSYC				
G105	105	PLCRVON	KILPLUS		SVF7	SVF4	SVF3	SVFZ	SVFX
G106	106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
		M2R08I	M2R07I	M2R06I	M2R05I	M2R04I	M2R03I	M2R02I	M2R01I
G107	107	KEOB	KH, I, K	KA, C	KP, Q	KT,.	KM, –	K9, G	K8, N
		M2SIND		M2SGN		M2R12I	M2R11I	M2R10I	M2R09I
G108	108				KSTART	KINPUT	KDELET	KINSRT	KALTER
		M3R08I	M3R07I	M3R06I	M3R05I	M3R04I	M3R03I	M3R02I	M3R01I
G109	109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
		M3SIND	M3SSIN	M3SGN		M3R12I	M3R11I	M3R10I	M3R09I
G110	110	KCAN				KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
		SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G111	111	KRESET							
						SHA11	SHA10	SHA09	SHA08
G112	112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
		SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G113	113	KFWH	KFUV	KFZJ	KFXY	KFRC	KFGB	KFN)	KFO(
						SHB11	SHB10	SHB09	SHB08
G114	114	KFL+	*KFT	KFS=	KFM#	KFF	KFE	KFK@	KFI,
G115	115	BFIN				TFIN	SFIN		MFIN
G116	116	HX/ROV1	ROV3D	ROV2D	ROV1D	–X	+X	SBK	BDT
G117	117	HZ/ROV2				–Z	+Z	MLK	MP1/MINP
G118	118	DRN				GR2	GR1		MP2
		H3				–3	+3		MP2

Lower low is selected when bit 5 of parameter No. 0031 is 1.

		#7	#6	#5	#4	#3	#2	#1	#0
G119	119	H4				–4	+4		*LDSP
G120	120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
G121	121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1

		#7	#6	#5	#4	#3	#2	#1	#0
G122	122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
G123	123		SPSTP	*SCPF	*SUCPF			RGTPN	COFF
		DRN	SPSTP	*SCPF	*SUCPF	GR2	GR1	RGTPN	COFF

Lower low is selected when bit 5 of parameter No. 0031 is 1.

		#7	#6	#5	#4	#3	#2	#1	#0
G124	124	R081	R071	R061	R051	R041	R031	R021	R011
G125	125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G126	126	CDZ	SMZ	*OV16	OVC	HI4	HI3	HIZ	HIX
G127	127		DLK	DNCI			*ABSM	MIZ	IGNVRY
G128	128					IT4	IT3	ITZ	ITX
G129	129		EXLM2						
G130	130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G131	131	UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8
G132	132	GOQSM	GOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G133	133	WOSET				SLSPB	SLSPA	NOWT	TRT2PC
G134	134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G135	135								

For order made macro

		#7	#6	#5	#4	#3	#2	#1	#0
G136	136								

For order made macro

		#7	#6	#5	#4	#3	#2	#1	#0
G137	137								

For order made macro

		#7	#6	#5	#4	#3	#2	#1	#0
G138	138	PKESS2	PKESS1		*–EDCZ	*–EDCX		*+EDCZ	*+EDCX
G139	139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G140	140	AOVR128	AOVR64	AOVR32	AOVR16	UINT			TLSKP
G141	141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G142	142	KXZ			BGEN	BGIALM	BGION		IOLACT
G143	143								
G144	144				EAX7	EAX4	EAX3	EAX2	EAX1
G145	145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G146	146	PC2SLC	SBRT			SPPHS	SPSYC	ROV2E	ROV1E
G147	147	DRNE	RTE	OVCE	*OV16E	*OV8E	*OV4E	*OV2E	*OV1E

Machine interface signal (CNC → PMC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
F148	148	OP	SA	STL	SPL	ZP4	ZP3	ZPZ	ZPX
F149	149	MA	DEN2	TAP	ENB	DEN	BAL	RST	AL
F150	150	BF		DST		TF	SF		MF
F151	151	M28	M24	M22	M21	M18	M14	M12	M11
F152	152	S28	S24	S22	S21	S18	S14	S12	S11
F153	153	T28	T24	T22	T21	T18	T14	T12	T11
F154	154								SPAL
F155	155							MMI2	MMI1
F156	156					BAL4	BAL3	BAL2	BAL1
F157	157			MF3	MF2	M38	M34	M32	M31
F158	158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F159	159	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
F160	160	PSYN	WATO					ESEND	EREND
F161	161					ZP24	ZP23	ZP2Z	ZP2X
F162	162	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
F163	163	UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
F164	164	PRTSF	RWD	CKGRP		ENB3	ENB2	SUCLP	SCLP
F165	165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F166	166		ZP27		ZP7		ZP47		ZP37
F167	167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F168	168					ZRF4	ZRF3	ZRFZ	ZRFX
F169	169	ZP44	ZP43	ZP4Z	ZP4X	ZP34	ZP33	ZP3Z	ZP3X
F170	170				TORQ7	TORQ4	TORQ3	TORQZ	TORQX
F171	171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F172	172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F173	173					R12O	R11O	R10O	R09O
F174	174	H4O	H3O	HZO	HXO	ZRNO	MD4O	MD2O	MD1O
F175	175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O

		#7	#6	#5	#4	#3	#2	#1	#0
F176	176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F177	177	–4O	+4O	–3O	+3O	–ZO	+ZO	–XO	+XO
F178	178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F179	179	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F180	180	TAL	BOFF	COSP	BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F181	181								
F182	182								
									For order made macro
F183	183								
F184	184	MV4	MV3	MVZ	MVX	INP4	INP3	INPZ	INPX
F185	185	S28	S24	S22	S21	S18	S14	S12	S11
F186	186	S48	S44	S42	S41	S38	S34	S32	S31
F187	187					S58	S54	S52	S51
F188	188	*EAXSL	CUT	EOV0	SRNMV	THRD		TLNW	TLCH
F189	189	RSMAX			SYN7OM	SYN4OM	SYN3OM	SYN2OM	SYN1OM
F190	190								
F191	191								
F192	192								
F193	193	M228	M224	M222	M221	M218	M214	M212	M211
F194	194	M318	M314	M312	M311	M238	M234	M232	M231
F195	195	M338	M334	M332	M331	M328	M324	M322	M321
F196	196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F197	197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F198	198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F199	199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Machine interface signal (PMC → CNC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
G200	200				PMC window DI				
to					to				
G209	209				PMC window DI				
G210	210	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA			EFINA
G211	211	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G212	212	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G213	213	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G214	214	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G215	215	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G216	216	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G217	217	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G218	218	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB			EFINB
G219	219	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G220	220	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G221	221	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G222	222	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G223	223	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G224	224	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G225	225	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G226	226								
G227	227								
G228	228								
G229	229	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G230	230	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G231	231								
G232	232								
G233	233	MRDYB	ORCMB	SFRB	ERVB	CTH1B	CTH2B	TLMHB	TLMLB
G234	234	PCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB

		#7	#6	#5	#4	#3	#2	#1	#0
G235	235								
G236	236								
G237	237				SYN7M	SYN4M	SYN3M	SYN2M	SYN1M
G238	238				PK7M	PK4M	PK3M	PK2M	PK1M
G239	239								
G240	240				Operator's panel DI				
to					to				
G249	249				Operator's panel DI				

Machine interface signal (CNC → PMC) (for tool post 1)

		#7	#6	#5	#4	#3	#2	#1	#0
F250	250				PMC window DO				
to					to				
F269	269				PMC window DO				
F270	270	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECXZA	EINPA
F271	271								EMFA
F272	272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F273	273	EBSYB	EOTNB	EOTPB	EGEND	EDENB	EIALB	ECKZB	EINPB
F274	274								EMFB
F275	275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F276	276	B7	B6	B5	B4	B3	B2	B1	B0
F277	277	B15	B14	B13	B12	B11	B10	B9	B8
F278	278	B23	B22	B21	B20	B19	B18	B17	B16
F279	279	B31	B30	B29	B28	B27	B26	B25	B24
F280	280								
F281	281	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F282	282					PCFNA	PCHPA	CFINA	CHPA
F283	283								
F284	284								
F285	285	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F286	286					PCFNB	PCHPB	CFINB	CHPB
F287	287								
F288	288								
F289	289								
F290	290				Operator’s panel DO				
to					to				
F299	299				Operator’s panel DO				

Machine interface signal (PMC → CNC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
G1300	100	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G1301	101	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G1302	102	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G1303	103	AFL	PRC	SPC	SPB	SPA	SPD		SPN
G1304	104		RRW	*FLWU	ESRSYC				
G1305	105	PLCRVON	KILPLUS			SVF4	SVF3	SVFZ	SVFX
G1306	106	K7, 0	K6, F	K5, Z	K4, X	K3, R	K2, W	K1, U	K0, S
		M2R08I	M2R07I	M2R06I	M2R05I	M2R04I	M2R03I	M2R02I	M2R01I
G1307	107	KEOB	KH, I, K	KA, C	KP, Q	KT,.	KM, –	K9, G	K8, N
		M2SIND		M2SGN		M2R12I	M2R11I	M2R10I	M2R09I
G1308	108				KSTART	KINPUT	KDELET	KINSRT	KALTER
		M3R08I	M3R07I	M3R06I	M3R05I	M3R04I	M3R03I	M3R02I	M3R01I
G1309	109			KALARM	KDGNOS	KPARAM	KOFSET	KPROGRM	KPOS
		M3SIND	M3SSIN	M3SGN		M3R12I	M3R11I	M3R10I	M3R09I
G1310	110	KCAN				KPAGE↑	KPAGE↓	KCUS↑	KCUS↓
		SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G1311	111	KRESET							
						SHA11	SHA10	SHA09	SHA08
G1312	112	KSL	KS1	KS2	KS3	KS4	KS5	KSR	
		SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G1313	113	KFWH	KFUV	KFZJ	KFXY	KFRC	KFGB	KFN)	KFO(
						SHB11	SHB10	SHB09	SHB08
G1314	114	KFL+	KFT*	KFS=	KFM#	KFF	KFE	KFK@	KFI,
G1315	115	BFIN				TFIN	SFIN		MFIN
G1316	116	HX/ROV1	ROV3D	ROV2D	ROV1D	–X	+X	SBK	BDT
G1317	117	HZ/ROV2				–Z	+Z	MLK	MP1/MINP
G1318	118	DRN				GR2	GR1		MP2
G1319	119								
G1320	120	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
G1321	121	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
G1322	122	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1

		#7	#6	#5	#4	#3	#2	#1	#0
G1323	123		SPSTP	*SCPF	*SUCPF			RGTPN	COFF
G1324	124	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G1325	125	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G1326	126	CDZ	SMZ	*OV16	OVC			HIZ	HIX
G1327	127		DLK	DNCI			*ABSM	MIZ	IGNVRY
G1328	128							ITZ	ITX
G1329	129		EXLM2						
G1330	130	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
G1331	131	UI15	UI14	UI13	UI12	UI11	UI10	UI19	UI18
G1332	132	GOQSM	GOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G1333	133	WOSET				SLPCB	SLPCA	NOWT	
G1334	134			MFIN3	MFIN2	EXWT	EXSTP	EXRD	ENBKY
G1335	135								
G1336	136								
G1337	137								
G1338	138	PKESS2	PKESS1		*–EDCZ	*–EDCX		*+EDCZ	*+EDCX
G1339	139	TLRST	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G1340	140	AOVR128	AOVR64	AOVR32	AOVR16	UINT			TLSKP
G1341	141	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G1342	142								
G1343	143								
G1344	144					EAX4	EAX3	EAX2	EAX1
G1345	145	GR31	GR21	*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G1346	146	PC2SLC	SBRT			SPPHS	SPSYC	ROV2E	ROV1E
G1347	147	DRNE	RTE	OVCE	*OV16E	*OV8E	*OV4E	*OV2E	*OV1E

Machine interface signal (CNC → PMC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
F1348	148	OP	SA	STL	SPL	ZP4	ZP3	ZPZ	ZPX
F1349	149	MA	DEN2	TAP	ENB	DEN	BAL	RST	AL
F1350	150	BF		DST		TF	SF		MF
F1351	151	M28	M24	M22	M21	M18	M14	M12	M11
F1352	152	S28	S24	S22	S21	S18	S14	S12	S11
F1353	153	T28	T24	T22	T21	T18	T14	T12	T11
F1354	154								SPAL
F1355	155							MMI2	MMI1
F1356	156					BAL4	BAL3	BAL2	BAL1
F1357	157			MF3	MF2	M38	M34	M32	M31
F1358	158	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F1359	159	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8
F1360	160	PSYN	WATO					ESEND	EREND
F1361	161					ZP24	ZP23	ZP2Z	ZP2X
F1362	162	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
F1363	163	UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8
F1364	164	PRTSF	RWD	CKGRD		ENB3	ENB2	SUCLP	SCLP
F1365	165	PX7	PX6	PX5	PX4	PX3	PX2	PX1	PX0
F1366	166								
F1367	167	PZ7	PZ6	PZ5	PZ4	PZ3	PZ2	PZ1	PZ0
F1368	168					ZRF4	ZRF3	ZRFZ	ZRFX
F1369	169	ZP44	ZP43	ZP4Z	ZP4X	ZP34	ZP33	ZP3Z	ZP3X
F1370	170				TORQ7	TORQ4	TORQ3	TORQZ	TORQX
F1371	171	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F1372	172	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F1373	173					R12O	R11O	R10O	R09O
F1374	174			HZO	HXO	ZRNO	MD4O	MD2O	MD1O
F1375	175	MP2O	MP1O	ROV2O	ROV1O	JV8O	JV4O	JV2O	JV1O

		#7	#6	#5	#4	#3	#2	#1	#0
F1376	176	DRNO	MLKO	SBKO	BDTO	OV8O	OV4O	OV2O	OV1O
F1377	177					–ZO	+ZO	–XO	+XO
F1378	178	SPO	RTO	KEYO	SYCAL	FSPPH	FSPSY	FSCSL	
F1379	179	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F1380	180	TAL	BOFF	COSP	BGEACT	RPALM	RPBSY	RPGDPL	INHKY
F1381	181								
F1382	182								
F1383	183								
F1384	184	MV4	MV3	MVZ	MVX	INP4	INP3	INPZ	INPX
F1385	185	S28	S24	S22	S21	S18	S14	S12	S11
F1386	186	S48	S44	S42	S41	S38	S34	S32	S31
F1387	187					S58	S54	S52	S51
F1388	188	*EAXSL	CUT	EOV0	SRNMV	THRD		TLNW	TLCH
F1389	189	RSMAX			SYN7OM	SYN4OM	SYN3OM	SYN2OM	SYN1OM
F1390	190								
F1391	191								
F1392	192								
F1393	193	M228	M224	M222	M221	M218	M214	M212	M211
F1394	194	M318	M314	M312	M311	M238	M234	M232	M231
F1395	195	M338	M334	M332	M331	M328	M324	M322	M321
F1396	196	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F1397	197	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F1398	198	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F1399	199	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Machine interface signal (PMC → CNC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
G1400	200				PMC window DI				
to					to				
G1409	209				PMC window DI				

Machine interface signal (CNC → PMC) (for tool post 2)

		#7	#6	#5	#4	#3	#2	#1	#0
F1450	250				PMC window DO				
to					to				
F1469	269				PMC window DO				
F1470	270	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECXZA	EINPA
F1471	271								EMFA
F1472	272	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F1473	273	EBSYB	EOTNB	EOTPB	EGEND	EGENB	EIALB	ECKZB	EINPB
F1474	274								EMFB
F1475	275	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F1476	276	B7	B6	B5	B4	B3	B2	B1	B0
F1477	277	B15	B14	B13	B12	B11	B10	B9	B8
F1478	278	B23	B22	B21	B20	B19	B18	B17	B16
F1479	279	B31	B30	B29	B28	B27	B26	B25	B24

List of addresses (without PMC)

(1) M series

(a) MT → CNC (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18–36	M18–21	M18–5	M18–35	M18–20	M18–34	M18–19	M18–33
X002	002								
		M18–24	M18–8	M18–38	M18–23	M18–7	M18–37	M18–22	M18–6
X004	004	4NG							
		M18–11	M18–41	M18–26	M18–10	M18–40	M18–25	M18–9	M18–39
X006	006								
		M18–45	M18–14	M18–44	M18–13	M18–43	M18–12	M18–42	M18–27
X008	008	SKIP		*RILK			ZAE	YAE	XAE
		M18–49	M18–18	M18–48	M18–17	M18–47	M18–16	M18–46	M18–15
X010	010					/	/	/	/
		M20–11	M20–41	M20–26	M20–10				
X012	012								
		M20–45	M20–14	M20–44	M20–13	M20–43	M20–12	M20–42	M20–27
X014	014								
		M20–49	M20–18	M20–48	M20–17	M20–47	M20–16	M20–46	M20–15
X016	016	HX/ROV1	/	*DECX	/	–X	+X	SBK	BDT
		M1–6		M1–38		M1–20	M1–21	M1–11	M1–12
X017	017	HY/ROV2	/	*DECY	/	–Y	+Y	MLK	*ILK
		M1–7		M1–39		M1–22	M1–23	M1–9	M1–10
X018	018	HZ/DRN	/	*DECZ	/	–Z	+Z	/	/
		M1–8		M1–40		M1–24	M1–25		
X019	019	H4	/	*DEC4	/	–4	+4	/	/
		M20–40		M20–25		M20–9	M20–39		
X020	020	ZRN	*SSTP	SOR	SAR	FIN	ST	MP2	MP1/MINP
		M1–13	M1–37	M1–5	M1–14	M1–15	M1–16	M1–17	M1–18
X021	021	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
		M1–41	M1–26	M1–27	M1–19	M1–33	M1–34	M1–35	M1–36
X022	022	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
		M1–42	M1–43	M1–44	M1–45	M1–46	M1–47	M1–48	M1–49

(b) CNC → MT (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048	OP	SA	STL	SPL	/	ZPZ/EF	ZPY	ZPX
		M2–5	M2–6	M2–7	M2–8		M2–27	M2–26	M2–25
Y049	049	MA	/	/	ENB	DEN	/	RST	AL
		M2–9			M2–41	M2–22		M2–23	M2–24
Y050	050	/	/	DST	/	TF	SF	/	MF
				M2–10		M2–20	M2–19		M2–21
Y051	051	M28	M24	M22	M21	M18	M14	M12	M11
		M2–33	M2–34	M2–35	M2–36	M2–37	M2–38	M2–39	M2–40
Y052	052	S28	S24	S22	S21	S18	S14/GR30	S12/GR20	S11/GR10
		M2–11	M2–12	M2–13	M2–14	M2–15	M2–16	M2–17	M2–18
Y053	053	T28	T24	T22	T21	T18	T14	T12	T11
		M2–42	M2–43	M2–44	M2–45	M2–46	M2–47	M2–48	M2–49
Y080	080								
		M19–8	M19–7	M19–6	M19–5	M19–4	M19–3	M19–2	M19–1
Y082	082								
		M19–16	M19–15	M19–14	M19–13	M19–12	M19–11	M19–10	M19–9
Y084	084					ZP4			
		M20–36	M20–21	M20–5	M20–35	M20–20	M20–34	M20–19	M20–33
Y086	086								
		M20–24	M20–8	M20–38	M20–23	M20–7	M20–37	M20–22	M20–6

(2) Tseries

(a) MT → CNC (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
X000	000								
		M18–36	M18–21	M18–5	M18–35	M18–20	M18–34	M18–19	M18–33
X002	002								
		M18–24	M18–8	M18–38	M18–23	M18–7	M18–37	M18–22	M18–6
X004	004								
		M18–11	M18–41	M18–26	M18–10	M18–40	M18–25	M18–9	M18–39
X006	006								
		M18–45	M18–14	M18–44	M18–13	M18–43	M18–12	M18–42	M18–27
		SKIP						ZAE	XAE
X008	008	SKIP			SKIP4	SKIP3	SKIP2	ZAE	XAE
		M18–49	M18–18	M18–48	M18–17	M18–47	M18–16	M18–46	M18–15
						0–GCC/GCD for lower stage			
X010	010					/	/	/	/
		M20–11	M20–41	M20–26	M20–10				
X012	012								
		M20–45	M20–14	M20–44	M20–13	M20–43	M20–12	M20–42	M20–27
X014	014								
		M20–49	M20–18	M20–48	M20–17	M20–47	M20–16	M20–46	M20–15
X016	016	HX/ROV1	/	*DECX	/	–X	+X	SBK	BDT
		M1–6		M1–38		M1–20	M1–21	M1–11	M1–12
X017	017	HY/ROV2	/	*DECY	/	–Z	+Z	MLK	MP1/MINP
		M1–7		M1–39		M1–22	M1–23	M1–9	M1–10
X018	018	DRN	/	*+LZ	/	GR2	GR1	/	/
		M1–8		M1–40		M1–24	M1–25		
X019	019	*DEC3	/	*DEC4	/			/	/
		M20–40		M20–25		M20–9	M20–39		
X020	020	ZRN	*SSTP	SOR	SAR	FIN	ST	STLK	MIX
		M1–13	M1–37	M1–5	M1–14	M1–15	M1–16	M1–17	M1–18
X021	021	ERS	RT	*SP	*ESP	*OV8	*OV4	*OV2	*OV1
		M1–41	M1–26	M1–27	M1–19	M1–33	M1–34	M1–35	M1–36
X022	022	PN8	PN4	PN2	PN1	KEY	MD4	MD2	MD1
		M1–42	M1–43	M1–44	M1–45	M1–46	M1–47	M1–48	M1–49

(b) CNC → MT (without PMC)

		#7	#6	#5	#4	#3	#2	#1	#0
Y048	048	OP	SA	STL	SPL	/		ZPT	ZPX
		M2–5	M2–6	M2–7	M2–8		M2–27	M2–26	M2–25
Y049	049	MA	/	/	ENB	DEN	/	RST	AL
		M2–9			M2–41	M2–22		M2–23	M2–24
Y050	050	/	/	DST	/	TF	SF	/	MF
				M2–10		M2–20	M2–19		M2–21
Y051	051	M28	M24	M22	M21	M18	M14	M12	M11
		M2–33	M2–34	M2–35	M2–36	M2–37	M2–38	M2–39	M2–40
Y052	052	S28	S24	S22	S21	S18	S14	S12	S11
		M2–11	M2–12	M2–13	M2–14	M2–15	M2–16	M2–17	M2–18
Y053	053	T28	T24	T22	T21	T18	T14	T12	T11
		M2–42	M2–43	M2–44	M2–45	M2–46	M2–47	M2–48	M2–49
Y080	080								
		M19–8	M19–7	M19–6	M19–5	M19–4	M19–3	M19–2	M19–1
Y082	082								
		M19–16	M19–15	M19–14	M19–13	M19–12	M19–11	M19–10	M19–9
Y084	084								
		M20–36	M20–21	M20–5	M20–35	M20–20	M20–34	M20–19	M20–33
Y086	086								
		M20–24	M20–8	M20–38	M20–23	M20–7	M20–37	M20–22	M20–6

A.2 LIST OF SIGNALS

A.2.1 List of Signals (In Order of Functions)

Meaning of symbols in the below table
○: Available
●: 0–TTC only
☆: Available for 0–GCC or 0–GSC only
Blank : Unavailable

Function	Signal name	Symbol	T	M	Address	Ref. item
Axis moving state output	Axis moving signal	MVX, MVY, MVZ, MV4		○	F188.4 to F188.7	1.2.4
		MVX, MVZ, MV3, MV4	○			
Mirror image	Mirror image check signal	MMI1, MMI2	○		F158.0, F158.1	1.2.5
	Mirror image check signal	MMI1, MMI2, MMI4		○	F158.0, F158.1, F158.3,	
	Mirror image signal	MIX, MIZ	○		G120.0, G127.1	
	Mirror image signal	MIRX, MIRY, MIR4		○	G127.0, G127.1, G127.7	
Follow–up	Follow–up signal	*FLWU	○	○	G104.5	1.2.6
Servo off (mechanical handle)	Servo off signal	SVFX, SVFZ, SVF3, SVF4	○		G105.0, G105.1, G105.2, G105.3	1.2.7
		SVFX, SVFY, SVFZ, SVF4		○		
Position switch	Position switch signal	PSW01 to PSW10	○	○	F190, F191.0, F191.1	1.2.8
Ignore–the–fourth–axis–signal	Ignore–the–fourth–axis–signal	4NG		○	X004.7	1.4.3
Cancel–the–Z–axis command signal	Cancel–the–Z–axis command signal	ZNG		○	G103.6	1.4.4
Simple synchronous control	Servo axis synchronization alarm signal	SYNAL		○	F192.7	1.6
	Signals to select the slave axis for simple synchronous control	SYNCX, SYNCZ, SYNC3, SYNC4	○		G237.0 to G237.3	
		SYNC4		○	G237.3	
	Signal for selecting the manual feed axis for simple synchronous control	SYNCJ		○	G133.6	
Axis recomposition	Synchronization control start signals (tool post 1)	PK1M, PK2M, PK3M, PK4M, PK7M	●		G238.0, G238.1, G238.2, G238.3, G238.4	1.7
	Synchronization control start signals (tool post 2)	PK1S, PK2S, PK3S, PK4S	●		G1438.0, G1438.1, G1438.2, G1438.3	
	Synchronization control start signals (tool post 1)	SYN1M, SYN2M, SYN3M, SYN4M, SYN7M	●		G237.0, G237.1, G237.2, G237.3, G237.4	
	Synchronization control start signals (tool post 2)	SYN1S, SYN2S, SYN3S, SYN4S	●		G1437.0, G1437.1, G1437.2, G1437.3	

Function	Signal name	Symbol	T	M	Address	Ref. item
Axis recomposition	Axis recomposition signals (tool post 1)	SYN1OM, SYN2OM, SYN3OM, SYN4OM, SYN7OM	●		F189.0, F189.1, F189.2, F189.3, F189.4	1.7
	Axis recomposition signals (tool post 2)	SYN1OS, SYN2OS, SYN3OS, SYN4OS	●		F1389.0, F1389.1, F1389.2, F1389.3	
	Composite control start signals	MIX1, MIX2, MIX3, MIX4	●		G1437.4, G1437.5, G1437.7, G1437.6,	
Angular axis control (0–GCC, 0–GSC)	Angular axis control–related Z–axis compensation movement signal	NOZAGC	☆		G133.6	1.8
				☆	G237.5	
Position signal output (T series)	Position signal output signal	PX0 to PX7, PZ0 to PZ7	○		F165.0 to F165.7, F167.0 to F167.7	1.9
Cf axis control (T series)	C–axis–off signal	COFF	○		G123.0	1.10
Emergency stop	Emergency stop signal	*ESP, *ESP	○	○	X021.4, G121.4	2.1
CNC ready signal	Servo ready signal	SA	○	○	F148.6	2.2
	CNC ready signal	MA	○	○	F149.7	
Overtravel check	Overtravel signal	*+LZ	○		X018.5	2.3.1
		*+LX, *+LY, *+LZ, *–LX, *–LY, *–LZ		○	X020.0, X020.1, X020.2, X020.3, X020.4, X020.5	
Stored stroke check 1	Stroke check external setting signals	+LMX, +LMY, +LMZ, –LMX, –LMY, –LMZ		○	G129.0, G129.1, G129.2, G129.3, G129.4, G129.5	2.3.2
	Stroke check release signal	RLSOT		○	G129.7	
	Stored stroke check select signal	EXLM2	○	○	G129.6	
Tool post interference check (0–TTC)	Tool post interference alarm signal	TAL	●		F180.7	2.3.4
	Tool post interference check signal	BOFF	●		F180.6	
Alarm signal	Alarm signal	AL	○	○	F149.0	2.4
	Battery alarm signal	BAL	○	○	F149.2	
	Absolute pulse coder battery alarm signal	BAL1 to BAL4,BAL7,BAL8		○	F159.0 to F159.5	
			○		F156.0 to F156.5	
Start lock/interlock	Interlock signal for each axis	*ITX, *ITY, *ITZ, *IT4		○	G128.0, G128.1, G128.2, G128.3	2.5
		ITX, ITZ, IT3, IT4	○			
	Interlock signal for each axis and direction	+MIT1, –MIT1, +MIT2, –MIT2	○		X008.2, X008.3 X008.4, X008.5	
		*+MITX, *+MITY, *+MITZ, *+MIT4, *–MITX, *–MITY, *–MITZ, *–MIT4		○	G142.0, G142.1, G142.2, G142.3, G142.4, G142.5, G142.6, G142.7	
	High–speed interlock signal	*RILK		○	X008.5	

Function	Signal name	Symbol	T	M	Address	Ref. item
Start lock/interlock	Start lock signal	STLK	○		G120.1	2.5
	Interlock signal	*ILK		○	G117.0	
Mode selection	Mode selection signal	MD1, MD2, MD4	○	○	G122.0, G122.1, G122.2	2.6
		DNCI	○	○	G127.5	
		ZRN	○	○	G120.7	
Tool post selection (0–TTC)	Tool post selection signal	TRT2, TRT2PC	●		X018.3, G133.0	2.7
Status output signal	Cutting feed signal	CUT	○	○	F188.6	2.8
	Canned cycle start signal	FXST		○	F161.4	
VRDY off alarm ignore signal	All–axis CRDY OFF alarm ignore signal	IGNVRY		○	G123.0	2.9
		IGNVRY	○		G127.0	
Jog feed/incremental feed	Feedrate override signal	*OV1, *OV2, *OV4, *OV8	○	○	G121.0, G121.1, G121.2, G121.3	3.1
	Feed axis and direction select signal	+X, –X, +Z, –Z, +3, –3, +4, –4	○		G116.2, G116.3, G117.2, G117.3, G118.2, G118.3, G119.2, G119.3	
		+X, –X, +Y, –Y, +Z, –Z, +4, –4		○		
	Jog feedrate override signal	JOV1 to JOV8		○	G104.0 to G104.3	
	Manual rapid traverse select signal	RT	○	○	G121.6	
Manual handle feed	Manual handle feed axis selection signal for Z axis	SLHZ0, SLHZ1		○	G133.0, G133.1	3.2
	Manual handle feed axis selection signal	HX, HZ, H3, H4	○		G116.7, G117.7, G118.7, G119.7	
		HX, HY, HZ, H4		○		
	Manual handle heed amount selection signal	MP1, MP2	○		G117.0, G118.0	
		MP1, MP2		○	G120.0, G120.1	
Manual handle interruption	Manual handle interrupt axis selection signal	HIX, HIY, HIZ, HI4		○	G126.0, G126.1, G126.2, G126.3	3.3
		HIX, HIZ, HI3, HI4	○			
Manual reference position return	Manual reference position return selection signal	ZRN	○	○	G120.7	4.1
	Reference position establishment signal	ZRFX, ZRFZ, ZRF3, ZRF4	○		F168.0, F168.1, F168.2, F168.3	
		ZRFX, ZRFY, ZRFZ, ZRF4		○		
	Reference position return completion signal	ZPX, ZPZ, ZP3, ZP4	○		F148.0, F148.1, F148.2, F148.3	
		ZPX, ZPY, ZPZ, ZP4		○		
	Reference position return deceleration signals	*DECX, *DECZ, *DEC3, *DEC4,	○		X016.5, X017.5, X018.5, X019.5	
		*DECX, *DECY, *DECZ, *DEC4,		○		

Function	Signal name	Symbol	T	M	Address	Ref. item
2nd reference position return / 3rd, 4th reference position return	Second reference position return completion signals	ZP2X, ZP2Z, ZP23, ZP24	○		F161.0, F161.1, F161.2, F161.3	4.5
		ZP2X, ZP2Y, ZP2Z, ZP24		○		
	Third reference position return completion signals	ZP3X, ZP3Z, ZP33, ZP34	○		F169.0, F169.1, F169.2, F169.3	
		ZP3X, ZP3Y, ZP3Z, ZP34		○		
	Fourth reference position return completion signals	ZP4X, ZP4Z, ZP43, ZP44	○		F169.4, F169.5, F169.6, F169.7	
		ZP4X, ZP4Y, ZP4Z, ZP44		○		
Cycle start/feed hold	Manual data input start signal	DST	○	○	F150.5	5.1
	Feed hold lamp signal	SPL	○	○	F148.4	
	Automatic operation signal	OP	○	○	F148.7	
	Feed hold signal	*SP	○	○	G121.5	
	Cycle start lamp signal	STL	○	○	F148.5	
	Cycle start signal	ST	○	○	G120.2	
Reset and rewind	Reset and rewind signal	RRW	○	○	G104.6	5.2
	Resetting signal	RST	○	○	F149.1	
	Rewinding signal	RWD	○	○	F164.6	
	External reset signal	ERS	○	○	G121.7	
Machine lock	All–axis machine lock signal	MLK	○	○	G117.1	5.3.1
Dry run	Dry run signal	DRN	○	○	G118.7	5.3.2
Single block	Single block signal	SBK	○	○	G116.1	5.3.3
Manual absolute on/off	Manual absolute signal	*ABSM	○	○	G127.2	5.4
Optional block skip / addition of optional block skip	Optional block skip signals	BDT1, BDT2 to BDT9	○	○	G116.0, G141.0 to G141.7	5.5
Program restart	Program restart signal	SRN	○	○	G103.0	5.7
	Program restart under way signal	SRNMV	○	○	F188.4	
DNC operation	DNC operation select signal	DNCI	○	○	G127.5	5.10
Romote buffer DI/DO signals	Output signals for remote buffer	RMTDO0 to RMTDO7	○		F289.0 to F289.7	5.11
				○	F290.0 to F290.7	
	Input signals for remote buffer	RMTDI0 to RMTDI7	○	○	G239.0 to G239.7	
In–feed control	In–feed control cut–in start signal	INFD		☆	G237.6	5.12
Thread cutting	Thread cutting signal	THRD	○		F188.3	6.4.1
Polygonal turning	Polygon synchronization under way signal	PSYN	○		F160.7	6.9.1
F1–digit feed	F1–digit feed select signal	F1D		○	G140.7	7.1.5
Rapid traverse override	Rapid traverse override signal	ROV1, ROV2	○	○	G116.7, G117.7	7.1.6.1

Function	Signal name	Symbol	T	M	Address	Ref. item
Feedrate override	Feedrate override signal	*OV1, *OV2, *OV4, *OV8	○	○	G121.0, G121.1, G121.2, G121.3	3.1, 7.1.6.2
		*AOV16, *AOV32, *AOV64, *AOV128		○	G116.4, G116.5, G116.6, G117.6	7.1.6.2
		*AOVR16, *AOVR32, *AOVR64, *AOVR128	○		G140.4, G140.5, G140.6, G140.7	
Rapid traverse override B	Rapid traverse override B signal	ROV1D, ROV2D, ROV3D	○		G116.4, G116.5, G116.6	7.1.6.3
Override cancel	Override cancel signal	OVC	○	○	G126.4	7.1.6.4
External deceleration	External deceleration signal	*+EDCX, *+EDCZ, *–EDCX , *–EDCZ	○		G138.0, G138.1, G138.3, G138.4	7.1.8
		*+EDCX, *+EDCY, *+EDCZ, *–EDCX, *–EDCY, *–EDCZ		○	G138.0, G138.1, G138.2, G138.3, G138.4, G138.5	
In–position check	In–position signals	INP1 to INP4	○	○	F184.0 to F184.3	7.2.5.1
Error detect	Error detect signal	SMZ	○		G126.6	7.2.5.3
Signals output according to the speed or travel along an axis (M series)	Signals output according to the speed or travel along an axis	SPDS1, SPDS2, SPDS3, SPDS4		○	F189.0, F189.1, F189.2, F189.3	7.2.5.4
Miscellaneous function / 2nd auxiliary function	Tool function code signals	T11 to T28	○		F153.0 to F153.7	8.1
		T11 to T48		○	F153.0 to F156.7	
	Tool function strobe signals	TF	○	○	F150.3	
	Second auxiliary function code signals	B0 to B31	○		F276.0 to F279.7	
		B11 to B38		○	F155.0 to F154.3	
	Second auxiliary function strobe signals	BF	○		F150.7	
		BF1, BF2		○	F150.7, F150.6	
	Decode M signals	M00, M01, M02, M30		○	F154.7, F154.6, F154.5, F154.4	
	Miscellaneous function code signal	M11, M12, M14, M18, M21, M22, M24, M28, M31, M32, M34, M38	○	○	F151.0, F151.1, F151.2, F151.3, F151.4, F151.5, F151.6, F151.7, F157.0, F157.1, F157.2, F157.3	
	Miscellaneous function strobe signal	MF	○	○	F150.0	
	End signal	FIN	○	○	G120.3	
	Distribution end signals	DEN	○	○	F149.3	
	Passing point signal	DEN2	○		F149.6	
	Spindle–speed code signals	S11, S12, S14, S18, S21, S22, S24, S28	○	○	F152.0, F152.1, F152.2, F152.3, F152.4, F152.5, F152.6, F152.7	8.1, 9.3

Function	Signal name	Symbol	T	M	Address	Ref. item
Miscellaneous function / 2nd auxiliary function	Spindle–speed strobe signals	SF	○	○	F150.2	8.1, 9.3
Auxiliary function lock	Auxiliary function lock signal	AFL	○	○	G103.7	8.2
Multiple M commands in a single block	2nd M function code signal	M211, M212, M214, M218, M221, M222, M224, M228, M231, M232, M234, M238	○	○	F193.0, F193.1, F193.2, F193.3, F193.4, F193.5, F193.6, F193.7, F194.0, F194.1, F194.2, F194.3	8.3
	3rd M function code signal	M311, M312, M314, M318, M321, M322, M324, M328, M331, M332, M334, M338	○	○	F194.4, F194.5, F194.6, F194.7, F195.0, F195.1, F195.2, F195.3, F195.4, F195.5, F195.6, F195.7	
	2nd, 3rd M function strobe signal	MF2, MF3	○	○	F157.4, F157.5	
High–speed M/S/T/B interface	Miscellaneous function completion signal	MFIN	○	○	G115.0	8.4
	2nd, 3rd M function completion signal	MFIN2, MFIN3	○	○	G134.4, G134.5	8.4
	External operation signal for high–speed interface	EF		○	F150.1	
	2nd auxiliary function completion signal	BFIN	○		G115.7	
		BFIN1, BFIN2		○	G115.7, G115.6	8.4
	Tool function completion signal	TFIN	○	○	G115.3	8.4
	Spindle function completion signal	SFIN	○	○	G115.2	
	External operation function completion signal	EFIN		○	G115.1	
Waiting M code (0–TTC)	Waiting signal	WATO	●		F160.6	8.5
	No–wait signal	NOWT	●		G133.1	
Spindle speed control	Spindle enable signal	ENB	○	○	F149.4	9.3
	Gear selection signal	GR1, GR2	○		G118.2, G118.3	9.3, 9.5, 9.8, 9.9, 9.10, 9.11
				○	G123.2, G123.3	
		GR1O, GR2O, GR3O		○	G152.0, G152.1, G152.2	9.3, 9.9
	Spindle speed override signal	SPA, SPB, SPC, SPD	○		G103.3, G103.4, G103.5, G103.2	9.3
	Spindle speed override signal	SPA, SPB, SPC		○	G103.3, G103.4, G103.5	

Function	Signal name	Symbol	T	M	Address	Ref. item
Spindle speed control	S12–bit code signal	R01O to R12O	○	○	F172.0 to F173.3	9.3, 15.4
	Spindle speed arrival signal	SAR	○	○	G120.4	9.3
	Spindle orientation signal	SOR	○	○	G120.5	
	Spindle stop signal	*SSTP	○	○	G120.6	
Spindle speed control for 0–TTC	Spindle command select signal	SLSPA, SLSPB	●		G133.2, G133.3	9.4
	Spindle feedback select signal;	SLPCA, SLPCB	●		G1333.2, G1333.3	
	Spindle command signal;	COSP	●		F180.5	
Spindle speed fluctuation detection	Spindle fluctuation detection alarm signal	SPAL	○		F154.0	9.6
Actual spindle speed otuput (T series)	Actual spindle speed signal	AR0 to AR15	○		F158.0 to F159.7	9.7
Spindle positioning (T series)	Spindle unclamp completion signal	*SUCPF	○		G123.4	9.8
	Spindle clamp signal	SCLP	○		F164.0	
	Spindle unclamp signal	SUCLP	○		F164.1	
	Spindle stop complete signal	SPSTP	○		G123.6	
	Spindle clamp completion signal	*SCPF	○		G123.5	
	Spindle orientation completion signal	ZP3	○		F148.2	
	Gear selection signal	CTH1A CTH2A	○	○	G229.3, G229.2	
Cs contour control	Spindle contour control change signal	COFF	○		G123.0	9.9
		CON		○	G123.7	
	Clutch/gear signal (serial spindle)	CTH1A CTH2A	○	○	G229.3, G229.2	
	Spindle contour control change completion signal	FSCSL		○	F178.1	
	Cs contour control axis reference position return completion signal	ZP3	○		F148.2	
		ZP4		○	F148.3	
Multi spindle	Individual spindle stop signal	*SSTP1, *SSTP2, *SSTP3	○		G145.3, G145.4, G145.5	9.10, 9.11
	Gear selection signal for 2nd spindle	GR21	○		G145.6	
	Gear selection signal for 3rd spindle	GR31	○		G145.7	
	Spindle selection signal	SWS1, SWS2, SWS3	○		G145.0, G145.1, G145.2	
	2nd position coder selection signal	PC2SLC	○		G146.7	

Function	Signal name	Symbol	T	M	Address	Ref. item
Spindle speed control	Spindle enable signal	ENB	○	○	F149.4	9.3, 9.10, 9.11
		ENB2, ENB3	○		F149.4, F164.2, F164.3	
	Gear selection signal	GR1O, GR2O, GR3O		○	G152.0, G152.1, G152.2	
Rigid tapping	Spindle rotation direction signal	RGSPM, RGSPP		○	F165.1, F165.0	9.11
	Spindle–speed function code signal (BCD output)	S11 to S58	○	○	F185.0 to F187.3	
	Spindle function strobe signal	SF	○	○	F150.2	
	Rigid tapping signal	RGTPN	○	○	G123.1	
Rigid tapping return	Tapping return start signal	RTNT		○	G123.6	9.12
	Tapping return completion signal	RTPT		○	F192.6	
Spindle synchronous control	Spindle synchronous control signal	SPSYC	○	○	G146.2	9.13
	Spindle phase synchronous control signal	SPPHS	○	○	G146.3	
	Spindle synchronous speed control completion signal	FSPSY	○	○	F178.2	
	Spindle phase synchronous control completion signal	FSPPH	○	○	F178.3	
	Spindle synchronous control alarm signal	SYCAL	○	○	F178.4	9.13, 9.18
Controlling the spindle speed ratio for serial interface spindles (0–TTC)	Spindle synchronous polygon code signal	SBRT	●		G146.6	9.14
	Serial spindle synchronization polygon signal	RSMAX	●		F189.7	
Spindle orientation	1st spindle orientation external stop position command signal	SHA00 to SHA11	○	○	G110.0 to G110.7, G111.0 to G111.3	9.15
	2nd spindle orientation external stop position command signal	SHB00 to SHB11	○	○	G112.0 to G112.7, G113.0 to G113.3	
Position coder feedback control function	Position coder return control signal	PLCRVON	○	○	G105.7	9.17
	Position coder feedback direction selection signal	KILPLUS	○	○	G105.6	
Serial spindle simple synchronous control	Parking signal for 1st spindle	PKESS1	○	○	G138.6	9.18
	Parking signal for 2nd spindle	PKESS2	○	○	G138.7	
Serial spindle simple synchronous control	Spindle simple synchronous control signal	ESRSYC	○	○	G104.4	9.18

Function	Signal name	Symbol	T	M	Address	Ref. item
Tool life management	Individual tool change signal	TLCHI		○	F192.0	10.3
	All tools' life expired signal	TLCHE		○	F192.1	
	New tool select signal	TLNW	○	○	F188.1	
	Remaining tool life expired signal	TLCHB		○	F192.2	
	Tool change signal	TLCH	○	○	F188.0	
	Individual tool change reset signal	TLRSTI		○	G140.4	
	Tool skip signal	TLSKP	○	○	G140.0	
	Tool group number select signal	TL01, TL02, TL04, TL08, TL16, TL32, TL64	○	○	G139.0, G139.1, G139.2, G139.3, G139.4, G139.5, G139.6	
	Tool change reset signal	TLRST	○	○	G139.7	
Custom macro	Custom macro output signal	UO000 to UO015, UO100 to UO131	○	○	F162.0 to F163.7, F196.0 to F199.7	11.6.1
	Custom macro input signal	UI000 to UI015	○	○	G130.0 to G131.7	
Interruption type custom macro	Interrupt signal for custom macro	UINT	○	○	G140.3	11.6.2
Canned cycle (M series) / Canned cycle for hole machining (T series)	Tapping signal	TAP	○	○	F149.5	11.7
Smalldiameter peck drilling cycle (M series)	Skip signal	SKIP	○	○	X008.7	11.8
	Small–diameter peck drilling cycle execution in progress signal	PECK2		○	F180.7	
External motion function (M series)	External operation signal	EF		○	F150.1	11.9
Canned cycle	Chamfering signal	CDZ	○		G126.7	11.10
Index table indexing function (M series)	B axis clamp completion signal	*BECLP		○	G143.7	11.12
	B axis unclamp signal	BUCLP		○	F188.2	
	B axis clamp signal	BCLP		○	F188.3	
	B axis unclamp completion signal	*BEUCP		○	G143.6	
Display/setting	Display and setting	*LDSP	○	○	G119.0	12.1
	Renewal disable signal of relative coordinate	DLK	○	○	G127.6	
Run hour and parts count display	Required parts count reached signal	PRTSF	○	○	F164.7	12.1.7
Graphic display / Dynamic graphic display	Drwing signal	CKGRP	○	○	F164.5	12.1.8

Function	Signal name	Symbol	T	M	Address	Ref. item
Software operator's panel	Software operator's panel general–purpose switch signal (software operator's panel signal)	OUT0 to OUT7	○	○	F171.0 to F171.7	12.1.10
	Jog rapid traverse select signal (software operator's panel signal)	RTO	○	○	F178.6	
	Feed hold signal (software operator's panel signal)	SPO	○	○	F178.7	
	Mode select signal (software operator's panel signal)	MD1O, MD2O, MD4O	○	○	F174.0, F174.1, F174.2	
	Manual pulse generator feed axis select signal (software operator's panel signal)	HXO, HYO, HZO, HZ4		○	F174.4, F174.5, F174.6, F174.7	
		HXO, HZO, H3O, H4O	○			
	Jog feedrate override signal (software operator's panel signal)	JV1O, JV2O, JV4O, JV8O	○	○	F175.0, F175.1, F175.2, F175.3	
	Mode select signal (software operator's panel signal)	ZRNO	○	○	F174.3	
	Dry run signal (software operator's panel signal)	DRNO	○	○	F176.7	
	Machine lock signal (software operator's panel signal)	MLKO	○	○	F176.6	
	Single block signal (software operator's panel signal)	SBKO	○	○	F176.5	
	Optional block skip signal (software operator's panel signal)	BDTO	○	○	F176.4	
	Jog feed axis select signal (software operator's panel signal)	+XO, –XO, +ZO, –ZO, +YO, –YO, +4O, –4O	○		F177.0, F177.1, F177.2, F177.3, F177.4, F177.5, F177.6, F177.7	
		+XO, –XO, +YO, –YO, +ZO, –ZO, +4O, –4O		○		
	Rapid traverse override signal (software operator's panel signal)	ROV1O, ROV2O	○	○	F175.4, F175.5	
	Program protect signal (software operator's panel signal)	KEYO	○	○	F178.5	
	Feedrate override signal (software operator's panel signal)	OV1O, OV2O, OV4O, OV8O	○	○	F176.0, F176.1, F176.2, F176.3	
	Select of magnification of manual pulse generator (software operator's panel signal)	MP1O, MP2O	○	○	F175.6, F175.7	
Memory protection key	Program protect signal	KEY	○	○	G122.3	12.2.3

Function	Signal name	Symbol	T	M	Address	Ref. item
External I/O device control	Read/punch alarm signal	RPALM	○	○	F180.3	13.5
	External read/punch stop signal	EXSTP	○	○	G134.2	
	External punch start signal	EXWT	○	○	G134.3	
	Background editing signal	BGEACT	○	○	F180.4	
	External read start signal	EXRD	○	○	G134.1	
	Read/punch busy signal	RPBSY	○	○	F180.2	
Simltaneous input and output operations	Input and run simultaneous mode select signal	STRD		○	G140.5	13.6
	Output and run simultaneous mode select signal	STWD		○	G140.6	
External program input	External program input start signal	MINP	○		G117.0	13.7
				○	G120.0	
Automatic tool length measurement (M series) / automatic tool offset (T series)	Measuring position reached signals	XAE, ZAE	○		X008.0, X008.1	14.2
		XAE, YAE, ZAE		○	X008.0, X008.1, X008.2	
Skip function	Skip signal	SKIP	○	○	X008.7	14.3.1, 14.3.3
Multi–step skip function	Skip signal	SKIP2, SKIP3, SKIP4	☆		X008.2, X008.3, X008.4	14.3.3
Torque limit skip (T series)	Torque limit reached signal	TORQ1, TORQ2, TORQ3, TORQ4, TORQ7, TORQ8	○		F170.0, F170.1, F170.2, F170.3, F170.4, F170.5	14.3.4
Input of tool offset value measured A	Position record signal	PRC	○		G103.6	14.4.1
Input of tool offset value measured B (T series)	Tool offset write mode select signal	GOQSM	○		G132.7	14.4.2
	Tool offset write signal	+MIT1, –MIT1, +MIT2, –MIT2	○		X008.2, X008.3 X008.4, X008.5	2.5, 14.4.2
	Tool offset number select signal	OFN0, OFN1, OFN2, OFN3, OFN4	○		G132.0, G132.1, G132.2, G132.3, G132.4	14.4.2
	Workpiece coordinate system shift value write signal	WOSET	○		G133.7	
	Workpiece coordinate system shift value write mode select signal	WOQSM	○		G132.6	
	Tool compensation number automatic selection disable signal	TNFS	○		G132.5	
Grinding wheel diameter automatic compensation	Grinding wheel diameter automatic compensation signal	GWLF		☆	F165.3	14.4.4

Function	Signal name	Symbol	T	M	Address	Ref. item
PMC axis control	Axis control command signal (PMC axis control)	EC0A to EC6A, EC0B to EC6B	○	○	G211.0 to G211.6, G219.0 to G219.6	15.1
	Auxiliary function executing signal (PMC axis control)	EDENA, EDENB	○	○	F270.3, F273.3	
	Alarm signal (PMC axis control)	EIALA, EIALB	○	○	F270.2, F273.2	
	Control axis selection status signal (PMC axis control)	*EAXSL	○	○	F188.7	
	Following zero checking signal (PMC axis control)	ECKZA, ECZKB	○	○	F270.1, F273.1	
	Override 0% signal (PMC axis control)	EOV0	○	○	F188.5	
	In–position signal (PMC axis control)	EINPA, EINPB	○	○	F270.0, F273.0	
	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A, EIF0B to EIF15B	○	○	G212.0 to G213.7, G220.0 to G221.7	
	Positive–direction overtravel signal (PMC axis control)	EOTPA, EOTPB	○	○	F270.5, F273.5	
	Skip signal (PMC axis control)	ESKIP	○	○	X008.6	
	Axis moving signal (PMC axis control)	EGENA, EGENB	○	○	F270.4, F273.4	
	Axis control data signal (PMC axis control)	EID0A to EID31A, EID0B to EID31B	○	○	G214.0 to G217.7, G222.0 to G225.7	
	Auxiliary function strobe signal	EMFA, EMFB	○	○	F271.0, F274.0	
	Axis control command read completion signal (PMC axis control)	EBSYA, EBSYB	○	○	F270.7, F273.7	
	Negative–direction overtravel signal (PMC axis control)	EOTNA, EOTNB	○	○	F270.6, F273.6	
	Dry run signal (PMC axis control)	DRNE	○	○	G147.7	
	Block stop signal (PMC axis control)	ESBKA, ESBKB	○	○	G210.3, G218.3	
	Auxiliary function completion signal (PMC axis control)	EFINA, EFINB	○	○	G210.0, G218.0	
	Servo off signal (PMC axis control)	ESOFA, ESOFB	○	○	G210.4, G218.4	
	Axis control temporary stop signal (PMC axis control)	ESTPA, ESTPB	○	○	G210.5, G218.5	
	Manual rapid traverse selection signal (PMC axis control)	RTE	○	○	G147.6	

Function	Signal name	Symbol	T	M	Address	Ref. item
PMC axis control	Auxiliary function code signal (PMC axis control)	EM11A to EM28A, EM11B to EM28B	○	○	F272.0 to F272.7, F275.0 to F275.7	15.1
	Reset signal (PMC axis control)	ECLRA, ECLRB	○	○	G210.6, G218.6	
	Axis control command read signal (PMC axis control)	EBUFA, EBUFB	○	○	G210.7, G218.7	
	PMC axis rapid traverse override signal	ROV1E, ROV2E	○	○	G146.0, G146.1	
	Block stop disable signal (PMC axis control)	EMSBKA, EMSBKB	○	○	F211.0, G219.0	
	Control axis selection signal (PMC axis control)	EAX1 to EAX8	○	○	G144.0 to G144.5	
	PMC axis override cancel signal	OVCE	○	○	G147.5	
	Feedrate override signal (PMC axis control)	*OV1E, *OV2E, *OV4E, *OV8E	○	○	G147.0, G147.1, G147.2, G147.3	
External data input	Read completion signal (for external data input)	EREND	○	○	F160.0	15.2
	Search completion signal (for external data input)	ESEND	○	○	F160.1	
	Read signal (for external data input)	ESTB	○	○	G102.7	
	Address signal (for external data input)	EA0 to EA6	○	○	G102.0 to G102.6	
	Data signal (for external data input)	ED0 to ED15	○	○	G100.0 to G101.7	
External workpiece number search	Workpiece number search signal	PN1, PN2, PN4, PN8	○	○	G122.4, G122.5, G122.6, G122.7	15.3
Spindle output control by the PMC	Motor speed command spindle input signal	R01I to R12I, R01I2 to R12I2, R01I3 to R12I3	○	○	G124.0 to G125.3, G106.0 to G107.3, G108.0 to G109.3	9.10, 15.4
	Spindle motor speed command signal	SIND, SIND2, SIND3	○	○	G125.7, G107.7, G109.7	
	Spindle motor command polarity select signal	SGN, SGN2, SGN3	○	○	G125.5, G107.5, G109.5	
	Spindle motor command polarity select signal	SSIN, SSIN2, SSIN3	○	○	G125.6, G107.6, G109.6	

Function	Signal name	Symbol	T	M	Address	Ref. item
External key input	Key signal	K0, S, K1, U, K2, W, K3, R, K4, X, K5, Z, K6, F, K7, 0, K8, N, K9, G, KA, C, KALARM, KALTER, KCAN, KCUS↑, KCUS÷, KDELET, KDGNOS, KEOB	○		G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G107.5, G109.5, G108.0, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7	15.5
		KINPUT, KINSRT, KM, –, KP, Q, KPAGE ↑, KPAGE ↓, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXZ, KXZ	○		G108.3, G108.1, G107.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G142.7, G110.4	
		KFF, KFFE, KFGB, KFI,, KFK%, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFUV, KFWH, KFXY, KFZJ, KH, I, K	○		G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.6, G113.7, G113.4, G113.5, G107.6	
		K0, S, K1, H, K2, F, K3, R, K4, X, K5, Y, K6, Z, K7, 0, K8, N, K9, G, KALARM, KALTER, KB, K4, D, KCAN, KCUS↑, KCUS↓, KDELET, KDGNOS, KEOB, / , .		○	G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G109.5, G108.0, G107.5, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7	

Function	Signal name	Symbol	T	M	Address	Ref. item
External key input	Key signal	KF4TH, KFF, KFF%, KFGE, KFI,, KFJA, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFXU, KFXW, KFYV, KH, J, K, KINPUT, KINSRT, KM, –, KOFSET, KP, Q, L, KPAGE↑, KPAGE↓, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXY, KYZ, KZX		○	G113.7, G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.4, G113.6, G113.5, G107.6, G108.3, G108.1, G107.2, G109.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G110.4, G110.6, G110.5	15.5
External key input	Key input disable signal	INHKY	○	○	F180.0	15.5
	Program screen display signal	PRGDPL	○	○	F180.1	
	External key input mode selection signal	ENBKY	○	○	G134.0	
Direct operation by MMC (T series)	Direct operation select signal	DMMC	○	○	G128.7	15.6
PMC Window function	Window request signal	WNRQ	○	○	G200.0	15.7
	Window completion signal	WNANS	○	○	F250.0	
	Window rewrite signal	WNWRT	○	○	G200.1	
	Window error signal	WNERR	○	○	F250.1	
Functions related to serial spindle	Soft start /stop cancel signal	SOCNA, SOCNB	○	○	G230.4, G234.4	Manual for serial spindle
	Spindle orientation stop position change signal	INDXA, INDXB	○	○	G231.0, G235.0	
	Spindle rotation direction command signal while changing the orientation stop position	ROTAA, ROTAB	○	○	G231.1, G235.1	
	Power line switch completion signal	MCFNA, MCFNB	○	○	G230.3, G234.3	
	Power line status check signal	RCHA, RCHB	○	○	G230.7, G234.7	
	Output switch request signal	RSLA, RSLB	○	○	G230.6, G234.6	
	Short–distant movement command while changing the orientation stop position signal	NRROA, NRROB	○	○	G231.2, G235.2	
	Speed reached signal	SARA, SARB	○	○	F281.3, F285.3	

Function	Signal name	Symbol	T	M	Address	Ref. item
Functions related to serial spindle	Output switching completion signal	PCFNA, PCFNB	○	○	F282.3, F286.3	Manual for serial spindle
	Output switching signal	PCHPA, PCHPB	○	○	F282.2, F286.2	
	Speed zero detection signal	SSTA, SSTB	○	○	F281.1, F285.1	
	Speed detection signal	SDTA, SDTB	○	○	F281.2, F285.2	
	Speed reached signal	SARA, SARB	○	○	F281.3, F285.3	
	Spindle switch completion signal	CFINA, CFINB	○	○	F282.1, F286.1	
	Load detection signal	LDT1A, LDT1B, LDT2A, LDT2B	○	○	F281.4, F285.4, F281.5, F285.5	
	Emergency stop signal for spindle	*ESPA, *ESPB	○	○	G230.1, G234.1	
	Torque limit signal	TLMA, TLMB	○	○	F281.6, F285.6	
	Spindle orientation completion signal	ORARA, ORARB	○	○	F281.7, F285.7	
	Power line switch signal	CHPA, CHPB	○	○	F282.0, F286.0	
	Spindle select signal	SPSLA, SPSLB	○	○	G230.2, G234.2	
	Spindle alarm signal	ALMA, ALMB		○	F281.0, F285.0	
	Alarm reset signal	ARSTA, ARSTB	○	○	G230.0, G234.0	
	Low speed torque limit signal	TLMLA, TLMLB	○	○	G229.0, G233.0	
	Machine ready signal	MRDYA, MRDYB	○	○	G229.7, G233.7	
	High–speed torque limit signal	TLMHA, TLMHB	○	○	G229.1, G233.1	
	Spindle CCW command signal	SRVA, SRVB	○	○	G229.4, G233.4	
	Orientation command signal	ORCMA, ORCMB	○	○	G229.6, G233.6	
	Spindle CW command signal	SFRA, SFRB	○	○	G229.5, G233.5	

A.2.2 List of Signals (In Order of Symbols)

Meaning of symbols in the below table
○: Available
●: 0–TTC only
☆: Available for 0–GCC or 0–GSC only
Blank : Unavailable

Symbol	Signal name	T	M	Address	Ref. item
*+EDCX, *+EDCZ, *–EDCX , *–EDCZ	External deceleration signal	○		G138.0, G138.1, G138.3, G138.4	7.1.8
*+EDCX, *+EDCY, *+EDCZ, *–EDCX, *–EDCY, *–EDCZ			○	G138.0, G138.1, G138.2, G138.3, G138.4, G138.5	
*+LX, *+LY, *+LZ, *–LX, *–LY, *–LZ	Overtravel signal		○	X020.0, X020.1, X020.2, X020.3, X020.4, X020.5	2.3.1
*+LZ		○		X018.5	
*+MITX, *+MITY, *+MITZ, *+MIT4, *–MITX, *–MITY, *–MITZ, *–MIT4	Interlock signal for each axis and direction		○	G142.0, G142.1, G142.2, G142.3, G142.4, G142.5, G142.6, G142.7	2.5
*ABSM	Manual absolute signal	○	○	G127.2	5.4
*AOV16, *AOV32, *AOV64, *AOV128	Feedrate override signal		○	G116.4, G116.5, G116.6, G117.6	7.1.6.2
*AOVR16, *AOVR32, *AOVR64, *AOVR128		○		G140.4, G140.5, G140.6, G140.7	
*BECLP	B axis clamp completion signal		○	G143.7	11.12
*BEUCP	B axis unclamp completion signal		○	G143.6	
*DECX, *DECY, *DECZ, *DEC4,	Reference position return deceleration signals		○	X016.5, X017.5, X018.5, X019.5	4.1
*DECX, *DECZ, *DEC3, *DEC4,		○			
*EAXSL	Control axis selection status signal (PMC axis control)	○	○	F188.7	15.1
*ESP, *ESP	Emergency stop signal	○	○	X021.4, G121.4	2.1
*ESPA, *ESPB	Emergency stop signal for spindle	○	○	G230.1, G234.1	Manual for serial spindle
*FLWU	Follow–up signal	○	○	G104.5	1.2.6
*ILK	Interlock signal		○	G117.0	2.5
*ITX, *ITY, *ITZ, *IT4	Interlock signal for each axis		○	G128.0, G128.1, G128.2, G128.3	
*LDSP	Display and setting	○	○	G119.0	12.1
*OV1, *OV2, *OV4, *OV8	Feedrate override signal	○	○	G121.0, G121.1, G121.2, G121.3	3.1
*OV1E, *OV2E, *OV4E, *OV8E	Feedrate override signal (PMC axis control)	○	○	G147.0, G147.1, G147.2, G147.3	15.1
*RILK	High–speed interlock signal		○	X008.5	2.5
*SCPF	Spindle clamp completion signal	○		G123.5	9.8

Symbol	Signal name	T	M	Address	Ref. item
*SP	Feed hold signal	○	○	G121.5	5.1
*SSTP	Spindle stop signal	○	○	G120.6	9.3
*SSTP1, *SSTP2, *SSTP3	Individual spindle stop signal	○		G145.3, G145.4, G145.5	9.10, 9.11
*SUCPF	Spindle unclamp completion signal	○		G123.4	9.8
+LMX, +LMY, +LMZ, –LMX, –LMY, –LMZ	Stroke check external setting signals		○	G129.0, G129.1, G129.2, G129.3, G129.4, G129.5	2.3.2
+MIT1, –MIT1, +MIT2, –MIT2	Interlock signal for each axis and direction	○		X008.2, X008.3 X008.4, X008.5	2.5
+X, –X, +Y, –Y, +Z, –Z, +4, –4	Feed axis and direction select signal		○	G116.2, G116.3, G117.2, G117.3, G118.2, G118.3, G119.2, G119.3	3.1
+X, –X, +Z, –Z, +3, –3, +4, –4		○			
+XO, –XO, +YO, –YO, +ZO, –ZO, +4O, –4O	Jog feed axis select signal (software operator’s panel signal)		○	F177.0, F177.1, F177.2, F177.3, F177.4, F177.5, F177.6, F177.7	12.1.10
+XO, –XO, +ZO, –ZO, +YO, –YO, +4O, –4O		○			
4NG	Ignore–the–fourth–axis–signal		○	X004.7	1.4.3
AFL	Auxiliary function lock signal	○	○	G103.7	8.2
AL	Alarm signal	○	○	F149.0	2.4
ALMA, ALMB	Spindle alarm signal		○	F281.0, F285.0	Manual for serial spindle
AR0 to AR15	Actual spindle speed signal	○		F158.0 to F159.7	9.7
ARSTA, ARSTB	Alarm reset signal	○	○	G230.0, G234.0	Manual for serial spindle
B0 to B31	Second auxiliary function code signals	○		F276.0 to F279.7	8.1
B11 to B38			○	F155.0 to F154.3	
BAL	Battery alarm signal	○	○	F149.2	2.4
BAL1 to BAL4,BAL7,BAL8	Absolute pulse coder battery alarm signal		○	F159.0 to F159.5	
		○		F156.0 to F156.5	
BCLP	B axis clamp signal		○	F188.3	11.12
BDT1, BDT2 to BDT9	Optional block skip signals	○	○	G116.0, G141.0 to G141.7	5.5
BDTO	Optional block skip signal (software operator’s panel signal)	○	○	F176.4	12.1.10
BF	Second auxiliary function strobe signals	○		F150.7	8.1
BF1, BF2			○	F150.7, F150.6	
BFIN	2nd auxiliary function completion signal	○		G115.7	8.4
BFIN1, BFIN2			○	G115.7, G115.6	
BGEACT	Background editing signal	○	○	F180.4	13.5
BOFF	Tool post interference check signal	●		F180.6	2.3.4

Symbol	Signal name	T	M	Address	Ref. item
BUCLP	B axis unclamp signal		○	F188.2	11.12
CDZ	Chamfering signal	○		G126.7	11.10
CFINA, CFINB	Spindle switch completion signal	○	○	F282.1, F286.1	Manual for serial spindle
CHPA, CHPB	Power line switch signal	○	○	F282.0, F286.0	
CKGRP	Drwing signal	○	○	F164.5	12.1.8
COFF	C–axis–off signal	○		G123.0	1.10
COFF	Spindle contour control change signal	○		G123.0	9.9
CON			○	G123.7	
COSP	Spindle command signal;	●		F180.5	9.4
CUT	Cutting feed signal	○	○	F188.6	2.8
CTH1A CTH2A	Gear selection signal	○	○	G229.3, G229.2	9.8
CTH1A CTH2A	Clutch/gear signal (serial spindle)	○	○	G229.3, G229.2	9.9
DEN	Distribution end signals	○	○	F149.3	8.1
DEN2	Passing point signal	○		F149.6	
DLK	Renewal disable signal of relative coordinate	○	○	G127.6	12.1
DMMC	Direct operation select signal	○	○	G128.7	15.6
DNCI	Mode selection signal	○	○	G127.5	2.6
	DNC operation select signal				5.10
DRNO	Dry run signal (software operator’s panel signal)	○	○	F176.7	12.1.10
DRNE	Dry run signal (PMC axis control)	○	○	G147.7	15.1
DRN	Dry run signal	○	○	G118.7	5.3.2
DST	Manual data input start signal	○	○	F150.5	5.1
EA0 to EA6	Address signal (for external data input)	○	○	G102.0 to G102.6	15.2
EAX1 to EAX8	Control axis selection signal (PMC axis control)	○	○	G144.0 to G144.5	15.1
EBSYA, EBSYB	Axis control command read completion signal (PMC axis control)	○	○	F270.7, F273.7	
EBUFA, EBUFB	Axis control command read signal (PMC axis control)	○	○	G210.7, G218.7	
EC0A to EC6A, EC0B to EC6B	Axis control command signal (PMC axis control)	○	○	G211.0 to G211.6, G219.0 to G219.6	
ECKZA, ECZKB	Following zero checking signal (PMC axis control)	○	○	F270.1, F273.1	
ECLRA, ECLRB	Reset signal (PMC axis control)	○	○	G210.6, G218.6	
ED0 to ED15	Data signal (for external data input)	○	○	G100.0 to G101.7	15.2
EDENA, EDENB	Auxiliary function executing signal (PMC axis control)	○	○	F270.3, F273.3	15.1

Symbol	Signal name	T	M	Address	Ref. item
EF	External operation signal		○	F150.1	11.9
EF	External operation signal for high–speed interface		○	F150.1	8.4
EFIN	External operation function completion signal		○	G115.1	
EFINA, EFINB	Auxiliary function completion signal (PMC axis control)	○	○	G210.0, G218.0	15.1
EGENA, EGENB	Axis moving signal (PMC axis control)	○	○	F270.4, F273.4	
EIALA, EIALB	Alarm signal (PMC axis control)	○	○	F270.2, F273.2	
EID0A to EID31A, EID0B to EID31B	Axis control data signal (PMC axis control)	○	○	G214.0 to G217.7, G222.0 to G225.7	
EIF0A to EIF15A, EIF0B to EIF15B	Axis control feedrate signal (PMC axis control)	○	○	G212.0 to G213.7, G220.0 to G221.7	
EINPA, EINPB	In–position signal (PMC axis control)	○	○	F270.0, F273.0	
EM11A to EM28A, EM11B to EM28B	Auxiliary function code signal (PMC axis control)	○	○	F272.0 to F272.7, F275.0 to F275.7	
EMFA, EMFB	Auxiliary function strobe signal	○	○	F271.0, F274.0	
EMSBKA, EMSBKB	Block stop disable signal (PMC axis control)	○	○	F211.0, G219.0	
ENB	Spindle enable signal	○	○	F149.4	9.3
ENB	Spindle enable signal	○	○	F149.4	9.3, 9.10, 9.11
ENB2, ENB3		○		F149.4, F164.2, F164.3	
ENBKY	External key input mode selection signal	○	○	G134.0	15.5
EOTNA, EOTNB	Negative–direction overtravel signal (PMC axis control)	○	○	F270.6, F273.6	15.1
EOTPA, EOTPB	Positive–direction overtravel signal (PMC axis control)	○	○	F270.5, F273.5	
EOV0	Override 0% signal (PMC axis control)	○	○	F188.5	
EREND	Read completion signal (for external data input)	○	○	F160.0	15.2
ERS	External reset signal	○	○	G121.7	5.2
ESBKA, ESBKB	Block stop signal (PMC axis control)	○	○	G210.3, G218.3	15.1
ESEND	Search completion signal (for external data input)	○	○	F160.1	15.2
ESKIP	Skip signal (PMC axis control)	○	○	X008.6	15.1
ESOFA, ESOFB	Servo off signal (PMC axis control)	○	○	G210.4, G218.4	
ESRSYC	Spindle simple synchronous control signal	○	○	G104.4	9.18
ESTB	Read signal (for external data input)	○	○	G102.7	15.2
ESTPA, ESTPB	Axis control temporary stop signal (PMC axis control)	○	○	G210.5, G218.5	15.1

Symbol	Signal name	T	M	Address	Ref. item
EXLM2	Stored stroke check select signal	○	○	G129.6	2.3.2
EXRD	External read start signal	○	○	G134.1	13.5
EXSTP	External read/punch stop signal	○	○	G134.2	
EXWT	External punch start signal	○	○	G134.3	
F1D	F1–digit feed select signal		○	G140.7	7.1.5
FIN	End signal	○	○	G120.3	8.1
FSCSL	Spindle contour control change completion signal		○	F178.1	9.9
FSPPH	Spindle phase synchronous control completion signal	○	○	F178.3	9.13
FSPSY	Spindle synchronous speed control completion signal	○	○	F178.2	
FXST	Canned cycle start signal		○	F161.4	2.8
GOQSM	Tool offset write mode select signal	○		G132.7	14.4.2
GR1, GR2	Gear selection signal	○		G118.2, G118.3	9.3, 9.5, 9.8, 9.9, 9.10, 9.11
			○	G123.2, G123.3	
GR1O, GR2O, GR3O	Gear selection signal		○	G152.0, G152.1, G152.2	9.3, 9.9, 9.10, 9.11
GR21	Gear selection signal for 2nd spindle	○		G145.6	9.10, 9.11
GR31	Gear selection signal for 3rd spindle	○		G145.7	
GWLF	Grinding wheel diameter automatic compensation signal		☆	F165.3	14.4.4
HIX, HIY, HIZ, HI4	Manual handle interrupt axis selection signal		○	G126.0, G126.1, G126.2, G126.3	3.3
HIX, HIZ, HI3, HI4		○			
HX, HZ, H3, H4	Manual handle feed axis selection signal	○		G116.7, G117.7, G118.7, G119.7	3.2
HX, HY, HZ, H4			○		
HXO, HYO, HZO, HZ4	Manual pulse generator feed axis select signal (software operator's panel signal)		○	F174.4, F174.5, F174.6, F174.7	12.1.10
HXO, HZO, H3O, H4O		○			
IGNVRY	All–axis CRDY OFF alarm ignore signal		○	G123.0	2.9
IGNVRY		○		G127.0	
INDXA, INDXB	Spindle orientation stop position change signal	○	○	G231.0, G235.0	Manual for serial spindle
INFD	In–feed control cut–in start signal		☆	G237.6	5.12
INHKY	Key input disable signal	○	○	F180.0	15.5
INP1 to INP4	In–position signals	○	○	F184.0 to F184.3	7.2.5.1
ITX, ITZ, IT3, IT4	Interlock signal for each axis	○		G128.0, G128.1, G128.2, G128.3	2.5

Symbol	Signal name	T	M	Address	Ref. item
JOV1 to JOV8	Jog feedrate override signal		○	G104.0 to G104.3	3.1
JV1O, JV2O, JV4O, JV8O	Jog feedrate override signal (software operator's panel signal)	○	○	F175.0, F175.1, F175.2, F175.3	12.1.10
K0, S, K1, H, K2, F, K3, R, K4, X, K5, Y, K6, Z, K7, 0, K8, N, K9, G, KALARM, KALTER, KB, K4, D, KCAN, KCUS↑, KCUS↓, KDELET, KDGNOS, KEOB, / , .	Key signal		○	G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G109.5, G108.0, G107.5, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7	15.5
K0, S, K1, U, K2, W, K3, R, K4, X, K5, Z, K6, F, K7, 0, K8, N, K9, G, KA, C, KALARM, KALTER, KCAN, KCUS↑, KCUS↓, KDELET, KDGNOS, KEOB	Key signal	○		G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G107.5, G109.5, G108.0, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7	15.5
KEY	Program protect signal	○	○	G122.3	12.2.3
KEYO	Program protect signal (software operator's panel signal)	○	○	F178.5	12.1.10
KF4TH, KFF, KFF%, KFGE, KFI,, KFJA, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFXU, KFXW, KFYV, KH, J, K, KINPUT, KINSRT, KM, –, KOFSET, KP, Q, L, KPAGE↑, KPAGE÷, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXY, KYZ, KZX	Key signal		○	G113.7, G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.4, G113.6, G113.5, G107.6, G108.3, G108.1, G107.2, G109.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G110.4, G110.6, G110.5	15.5
KFF, KFFE, KFGB, KFI,, KFK%, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFUV, KFWH, KFXY, KFZJ, KH, I, K	Key signal	○		G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.6, G113.7, G113.4, G113.5, G107.6	15.5
KILPLUS	Position coder feedback direction selection signal	○	○	G105.6	9.17
KINPUT, KINSRT, KM, –, KP, Q, KPAGE ↑, KPAGE ÷, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXZ, KXZ	Key signal	○		G108.3, G108.1, G107.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G142.7, G110.4	15.5
LDT1A, LDT1B, LDT2A, LDT2B	Load detection signal	○	○	F281.4, F285.4, F281.5, F285.5	Manual for serial spindle

Symbol	Signal name	T	M	Address	Ref. item
M00, M01, M02, M30	Decode M signals		○	F154.7, F154.6, F154.5, F154.4	8.1
M11, M12, M14, M18, M21, M22, M24, M28, M31, M32, M34, M38	Miscellaneous function code signal	○	○	F151.0, F151.1, F151.2, F151.3, F151.4, F151.5, F151.6, F151.7, F157.0, F157.1, F157.2, F157.3	8.3
M211, M212, M214, M218, M221, M222, M224, M228, M231, M232, M234, M238	2nd M function code signal	○	○	F193.0, F193.1, F193.2, F193.3, F193.4, F193.5, F193.6, F193.7, F194.0, F194.1, F194.2, F194.3	
M311, M312, M314, M318, M321, M322, M324, M328, M331, M332, M334, M338	3rd M function code signal	○	○	F194.4, F194.5, F194.6, F194.7, F195.0, F195.1, F195.2, F195.3, F195.4, F195.5, F195.6, F195.7	
MCFNA, MCFNB	Power line switch completion signal	○	○	G230.3, G234.3	Manual for serial spindle
MD1O, MD2O, MD4O	Mode select signal (software operator's panel signal)	○	○	F174.0, F174.1, F174.2	12.1.10
MINP	External program input start signal	○		G117.0	13.7
			○	G120.0	
MIRX, MIRY, MIR4	Mirror image signal		○	G127.0, G127.1, G127.7	1.2.5
MIX, MIZ		○		G120.0, G127.1	
MIX1, MIX2, MIX3, MIX4	Composite control start signals	●		G1437.4, G1437.5, G1437.7, G1437.6,	1.7
MF2, MF3	2nd, 3rd M function strobe signal	○	○	F157.4, F157.5	8.3
MFIN	Miscellaneous function completion signal	○	○	G115.0	8.4
MFIN2, MFIN3	2nd, 3rd M function completion signal	○	○	G134.4, G134.5	
MMI1, MMI2	Mirror image check signal	○		F158.0, F158.1	1.2.5
MMI1, MMI2, MMI4			○	F158.0, F158.1, F158.3,	
MP1, MP2	Manual handle heed amount selection signal	○		G117.0, G118.0	3.2
			○	G120.0, G120.1	
MP1O, MP2O	Select of magnification of manual pulse generator (software operator's panel signal)	○	○	F175.6, F175.7	12.1.10
MRDYA, MRDYB	Machine ready signal	○	○	G229.7, G233.7	Manual for serial spindle
MVX, MVY, MVZ, MV4	Axis moving signal		○	F184.4 to F184.7	1.2.4
MVX, MVZ, MV3, MV4		○			
NOWT	No–wait signal	●		G133.1	8.5

Symbol	Signal name	T	M	Address	Ref. item
NOZAGC	Angular axis control–related Z–axis compensation movement signal	☆		G133.6	1.8
			☆	G237.5	
NRROA, NRROB	Short–distant movement command while changing the orientation stop position signal	○	○	G231.2, G235.2	Manual for serial spindle
OFN0, OFN1, OFN2, OFN3, OFN4	Tool offset number select signal	○		G132.0, G132.1, G132.2, G132.3, G132.4	14.4.2
OP	Automatic operation signal	○	○	F148.7	5.1
ORARA, ORARB	Spindle orientation completion signal	○	○	F281.7, F285.7	Manual for serial spindle
ORCMA, ORCMB	Orientation command signal	○	○	G229.6, G233.6	
OV1O, OV2O, OV4O, OV8O	Feedrate override signal (software operator's panel signal)	○	○	F176.0, F176.1, F176.2, F176.3	12.1.10
OVC	Override cancel signal	○	○	G126.4	7.1.6.4
OVCE	PMC axis override cancel signal	○	○	G147.5	15.1
OUT0 to OUT7	Software operator's panel general–purpose switch signal (software operator's panel signal)	○	○	F171.0 to F171.7	12.1.10
PC2SLC	2nd position coder selection signal	○		G146.7	9.10, 9.11
PCFNA, PCFNB	Output switching completion signal	○	○	F282.3, F286.3	Manual for serial spindle
PCHPA, PCHPB	Output switching signal	○	○	F282.2, F286.2	
PECK2	Small–diameter peck drilling cycle execution in progress signal		○	F180.7	11.8
PK1M, PK2M, PK3M, PK4M, PK7M	Synchronization control start signals (tool post 1)	●		G238.0, G238.1, G238.2, G238.3, G238.4	1.7
PK1S, PK2S, PK3S, PK4S	Synchronization control start signals (tool post 2)	●		G1438.0, G1438.1, G1438.2, G1438.3	
PKESS1	Parking signal for 1st spindle	○	○	G138.6	9.18
PKESS2	Parking signal for 2nd spindle	○	○	G138.7	
PLCRVON	Position coder return control signal	○	○	G105.7	9.17
PN1, PN2, PN4, PN8	Workpiece number search signal	○	○	G122.4, G122.5, G122.6, G122.7	15.3
PRC	Position record signal	○		G103.6	14.4.1
PRGDPL	Program screen display signal	○	○	F180.1	15.5
PRTSF	Required parts count reached signal	○	○	F164.7	12.1.7
PSW01 to PSW10	Position switch signal	○	○	F190, F191.0, F191.1	1.2.8
PSYN	Polygon synchronization under way signal	○		F160.7	6.9.1
PX0 to PX7, PZ0 to PZ7	Position signal output signal	○		F165.0 to F165.7, F167.0 to F167.7	1.9

Symbol	Signal name	T	M	Address	Ref. item
R01I to R12I, R01I2 to R12I2, R01I3 to R12I3	Motor speed command spindle input signal	○	○	G124.0 to G125.3, G106.0 to G107.3, G108.0 to G109.3	9.10, 15.4
R01O to R12O	S12–bit code signal	○	○	F172.0 to F173.3	9.3, 15.4
RCHA, RCHB	Power line status check signal	○	○	G230.7, G234.7	Manual for serial spindle
RGSPM, RGSPP	Spindle rotation direction signal		○	F165.1, F165.0	9.11
RGTPN	Rigid tapping signal	○	○	G123.1	
RLSOT	Stroke check release signal		○	G129.7	2.3.2
RMTDI0 to RMTDI7	Input signals for remote buffer	○	○	G239.0 to G239.7	5.11
RMTDO0 to RMTDO7	Output signals for remote buffer	○		F289.0 to F289.7	
			○	F290.0 to F290.7	
ROTAA, ROTAB	Spindle rotation direction command signal while changing the orientation stop position	○	○	G231.1, G235.1	Manual for serial spindle
ROV1, ROV2	Rapid traverse override signal	○	○	G116.7, G117.7	7.1.6.1
ROV1D, ROV2D, ROV3D	Rapid traverse override B signal	○		G116.4, G116.5, G116.6	7.1.6.3
ROV1E, ROV2E	PMC axis rapid traverse override signal	○	○	G146.0, G146.1	15.1
ROV1O, ROV2O	Rapid traverse override signal (software operator's panel signal)	○	○	F175.4, F175.5	12.1.10
RPALM	Read/punch alarm signal	○	○	F180.3	13.5
RPBSY	Read/punch busy signal	○	○	F180.2	
RRW	Reset and rewind signal	○	○	G104.6	5.2
RSLA, RSLB	Output switch request signal	○	○	G230.6, G234.6	Manual for serial spindle
RSMAX	Serial spindle synchronization polygon signal	●		F189.7	9.14
RST	Resetting signal	○	○	F149.1	5.2
RT	Manual rapid traverse select signal	○	○	G121.6	3.1
RTE	Manual rapid traverse selection signal (PMC axis control)	○	○	G147.6	15.1
RTO	Jog rapid traverse select signal (software operator's panel signal)	○	○	F178.6	12.1.10
RTNT	Tapping return start signal		○	G123.6	9.12
RTPT	Tapping return completion signal		○	F192.6	
RWD	Rewinding signal	○	○	F164.6	5.2

Symbol	Signal name	T	M	Address	Ref. item
S11, S12, S14, S18, S21, S22, S24, S28	Spindle–speed code signals	○	○	F152.0, F152.1, F152.2, F152.3, F152.4, F152.5, F152.6, F152.7	8.1, 9.3
S11 to S58	Spindle–speed function code signal (BCD output)	○	○	F185.0 to F187.3	9.11
SA	Servo ready signal	○	○	F148.6	2.2
SAR	Spindle speed arrival signal	○	○	G120.4	9.3
SARA, SARB	Speed reached signal	○	○	F281.3, F285.3	Manual for serial spindle
SBK	Single block signal	○	○	G116.1	5.3.3
SBKO	Single block signal (software operator's panel signal)	○	○	F176.5	12.1.10
SBRT	Spindle synchronous polygon code signal	●		G146.6	9.14
SCLP	Spindle clamp signal	○		F164.0	9.8
SDTA, SDTB	Speed detection signal	○	○	F281.2, F285.2	Manual for serial spindle
SF	Spindle–speed strobe signals	○	○	F150.2	8.1, 9.3, 9.11
SFIN	Spindle function completion signal	○	○	G115.2	8.4
SFRA, SFRB	Spindle CW command signal	○	○	G229.5, G233.5	Manual for serial spindle
SGN, SGN2, SGN3	Spindle motor command polarity select signal	○	○	G125.5, G107.5, G109.5	9.10, 15.4
SHA00 to SHA11	1st spindle orientation external stop position command signal	○	○	G110.0 to G110.7, G111.0 to G111.3	9.15
SHB00 to SHB11	2nd spindle orientation external stop position command signal	○	○	G112.0 to G112.7, G113.0 to G113.3	
SIND, SIND2, SIND3	Spindle motor speed command signal	○	○	G125.7, G107.7, G109.7	9.10, 15.4
SKIP	Skip signal	○	○	X008.7	11.8, 14.3.1, 14.3.3
SKIP2, SKIP3, SKIP4	Skip signal	☆		X008.2, X008.3, X008.4	14.3.3
SLHZ0, SLHZ1	Manual handle feed axis selection signal for Z axis		○	G133.0, G133.1	3.2
SLPCA, SLPCB	Spindle feedback select signal;	●		G1333.2, G1333.3	9.4
SLSPA, SLSPB	Spindle command select signal	●		G133.2, G133.3	
SMZ	Error detect signal	○		G126.6	7.2.5.3

Symbol	Signal name	T	M	Address	Ref. item
SOR	Spindle orientation signal	○	○	G120.5	9.3
SPA, SPB, SPC, SPD	Spindle speed override signal	○		G103.3, G103.4, G103.5, G103.2	
SPA, SPB, SPC			○	G103.3, G103.4, G103.5	
SPAL	Spindle fluctuation detection alarm signal	○		F154.0	9.6
SPDS1, SPDS2, SPDS3, SPDS4	Signals output according to the speed or travel along an axis		○	F189.0, F189.1, F189.2, F189.3	7.2.5.4
SPL	Feed hold lamp signal	○	○	F148.4	5.1
SPO	Feed hold signal (software operator's panel signal)	○	○	F178.7	12.1.10
SPPHS	Spindle phase synchronous control signal	○	○	G146.3	9.13
SPSLA, SPSLB	Spindle select signal	○	○	G230.2, G234.2	Manual for serial spindle
SPSTP	Spindle stop complete signal	○		G123.6	9.8
SPSYC	Spindle synchronous control signal	○	○	G146.2	9.13
SOCNA, SOCNB	Soft start /stop cancel signal	○	○	G230.4, G234.4	Manual for serial spindle
SRN	Program restart signal	○	○	G103.0	5.7
SRNMV	Program restart under way signal	○	○	F188.4	
SRVA, SRVB	Spindle CCW command signal	○	○	G229.4, G233.4	Manual for serial spindle
SSIN, SSIN2, SSIN3	Spindle motor command polarity select signal	○	○	G125.6, G107.6, G109.6	9.10, 15.4
SSTA, SSTB	Speed zero detection signal	○	○	F281.1, F285.1	Manual for serial spindle
ST	Cycle start signal	○	○	G120.2	5.1
STL	Cycle start lamp signal	○	○	F148.5	5.1
STLK	Start lock signal	○		G120.1	2.5
STRD	Input and run simultaneous mode select signal		○	G140.5	13.6
STWD	Output and run simultaneous mode select signal		○	G140.6	
SUCLP	Spindle unclamp signal	○		F164.1	9.8
SVFX, SVFZ, SVF3, SVF4	Servo off signal	○		G105.0, G105.1, G105.2, G105.3	1.2.7
SVFX, SVFY, SVFZ, SVF4			○		

Symbol	Signal name	T	M	Address	Ref. item
SWS1, SWS2, SWS3	Spindle selection signal	○		G145.0, G145.1, G145.2	9.10, 9.11
SYCAL	Spindle synchronous control alarm signal	○	○	F178.4	9.13, 9.18
SYN1M, SYN2M, SYN3M, SYN4M, SYN7M	Synchronization control start signals (tool post 1)	●		G237.0, G237.1, G237.2, G237.3, G237.4	1.7
SYN1OM, SYN2OM, SYN3OM, SYN4OM, SYN7OM	Axis recomposition signals (tool post 1)	●		F189.0, F189.1, F189.2, F189.3, F189.4	
SYN1OS, SYN2OS, SYN3OS, SYN4OS	Axis recomposition signals (tool post 2)	●		F1389.0, F1389.1, F1389.2, F1389.3	
SYN1S, SYN2S, SYN3S, SYN4S	Synchronization control start signals (tool post 2)	●		G1437.0, G1437.1, G1437.2, G1437.3	
SYNAL	Servo axis synchronization alarm signal		○	F192.7	1.6
SYNCX, SYNCZ, SYNC3, SYNC4	Signals to select the slave axis for simple synchronous control	○		G237.0 to G237.3	
SYNC4			○	G237.3	
SYNCJ	Signal for selecting the manual feed axis for simple synchronous control		○	G133.6	
T11 to T28	Tool function code signals	○		F153.0 to F153.7	8.1
T11 to T48			○	F153.0 to F156.7	
TAL	Tool post interference alarm signal	●		F180.7	2.3.4
TF	Tool function strobe signals	○	○	F150.3	8.1
TFIN	Tool function completion signal	○	○	G115.3	8.4
THRD	Thread cutting signal	○		F188.3	6.4.1
TL01, TL02, TL04, TL08, TL16, TL32, TL64	Tool group number select signal	○	○	G139.0, G139.1, G139.2, G139.3, G139.4, G139.5, G139.6	10.3
TLCH	Tool change signal	○	○	F188.0	10.3
TLCHB	Remaining tool life expired signal		○	F192.2	
TLCHE	All tools' life expired signal		○	F192.1	
TLCHI	Individual tool change signal		○	F192.0	
TLMA, TLMB	Torque limit signal	○	○	F281.6, F285.6	Manual for serial spindle
TLMHA, TLMHB	High–speed torque limit signal	○	○	G229.1, G233.1	
TLMLA, TLMLB	Low speed torque limit signal	○	○	G229.0, G233.0	
TLNW	New tool select signal	○	○	F188.1	10.3
TLRST	Tool change reset signal	○	○	G139.7	
TLRSTI	Individual tool change reset signal		○	G140.4	
TLSKP	Tool skip signal	○	○	G140.0	
TNFS	Tool compensation number automatic selection disable signal	○		G132.5	14.4.2

Symbol	Signal name	T	M	Address	Ref. item
TORQ1, TORQ2, TORQ3, TORQ4, TORQ7, TORQ8	Torque limit reached signal	○		F170.0, F170.1, F170.2, F170.3, F170.4, F170.5	14.3.4
TRT2, TRT2PC	Tool post selection signal	●		X018.3, G133.0	2.7
UI000 to UI015	Custom macro input signal	○	○	G130.0 to G131.7	11.6.1
UINT	Interrupt signal for custom macro	○	○	G140.3	11.6.2
UO000 to UO015, UO100 to UO131	Custom macro output signal	○	○	F162.0 to F163.7, F196.0 to F199.7	11.6.1
WNANS	Window completion signal	○	○	F250.0	15.7
WNERR	Window error signal	○	○	F250.1	
WNRQ	Window request signal	○	○	G200.0	
WNWRT	Window rewrite signal	○	○	G200.1	
WOQSM	Workpiece coordinate system shift value write mode select signal	○		G132.6	14.4.2
WOSET	Workpiece coordinate system shift value write signal	○		G133.7	
XAE, YAE, ZAE	Measuring position reached signals		○	X008.0, X008.1, X008.2	14.2
XAE, ZAE		○		X008.0, X008.1	
ZNG	Cancel–the–Z–axis command signal		○	G103.6	1.4.4
ZP2X, ZP2Z, ZP23, ZP24	Second reference position return completion signals	○		F161.0, F161.1, F161.2, F161.3	4.5
ZP2X, ZP2Y, ZP2Z, ZP24			○		
ZP3	Cs contour control axis reference position return completion signal	○		F148.2	9.9
	Spindle orientation completion signal	○			9.8
ZP3X, ZP3Y, ZP3Z, ZP34	Third reference position return completion signals		○	F169.0, F169.1, F169.2, F169.3	4.5
ZP3X, ZP3Z, ZP33, ZP34		○			
ZP4	Cs contour control axis reference position return completion signal		○	F148.3	9.9
ZP4X, ZP4Y, ZP4Z, ZP44	Fourth reference position return completion signals		○	F169.4, F169.5, F169.6, F169.7	4.5
ZP4X, ZP4Z, ZP43, ZP44		○			
ZPX, ZPY, ZPZ, ZP4	Reference position return completion signal		○	F148.0, F148.1, F148.2, F148.3	4.1
ZPX, ZPZ, ZP3, ZP4		○			
ZRFX, ZRFY, ZRFZ, ZRF4	Reference position establishment signal		○	F168.0, F168.1, F168.2, F168.3	4.1
ZRFX, ZRFZ, ZRF3, ZRF4		○			
ZRN	Mode selection signal	○	○	G120.7	2.6
	Manual reference position return selection signal				4.1
ZRNO	Mode select signal (software operator's panel signal)	○	○	F174.3	12.1.10

A.2.3 List of Signals (In Order of Addresses)

Meaning of symbols in the below table
○: Available
●: 0–TTC only
☆: Available for 0–GCC or 0–GSC only
Blank : Unavailable

Address	Signal name	Symbol	T	M	Ref. item
F148.0, F148.1, F148.2, F148.3	Reference position return completion signal	ZPX, ZPZ, ZP3, ZP4	○		4.1
		ZPX, ZPY, ZPZ, ZP4		○	
F148.2	Spindle orientation completion signal	ZP3	○		9.8
F148.2	Cs contour control axis reference position return completion signal	ZP3	○		9.9
F148.3		ZP4		○	
F148.4	Feed hold lamp signal	SPL	○	○	5.1
F184.4 to F184.7	Axis moving signal	MVX, MVY, MVZ, MV4		○	1.2.4
		MVX, MVZ, MV3, MV4	○		
F148.5	Cycle start lamp signal	STL	○	○	5.1
F148.7	Automatic operation signal	OP	○	○	
F148.6	Servo ready signal	SA	○	○	2.2
F149.0	Alarm signal	AL	○	○	2.4
F149.1	Resetting signal	RST	○	○	5.2
F149.2	Battery alarm signal	BAL	○	○	2.4
F149.3	Distribution end signals	DEN	○	○	8.1
F149.4	Spindle enable signal	ENB	○	○	9.3, 9.10, 9.11
F149.4, F164.2, F164.3		ENB2, ENB3	○		
F149.5	Tapping signal	TAP	○	○	11.7
F149.6	Passing point signal	DEN2	○		8.1
F149.7	CNC ready signal	MA	○	○	2.2
F150.0	Miscellaneous function strobe signal	MF	○	○	8.1
F150.1	External operation signal	EF		○	11.9
F150.1	External operation signal for high–speed interface	EF		○	8.4
F150.2	Spindle function strobe signal	SF	○	○	8.1, 9.3, 9.11
F150.3	Tool function strobe signals	TF	○	○	8.1
F150.5	Manual data input start signal	DST	○	○	5.1
F150.7	Second auxiliary function strobe signals	BF	○		8.1
F150.7, F150.6		BF1, BF2		○	

Address	Signal name	Symbol	T	M	Ref. item
F151.0, F151.1, F151.2, F151.3, F151.4, F151.5, F151.6, F151.7, F157.0, F157.1, F157.2, F157.3	Miscellaneous function code signal	M11, M12, M14, M18, M21, M22, M24, M28, M31, M32, M34, M38	○	○	8.1
F152.0, F152.1, F152.2, F152.3, F152.4, F152.5, F152.6, F152.7	Spindle–speed code signals	S11, S12, S14, S18, S21, S22, S24, S28	○	○	8.1, 9.3
F153.0 to F153.7	Tool function code signals	T11 to T28	○		8.1
F153.0 to F156.7		T11 to T48		○	
F154.0	Spindle fluctuation detection alarm signal	SPAL	○		9.6
F154.7, F154.6, F154.5, F154.4	Decode M signals	M00, M01, M02, M30		○	8.1
F155.0 to F154.3	Second auxiliary function code signals	B11 to B38		○	8.1
F156.0 to F156.5	Absolute pulse coder battery alarm signal	BAL1 to BAL4,BAL7,BAL8	○		2.4
F157.4, F157.5	2nd, 3rd M function strobe signal	MF2, MF3	○	○	8.3
F158.0 to F159.7	Actual spindle speed signal	AR0 to AR15	○		9.7
F158.0, F158.1	Mirror image check signal	MMI1, MMI2	○		1.2.5
F158.0, F158.1, F158.3,		MMI1, MMI2, MMI4		○	
F159.0 to F159.5	Absolute pulse coder battery alarm signal	BAL1 to BAL4,BAL7,BAL8		○	2.4
F160.0	Read completion signal (for external data input)	EREND	○	○	15.2
F160.1	Search completion signal (for external data input)	ESEND	○	○	
F160.6	Waiting signal	WATO	●		8.5
F160.7	Polygon synchronization under way signal	PSYN	○		6.9.1
F161.0, F161.1, F161.2, F161.3	Second reference position return completion signals	ZP2X, ZP2Z, ZP23, ZP24	○		4.5
		ZP2X, ZP2Y, ZP2Z, ZP24		○	
F161.4	Canned cycle start signal	FXST		○	2.8
F162.0 to F163.7, F196.0 to F199.7	Custom macro output signal	UO000 to UO015, UO100 to UO131	○	○	11.6.1
F164.0	Spindle clamp signal	SCLP	○		9.8
F164.1	Spindle unclamp signal	SUCLP	○		
F164.5	Drwing signal	CKGRP	○	○	12.1.8
F164.6	Rewinding signal	RWD	○	○	5.2
F164.7	Required parts count reached signal	PRTSF	○	○	12.1.7
F165.0 to F165.7, F167.0 to F167.7	Position signal output signal	PX0 to PX7, PZ0 to PZ7	○		1.9
F165.1, F165.0	Spindle rotation direction signal	RGSPM, RGSPP		○	9.11

Address	Signal name	Symbol	T	M	Ref. item
F165.3	Grinding wheel diameter automatic compensation signal	GWLF		☆	14.4.4
F168.0, F168.1, F168.2, F168.3	Reference position establishment signal	ZRFX, ZRFZ, ZRF3, ZRF4	○		4.1
		ZRFX, ZRFY, ZRFZ, ZRF4		○	
F169.0, F169.1, F169.2, F169.3	Third reference position return completion signals	ZP3X, ZP3Z, ZP33, ZP34	○		4.5
		ZP3X, ZP3Y, ZP3Z, ZP34		○	
F169.4, F169.5, F169.6, F169.7	Fourth reference position return completion signals	ZP4X, ZP4Z, ZP43, ZP44	○		
		ZP4X, ZP4Y, ZP4Z, ZP44		○	
F170.0, F170.1, F170.2, F170.3, F170.4, F170.5	Torque limit reached signal	TORQ1, TORQ2, TORQ3, TORQ4, TORQ7, TORQ8	○		14.3.4
F171.0 to F171.7	Software operator's panel general–purpose switch signal (software operator's panel signal)	OUT0 to OUT7	○	○	12.1.10
F172.0 to F173.3	S12–bit code signal	R01O to R12O	○	○	9.3, 15.4
F174.0, F174.1, F174.2	Mode select signal (software operator's panel signal)	MD1O, MD2O, MD4O	○	○	12.1.10
F174.3	Mode select signal (software operator's panel signal)	ZRNO	○	○	
F174.4, F174.5, F174.6, F174.7	Manual pulse generator feed axis select signal (software operator's panel signal)	HXO, HYO, HZO, HZ4		○	
		HXO, HZO, H3O, H4O	○		
F175.0, F175.1, F175.2, F175.3	Jog feedrate override signal (software operator's panel signal)	JV1O, JV2O, JV4O, JV8O	○	○	
F175.4, F175.5	Rapid traverse override signal (software operator's panel signal)	ROV1O, ROV2O	○	○	
F175.6, F175.7	Select of magnification of manual pulse generator (software operator's panel signal)	MP1O, MP2O	○	○	
F176.0, F176.1, F176.2, F176.3	Feedrate override signal (software operator's panel signal)	OV1O, OV2O, OV4O, OV8O	○	○	
F176.4	Optional block skip signal (software operator's panel signal)	BDTO	○	○	
F176.5	Single block signal (software operator's panel signal)	SBKO	○	○	
F176.6	Machine lock signal (software operator's panel signal)	MLKO	○	○	
F176.7	Dry run signal (software operator's panel signal)	DRNO	○	○	
F177.0, F177.1, F177.2, F177.3, F177.4, F177.5, F177.6, F177.7	Jog feed axis select signal (software operator's panel signal)	+XO, –XO, +ZO, –ZO, +YO, –YO, +4O, –4O	○		
		+XO, –XO, +YO, –YO, +ZO, –ZO, +4O, –4O		○	
F178.1	Spindle contour control change completion signal	FSCSL		○	9.9

Address	Signal name	Symbol	T	M	Ref. item
F178.2	Spindle synchronous speed control completion signal	FSPSY	○	○	9.13
F178.3	Spindle phase synchronous control completion signal	FSPPH	○	○	
F178.4	Spindle synchronous control alarm signal	SYCAL	○	○	9.13, 9.18
F178.5	Program protect signal (software operator's panel signal)	KEYO	○	○	12.1.10
F178.6	Jog rapid traverse select signal (software operator's panel signal)	RTO	○	○	12.1.10
F178.7	Feed hold signal (software operator's panel signal)	SPO	○	○	
F180.0	Key input disable signal	INHKY	○	○	15.5
F180.1	Program screen display signal	PRGDPL	○	○	
F180.2	Read/punch busy signal	RPBSY	○	○	13.5
F180.3	Read/punch alarm signal	RPALM	○	○	
F180.4	Background editing signal	BGEACT	○	○	
F180.5	Spindle command signal;	COSP	●		9.4
F180.6	Tool post interference check signal	BOFF	●		2.3.4
F180.7	Tool post interference alarm signal	TAL	●		
F180.7	Small–diameter peck drilling cycle execution in progress signal	PECK2		○	11.8
F184.0 to F184.3	In–position signals	INP1 to INP4	○	○	7.2.5.1
F185.0 to F187.3	Spindle–speed function code signal (BCD output)	S11 to S58	○	○	9.11
F188.0	Tool change signal	TLCH	○	○	10.3
F188.1	New tool select signal	TLNW	○	○	
F188.2	B axis unclamp signal	BUCLP		○	11.12
F188.3	B axis clamp signal	BCLP		○	
F188.3	Thread cutting signal	THRD	○		6.4.1
F188.4	Program restart under way signal	SRNMV	○	○	5.7
F188.5	Override 0% signal (PMC axis control)	EOV0	○	○	15.1
F188.6	Cutting feed signal	CUT	○	○	2.8
F188.7	Control axis selection status signal (PMC axis control)	*EAXSL	○	○	15.1
F189.0, F189.1, F189.2, F189.3	Signals output according to the speed or travel along an axis	SPDS1, SPDS2, SPDS3, SPDS4		○	7.2.5.4
F189.0, F189.1, F189.2, F189.3, F189.4	Axis recomposition signals (tool post 1)	SYN1OM, SYN2OM, SYN3OM, SYN4OM, SYN7OM	●		1.7
F189.7	Serial spindle synchronization polygon signal	RSMAX	●		9.14
F190, F191.0, F191.1	Position switch signal	PSW01 to PSW10	○	○	1.2.8

Address	Signal name	Symbol	T	M	Ref. item
F192.0	Individual tool change signal	TLCHI		○	10.3
F192.1	All tools' life expired signal	TLCHE		○	
F192.2	Remaining tool life expired signal	TLCHB		○	
F192.6	Tapping return completion signal	RTPT		○	9.12
F192.7	Servo axis synchronization alarm signal	SYNAL		○	1.6
F193.0, F193.1, F193.2, F193.3, F193.4, F193.5, F193.6, F193.7, F194.0, F194.1, F194.2, F194.3	2nd M function code signal	M211, M212, M214, M218, M221, M222, M224, M228, M231, M232, M234, M238	○	○	8.3
F194.4, F194.5, F194.6, F194.7, F195.0, F195.1, F195.2, F195.3, F195.4, F195.5, F195.6, F195.7	3rd M function code signal	M311, M312, M314, M318, M321, M322, M324, M328, M331, M332, M334, M338	○	○	
F211.0, G219.0	Block stop disable signal (PMC axis control)	EMSBKA, EMSBKB	○	○	15.1
F250.0	Window completion signal	WNANS	○	○	15.7
F250.1	Window error signal	WNERR	○	○	
F270.0, F273.0	In–position signal (PMC axis control)	EINPA, EINPB	○	○	15.1
F270.1, F273.1	Following zero checking signal (PMC axis control)	ECKZA, ECZKB	○	○	
F270.2, F273.2	Alarm signal (PMC axis control)	EIALA, EIALB	○	○	
F270.3, F273.3	Auxiliary function executing signal (PMC axis control)	EDENA, EDENB	○	○	
F270.4, F273.4	Axis moving signal (PMC axis control)	EGENA, EGENB	○	○	
F270.5, F273.5	Positive–direction overtravel signal (PMC axis control)	EOTPA, EOTPB	○	○	
F270.6, F273.6	Negative–direction overtravel signal (PMC axis control)	EOTNA, EOTNB	○	○	
F270.7, F273.7	Axis control command read completion signal (PMC axis control)	EBSYA, EBSYB	○	○	
F271.0, F274.0	Auxiliary function strobe signal	EMFA, EMFB	○	○	
F272.0 to F272.7, F275.0 to F275.7	Auxiliary function code signal (PMC axis control)	EM11A to EM28A, EM11B to EM28B	○	○	
F276.0 to F279.7	Second auxiliary function code signals	B0 to B31	○		8.1
F281.0, F285.0	Spindle alarm signal	ALMA, ALMB		○	Manual for serial spindle
F281.1, F285.1	Speed zero detection signal	SSTA, SSTB	○	○	
F281.2, F285.2	Speed detection signal	SDTA, SDTB	○	○	
F281.3, F285.3	Speed reached signal	SARA, SARB	○	○	
F281.3, F285.3	Speed reached signal	SARA, SARB	○	○	
F281.4, F285.4, F281.5, F285.5	Load detection signal	LDT1A, LDT1B, LDT2A, LDT2B	○	○	
F281.6, F285.6	Torque limit signal	TLMA, TLMB	○	○	

Address	Signal name	Symbol	T	M	Ref. item
F281.7, F285.7	Spindle orientation completion signal	ORARA, ORARB	○	○	Manual for serial spindle
F282.0, F286.0	Power line switch signal	CHPA, CHPB	○	○	
F282.1, F286.1	Spindle switch completion signal	CFINA, CFINB	○	○	
F282.2, F286.2	Output switching signal	PCHPA, PCHPB	○	○	
F282.3, F286.3	Output switching completion signal	PCFNA, PCFNB	○	○	
F289.0 to F289.7	Output signals for remote buffer	RMTDO0 to RMTDO7	○		5.11
F290.0 to F290.7				○	
F1389.0, F1389.1, F1389.2, F1389.3	Axis recomposition signals (tool post 2)	SYN1OS, SYN2OS, SYN3OS, SYN4OS	●		1.7
G100.0 to G101.7	Data signal (for external data input)	ED0 to ED15	○	○	15.2
G102.0 to G102.6	Address signal (for external data input)	EA0 to EA6	○	○	
G102.7	Read signal (for external data input)	ESTB	○	○	
G103.0	Program restart signal	SRN	○	○	5.7
G103.3, G103.4, G103.5, G103.2	Spindle speed override signal	SPA, SPB, SPC, SPD	○		9.3
G103.3, G103.4, G103.5	Spindle speed override signal	SPA, SPB, SPC		○	
G103.6	Cancel–the–Z–axis command signal	ZNG		○	1.4.4
G103.6	Position record signal	PRC	○		14.4.1
G103.7	Auxiliary function lock signal	AFL	○	○	8.2
G104.0 to G104.3	Jog feedrate override signal	JOV1 to JOV8		○	3.1
G104.4	Spindle simple synchronous control signal	ESRSYC	○	○	9.18
G104.5	Follow–up signal	*FLWU	○	○	1.2.6
G104.6	Reset and rewind signal	RRW	○	○	5.2
G105.0, G105.1, G105.2, G105.3	Servo off signal	SVFX, SVFZ, SVF3, SVF4	○		1.2.7
		SVFX, SVFY, SVFZ, SVF4		○	
G105.6	Position coder feedback direction selection signal	KILPLUS	○	○	9.17
G105.7	Position coder return control signal	PLCRVON	○	○	
G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G109.5, G108.0, G107.5, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7	Key signal	K0, S, K1, H, K2, F, K3, R, K4, X, K5, Y, K6, Z, K7, 0, K8, N, K9, G, KALARM, KALTER, KB, K4, D, KCAN, KCUS↑, KCUS↓, KDELET, KDGNOS, KEOB, / , .		○	15.5
G106.0, G106.1, G106.2, G106.3, G106.4, G106.5, G106.6, G106.7, G107.0, G107.1, G107.5, G109.5, G108.0, G110.7, G110.1, G110.0, G108.2, G109.4, G107.7		K0, S, K1, U, K2, W, K3, R, K4, X, K5, Z, K6, F, K7, 0, K8, N, K9, G, KA, C, KALARM, KALTER, KCAN, KCUS↑, KCUS÷, KDELET, KDGNOS, KEOB	○		

Address	Signal name	Symbol	T	M	Ref. item
G108.3, G108.1, G107.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G142.7, G110.4	Key signal	KINPUT, KINSRT, KM, –, KP, Q, KPAGE ↑, KPAGE ↓, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXZ, KXZ	○		15.5
G110.0 to G110.7, G111.0 to G111.3	1st spindle orientation external stop position command signal	SHA00 to SHA11	○	○	9.15
G112.0 to G112.7, G113.0 to G113.3	2nd spindle orientation external stop position command signal	SHB00 to SHB11	○	○	
G113.7, G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.4, G113.6, G113.5, G107.6, G108.3, G108.1, G107.2, G109.2, G107.4, G110.3, G110.2, G109.3, G109.0, G109.1, G111.7, G112.6, G112.5, G112.4, G112.3, G112.2, G112.7, G112.1, G108.4, G107.3, G110.4, G110.6, G110.5	Key signal	KF4TH, KFF, KFF%, KFGE, KFI,, KFJA, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFXU, KFXW, KFYV, KH, J, K, KINPUT, KINSRT, KM, –, KOFSET, KP, Q, L, KPAGE↑, KPAGE↓, KPARAM, KPOS, KPROGRM, KRESET, KS1, KS2, KS3, KS4, KS5, KSL, KSR, KSTART, KT,., KXY, KYZ, KZX		○	15.5
G114.3, G114.2, G113.2, G114.0, G114.1, G114.7, G114.4, G113.1, G113.0, G113.3, G114.5, G114.6, G113.6, G113.7, G113.4, G113.5, G107.6		KFF, KFFE, KFGB, KFI,, KFK%, KFL+, KFM., KFNK, KFOJ, KFRC, KFS=, KFT*, KFUV, KFWH, KFXY, KFZJ, KH, I, K	○		
G115.0	Miscellaneous function completion signal	MFIN	○	○	8.4
G115.1	External operation function completion signal	EFIN		○	
G115.2	Spindle function completion signal	SFIN	○	○	
G115.3	Tool function completion signal	TFIN	○	○	
G115.7	2nd auxiliary function completion signal	BFIN	○		
G115.7, G115.6		BFIN1, BFIN2		○	
G116.0, G141.0 to G141.7	Optional block skip signals	BDT1, BDT2 to BDT9	○	○	5.5
G116.1	Single block signal	SBK	○	○	5.3.3
G116.2, G116.3, G117.2, G117.3, G118.2, G118.3, G119.2, G119.3	Feed axis and direction select signal	+X, –X, +Z, –Z, +3, –3, +4, –4	○		3.1
		+X, –X, +Y, –Y, +Z, –Z, +4, –4		○	
G116.4, G116.5, G116.6	Rapid traverse override B signal	ROV1D, ROV2D, ROV3D	○		7.1.6.3
G116.4, G116.5, G116.6, G117.6	Feedrate override signal	*AOV16, *AOV32, *AOV64, *AOV128		○	7.1.6.2
G116.7, G117.7	Rapid traverse override signal	ROV1, ROV2	○	○	7.1.6.1

Address	Signal name	Symbol	T	M	Ref. item
G116.7, G117.7, G118.7, G119.7	Manual handle feed axis selection signal	HX, HZ, H3, H4	○		3.2
		HX, HY, HZ, H4		○	
G117.0	External program input start signal	MINP	○		13.7
G117.0, G118.0	Manual handle heed amount selection signal	MP1, MP2	○		3.2
G117.0	Interlock signal	*ILK		○	2.5
G117.1	All–axis machine lock signal	MLK	○	○	5.3.1
G118.2, G118.3	Gear selection signal	GR1, GR2	○		9.3, 9.10, 9.11
G118.2, G118.3	Gear selection signal	GR1, GR2	○		9.3, 9.5, 9.8, 9.9
G118.7	Dry run signal	DRN	○	○	5.3.2
G119.0	Display and setting	*LDSP	○	○	12.1
G120.0	External program input start signal	MINP		○	13.7
G120.0, G120.1	Manual handle heed amount selection signal	MP1, MP2		○	3.2
G120.0, G127.1	Mirror image signal	MIX, MIZ	○		1.2.5
G120.1	Start lock signal	STLK	○		2.5
G120.2	Cycle start signal	ST	○	○	5.1
G120.3	End signal	FIN	○	○	8.1
G120.4	Spindle speed arrival signal	SAR	○	○	9.3
G120.5	Spindle orientation signal	SOR	○	○	
G120.6	Spindle stop signal	*SSTP	○	○	
G120.7	Mode selection signal	ZRN	○	○	2.6
G120.7	Manual reference position return selection signal	ZRN	○	○	4.1
G121.0, G121.1, G121.2, G121.3	Feedrate override signal	*OV1, *OV2, *OV4, *OV8	○	○	3.1, 7.1.6.2
G121.5	Feed hold signal	*SP	○	○	5.1
G121.6	Manual rapid traverse select signal	RT	○	○	3.1
G121.7	External reset signal	ERS	○	○	5.2
G122.0, G122.1, G122.2	Mode selection signal	MD1, MD2, MD4	○	○	2.6
G122.3	Program protect signal	KEY	○	○	12.2.3
G122.4, G122.5, G122.6, G122.7	Workpiece number search signal	PN1, PN2, PN4, PN8	○	○	15.3
G123.0	C–axis–off signal	COFF	○		1.10
	Spindle contour control change signal				9.9
G123.0	All–axis CRDY OFF alarm ignore signal	IGNVRY		○	2.9
G123.1	Rigid tapping signal	RGTPN	○	○	9.11

Address	Signal name	Symbol	T	M	Ref. item
G124.0 to G125.3, G106.0 to G107.3, G108.0 to G109.3	Motor speed command spindle input signal	R01I to R12I, R01I2 to R12I2, R01I3 to R12I3	○	○	9.10, 15.4
G123.2, G123.3	Gear selection signal	GR1, GR2		○	9.3, 9.5, 9.8, 9.9, 9.10, 9.11
G123.4	Spindle unclamp completion signal	*SUCPF	○		9.8
G123.5	Spindle clamp completion signal	*SCPF	○		
G123.6	Spindle stop complete signal	SPSTP	○		
G123.6	Tapping return start signal	RTNT		○	9.12
G123.7	Spindle contour control change signal	CON		○	9.9
G125.5, G107.5, G109.5	Spindle motor command polarity select signal	SGN, SGN2, SGN3	○	○	9.10, 15.4
G125.6, G107.6, G109.6		SSIN, SSIN2, SSIN3			
G125.7, G107.7, G109.7	Spindle motor speed command signal	SIND, SIND2, SIND3	○	○	9.10, 15.4
G126.0, G126.1, G126.2, G126.3	Manual handle interrupt axis selection signal	HIX, HIY, HIZ, HI4		○	3.3
		HIX, HIZ, HI3, HI4	○		
G126.4	Override cancel signal	OVC	○	○	7.1.6.4
G126.6	Error detect signal	SMZ	○		7.2.5.3
G126.7	Chamfering signal	CDZ	○		11.10
G127.0	All–axis CRDY OFF alarm ignore signal	IGNVRY	○		2.9
G127.0, G127.1, G127.7	Mirror image signal	MIRX, MIRY, MIR4		○	1.2.5
G127.2	Manual absolute signal	*ABSM	○	○	5.4
G127.5	Mode selection signal	DNCI	○	○	2.6
	DNC operation select signal				5.10
G127.6	Renewal disable signal of relative coordinate	DLK	○	○	12.1
G128.0, G128.1, G128.2, G128.3	Interlock signal for each axis	*ITX, *ITY, *ITZ, *IT4		○	2.5
		ITX, ITZ, IT3, IT4	○		
G128.7	Direct operation select signal	DMMC	○	○	15.6
G129.0, G129.1, G129.2, G129.3, G129.4, G129.5	Stroke check external setting signals	+LMX, +LMY, +LMZ, –LMX, –LMY, –LMZ		○	2.3.2
G129.6	Stored stroke check select signal	EXLM2	○	○	
G129.7	Stroke check release signal	RLSOT		○	
G130.0 to G131.7	Custom macro input signal	UI000 to UI015	○	○	11.6.1
G132.0, G132.1, G132.2, G132.3, G132.4	Tool offset number select signal	OFN0, OFN1, OFN2, OFN3, OFN4	○		14.4.2
G133.0, G133.1	Manual handle feed axis selection signal for Z axis	SLHZ0, SLHZ1		○	3.2

Address	Signal name	Symbol	T	M	Ref. item
G132.5	Tool compensation number automatic selection disable signal	TNFS	○		14.4.2
G132.6	Workpiece coordinate system shift value write mode select signal	WOQSM	○		
G132.7	Tool offset write mode select signal	GOQSM	○		
G133.1	No–wait signal	NOWT	●		8.5
G133.2, G133.3	Spindle command select signal	SLSPA, SLSPB	●		9.4
G133.6	Signal for selecting the manual feed axis for simple synchronous control	SYNCJ		○	1.6
G133.6	Angular axis control–related Z–axis compensation movement signal	NOZAGC	☆		1.8
G133.7	Workpiece coordinate system shift value write signal	WOSET	○		14.4.2
G134.0	External key input mode selection signal	ENBKY	○	○	15.5
G134.1	External read start signal	EXRD	○	○	13.5
G134.2	External read/punch stop signal	EXSTP	○	○	
G134.3	External punch start signal	EXWT	○	○	
G134.4, G134.5	2nd, 3rd M function completion signal	MFIN2, MFIN3	○	○	8.4
G138.0, G138.1, G138.3, G138.4	External deceleration signal	*+EDCX, *+EDCZ, *–EDCX , *–EDCZ	○		7.1.8
G138.0, G138.1, G138.2, G138.3, G138.4, G138.5		*+EDCX, *+EDCY, *+EDCZ, *–EDCX, *–EDCY, *–EDCZ		○	
G138.6	Parking signal for 1st spindle	PKESS1	○	○	9.18
G138.7	Parking signal for 2nd spindle	PKESS2	○	○	
G139.0, G139.1, G139.2, G139.3, G139.4, G139.5, G139.6	Tool group number select signal	TL01, TL02, TL04, TL08, TL16, TL32, TL64	○	○	10.3
G139.7	Tool change reset signal	TLRST	○	○	
G140.0	Tool skip signal	TLSKP	○	○	
G140.3	Interrupt signal for custom macro	UINT	○	○	11.6.2
G140.4	Individual tool change reset signal	TLRSTI		○	10.3
G140.4, G140.5, G140.6, G140.7	Feedrate override signal	*AOVR16, *AOVR32, *AOVR64, *AOVR128	○		7.1.6.2
G140.5	Input and run simultaneous mode select signal	STRD		○	13.6
G140.6	Output and run simultaneous mode select signal	STWD		○	
G140.7	F1–digit feed select signal	F1D		○	7.1.5
G142.0, G142.1, G142.2, G142.3, G142.4, G142.5, G142.6, G142.7	Interlock signal for each axis and direction	*+MITX, *+MITY, *+MITZ, *+MIT4, *–MITX, *–MITY, *–MITZ, *–MIT4		○	2.5
G143.6	B axis unclamp completion signal	*BEUCP		○	11.12

Address	Signal name	Symbol	T	M	Ref. item
G143.7	B axis clamp completion signal	*BECLP		○	11.12
G144.0 to G144.5	Control axis selection signal (PMC axis control)	EAX1 to EAX8	○	○	15.1
G145.0, G145.1, G145.2	Spindle selection signal	SWS1, SWS2, SWS3	○		9.10, 9.11
G145.3, G145.4, G145.5	Individual spindle stop signal	*SSTP1, *SSTP2, *SSTP3	○		
G145.6	Gear selection signal for 2nd spindle	GR21	○		
G145.7	Gear selection signal for 3rd spindle	GR31	○		
G146.0, G146.1	PMC axis rapid traverse override signal	ROV1E, ROV2E	○	○	15.1
G146.2	Spindle synchronous control signal	SPSYC	○	○	9.13
G146.3	Spindle phase synchronous control signal	SPPHS	○	○	
G146.6	Spindle synchronous polygon code signal	SBRT	●		9.14
G146.7	2nd position coder selection signal	PC2SLC	○		9.10, 9.11
G147.0, G147.1, G147.2, G147.3	Feedrate override signal (PMC axis control)	*OV1E, *OV2E, *OV4E, *OV8E	○	○	15.1
G147.5	PMC axis override cancel signal	OVCE	○	○	
G147.6	Manual rapid traverse selection signal (PMC axis control)	RTE	○	○	
G147.7	Dry run signal (PMC axis control)	DRNE	○	○	
G152.0, G152.1, G152.2	Gear selection signal	GR1O, GR2O, GR3O		○	9.3, 9.10, 9.11
G152.0, G152.1, G152.2	Gear selection signal	GR1O, GR2O, GR3O		○	9.3, 9.9
G200.0	Window request signal	WNRQ	○	○	15.7
G200.1	Window rewrite signal	WNWRT	○	○	
G210.0, G218.0	Auxiliary function completion signal (PMC axis control)	EFINA, EFINB	○	○	15.1
G210.3, G218.3	Block stop signal (PMC axis control)	ESBKA, ESBKB	○	○	
G210.4, G218.4	Servo off signal (PMC axis control)	ESOFA, ESOFB	○	○	
G210.5, G218.5	Axis control temporary stop signal (PMC axis control)	ESTPA, ESTPB	○	○	
G210.6, G218.6	Reset signal (PMC axis control)	ECLRA, ECLRB	○	○	
G210.7, G218.7	Axis control command read signal (PMC axis control)	EBUFA, EBUFB	○	○	
G211.0 to G211.6, G219.0 to G219.6	Axis control command signal (PMC axis control)	EC0A to EC6A, EC0B to EC6B	○	○	
G212.0 to G213.7, G220.0 to G221.7	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A, EIF0B to EIF15B	○	○	
G214.0 to G217.7, G222.0 to G225.7	Axis control data signal (PMC axis control)	EID0A to EID31A, EID0B to EID31B	○	○	

Address	Signal name	Symbol	T	M	Ref. item
G229.0, G233.0	Low speed torque limit signal	TLMLA, TLMLB	○	○	Manual for serial spindle
G229.1, G233.1	High–speed torque limit signal	TLMHA, TLMHB	○	○	
G229.3, G229.2	Gear selection signal	CTH1A CTH2A	○	○	9.8
G229.3, G229.2	Clutch/gear signal (serial spindle)	CTH1A CTH2A	○	○	9.9
G229.4, G233.4	Spindle CCW command signal	SRVA, SRVB	○	○	Manual for serial spindle
G229.5, G233.5	Spindle CW command signal	SFRA, SFRB	○	○	
G229.6, G233.6	Orientation command signal	ORCMA, ORCMB	○	○	
G229.7, G233.7	Machine ready signal	MRDYA, MRDYB	○	○	
G230.0, G234.0	Alarm reset signal	ARSTA, ARSTB	○	○	
G230.1, G234.1	Emergency stop signal for spindle	*ESPA, *ESPB	○	○	
G230.2, G234.2	Spindle select signal	SPSLA, SPSLB	○	○	
G230.3, G234.3	Power line switch completion signal	MCFNA, MCFNB	○	○	
G230.4, G234.4	Soft start /stop cancel signal	SOCNA, SOCNB	○	○	
G230.6, G234.6	Output switch request signal	RSLA, RSLB	○	○	
G230.7, G234.7	Power line status check signal	RCHA, RCHB	○	○	
G231.0, G235.0	Spindle orientation stop position change signal	INDXA, INDXB	○	○	
G231.1, G235.1	Spindle rotation direction command signal while changing the orientation stop position	ROTAA, ROTAB	○	○	
G231.2, G235.2	Short–distant movement command while changing the orientation stop position signal	NRROA, NRROB	○	○	
G237.0, G237.1, G237.2, G237.3, G237.4	Synchronization control start signals (tool post 1)	SYN1M, SYN2M, SYN3M, SYN4M, SYN7M	●		1.7
G237.0 to G237.3	Signals to select the slave axis for simple synchronous control	SYNCX, SYNCZ, SYNC3, SYNC4	○		1.6
G237.3		SYNC4		○	
G237.5	Angular axis control–related Z–axis compensation movement signal	NOZAGC		☆	1.8
G237.6	In–feed control cut–in start signal	INFD		☆	5.12
G238.0, G238.1, G238.2, G238.3, G238.4	Synchronization control start signals (tool post 1)	PK1M, PK2M, PK3M, PK4M, PK7M	●		1.7
G239.0 to G239.7	Input signals for remote buffer	RMTDI0 to RMTDI7	○	○	5.11
G1333.2, G1333.3	Spindle feedback select signal;	SLPCA, SLPCB	●		9.4
G1437.0, G1437.1, G1437.2, G1437.3	Synchronization control start signals (tool post 2)	SYN1S, SYN2S, SYN3S, SYN4S	●		1.7
G1437.4, G1437.5, G1437.7, G1437.6,	Composite control start signals	MIX1, MIX2, MIX3, MIX4	●		
G1438.0, G1438.1, G1438.2, G1438.3	Synchronization control start signals (tool post 2)	PK1S, PK2S, PK3S, PK4S	●		

Address	Signal name	Symbol	T	M	Ref. item
X004.7	Ignore–the–fourth–axis–signal	4NG		○	1.4.3
X008.0, X008.1	Measuring position reached signals	XAE, ZAE	○		14.2
X008.0, X008.1, X008.2		XAE, YAE, ZAE		○	
X008.2, X008.3 X008.4, X008.5	Interlock signal for each axis and direction	+MIT1, –MIT1, +MIT2, –MIT2	○		2.5
X008.2, X008.3, X008.4	Skip signal	SKIP2, SKIP3, SKIP4	☆		14.3.3
X008.2, X008.3 X008.4, X008.5	Tool offset write signal	+MIT1, –MIT1, +MIT2, –MIT2	○		2.5, 14.4.2
X008.5	High–speed interlock signal	*RILK		○	2.5
X008.6	Skip signal (PMC axis control)	ESKIP	○	○	15.1
X008.7	Skip signal	SKIP	○	○	11.8, 14.3.1, 14.3.3
X016.5, X017.5, X018.5, X019.5	Reference position return deceleration signals	*DECX, *DECZ, *DEC3, *DEC4,	○		4.1
		*DECX, *DECY, *DECZ, *DEC4,		○	
X018.3, G133.0	Tool post selection signal	TRT2, TRT2PC	●		2.7
X018.5	Overtravel signal	*+LZ	○		2.3.1
X020.0, X020.1, X020.2, X020.3, X020.4, X020.5		*+LX, *+LY, *+LZ, *–LX, *–LY, *–LZ		○	
X021.4, G121.4	Emergency stop signal	*ESP, *ESP	○	○	2.1

[Number]

[A]

[B]

[C]

[D]

[E]

[F]

[G]

[H]

[I]

[J]

[L]

[M]

[N]

[O]

[P]

[R]

[S]

[T]

[V]

[W]

Revision Record

FANUC Series 0/00/0–Mate CONNECTION MANUAL (FUNCTION) (B–61393E–2)

Edition	Date	Contents	Edition	Date	Contents
02	Sep., ’98	● Addition of Series 0–DII ● Correction of errors			
01	Jul., ’97				