GE Fanuc Automation

Computer Numerical Control Products

Series 16*i* / 18*i* / 160*i* / 180*i* – Model A
Series 21*i* / 210*i* – Model A

Connection Manual (Function)

GFZ-63003EN-1/01 April 1997

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 damage of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

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

NOTE

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

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

PREFACE

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

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

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

3. Parameters
Describes parameters related with a function.

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

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

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

Applicable models

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

Model name	Abbreviation	
FANUC Series 16*i*–TA	16*i*–TA	Series 16*i*
FANUC Series 16*i*–MA	16*i*–MA	
FANUC Series 160*i*–TA	160*i*–TA	Series 160*i*
FANUC Series 160*i*–MA	160*i*–MA	
FANUC Series 18*i*–TA	18*i*–TA	Series 18*i*
FANUC Series 18*i*–MA	18*i*–MA	
FANUC Series 180*i*–TA	180*i*–TA	Series 180*i*
FANUC Series 180*i*–MA	180*i*–MA	
FANUC Series 21*i*–TA	21*i*–TA	Series 21*i*
FANUC Series 21*i*–MA	21*i*–MA	
FANUC Series 210*i*–TA	210*i*–TA	Series 210*i*
FANUC Series 210*i*–MA	210*i*–MA	

For ease of understanding, the models are categorized as follows:

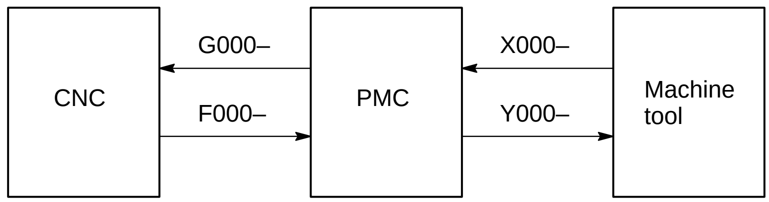
T series: 16*i*–TA, 160*i*–TA, 18*i*–TA, 180*i*–TA, 21*i*–TA, 210*i*–TA
M series: 16*i*–MA, 160*i*–MA, 18*i*–MA, 180*i*–MA, 21*i*–MA, 210*i*–MA

> **NOTE**
> Some functions described in this manual may not be applied to some products.
> For details, refer to the DESCRIPTIONS manual (B–63002EN).

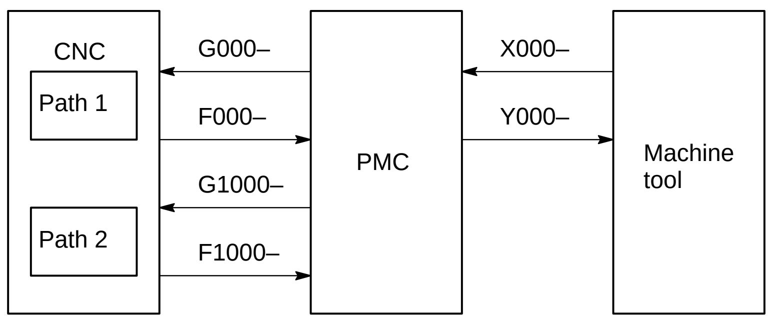
Signal description

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

[For one–path control]



[For two–path control]



> **NOTE**
> 1 In two–path control, the signals of the same functions are prepared for both of path 1 and path 2. These signals have suffix #1 and #2 to their signal names on path 1 and path 2, respectively.
> When a signal is common to both paths, the signal is prepared only to path 1 and the suffix #1 and #2 are not attached.
> 2 In the context, signals are described on path 1 only. Refer to Appendix A.1.2 List of addresses for two–path control for signals on path 2.
> 3 For the signals, a single data number is assigned to 8 bits. Each bit has a different meaning.

Expression of signals

One address accommodates eight signals.

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

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

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

[Example 1]

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

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

Parameter description

Parameters are classified by data type as follows :

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

NOTE

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

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

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

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

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

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

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

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

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

[Example 1]

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

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

[Example 2]

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

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

[Example 3]

The following parameter is provided only for the M series.

1450		T series
	F1 digit feed . . .	M series

Related Manuals

The table below lists manuals related to MODEL A of Series 16*i*, Series 18*i*, Series 160*i* and Series 180*i*.
In the table, this manual is marked with an asterisk(*).

Table 1 Related manuals

Manual name	Specification Number	
DESCRIPTIONS	B–63002EN	
CONNECTION MANUAL (HARDWARE)	B–63003EN	
CONNECTION MANUAL (FUNCTION)	B–63003EN–1	*
OPERATOR'S MANUAL For Lathe	B–63004EN	
OPERATOR'S MANUAL For Maching Center	B–63014EN	
MAINTENANCE MANUAL	B–63005EN	
PARAMETER MANUAL	B–63010EN	
PROGRAMMING MANUAL (Macro Compiler / Macro Executer)	B–61803E–1	
FAPT MACRO COMPILER PROGRAMMING MANUAL	B–66102E	
FANUC Super CAP T / Super CAP II T OPERATOR'S MANUAL	B–62444E–1	
FANUC Super CAP M / Super CAP II M OPERATOR'S MANUAL	B–62154E	
FANUC Super CAP M PROGRAMMING MANUAL	B–62153E	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION I FOR LATHE OPERATOR'S MANUAL	B–61804E–1	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION FOR LATHE OPERATOR'S MANUAL	B–61804E–2	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION FOR MACHINING CENTER OPERATOR'S MANUAL	B–61874E–1	

Manuals related to Series 21*i*/210*i*–MODEL A

The table below lists the manuals related to the FANUC Series 21*i*/210*i*–MODEL A. This manual is indicated by an asterisk (*).

Table 2 Manuals related to the Series 21*i*/210*i*–MODEL A

Manual name	Order No.	
DESCRIPTION	B–63002EN	
CONNECTION MANUAL (HARDWARE)	B–63083EN	
CONNECTION MANUAL (FUNCTION)	B–63003EN–1	*
OPERATOR'S MANUAL (For Lathe)	B–63084EN	
OPERATOR'S MANUAL (For Machining Center)	B–63094EN	
MAINTENANCE MANUAL	B–63085EN	
PARAMETER MANUAL	B–63090EN	
MACRO COMPILER/MACRO EXECUTOR, PROGRAMMING MANUAL	B–61803E–1	
FAPT MACRO COMPILER (For PCs), PROGRAMMING MANUAL	B–66102E	

Manuals related to the Control Motor α series

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

Table 3 Manuals related to the Control 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 CONTROL MOTOR 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

Series 16*i*, Series 160*i*

Item		M series	T series
No. of basic controlled axes	1–path	3 axes	2 axes
	2–path	3 axes per path	2 axes per path
Controlled axes expansion (total)	1–path	Max. 8 axes (Including the Cs axis)	Max. 8 axes (Including the Cs axis)
	2–path	Max. 7 axes per path (Feed 6 axes + Cs axis)	Max. 7 axes per path (Feed 6 axes + Cs axis)
Basic simultaneously controlled axes	1–path	2 axes	2 axes
	2–path	2 axes per path	2 axes per path
Simultaneously controlled axes expansion (total)	1–path	Max. 6 axes	Max. 6 axes
	2–path	Max. 6 axes per path	Max. 6 axes per path

Series 18*i*, Series 180*i*

Item		M series	T series
No. of basic controlled axes	1–path	3 axes	2 axes
	2–path	—	2 axes per path
Controlled axes expansion (total)	1–path	Max. 6 axes (Including the Cs axis)	Max. 6 axes (Including the Cs axis)
	2–path	—	Max. 5 axes per path (Feed 4 axes + Cs axis)
Basic simultaneously controlled axes	1–path	2 axes	2 axes
	2–path	—	2 axes per path
Simultaneously controlled axes expansion (total)	1–path	Max. 4 axes	Max. 4 axes
	2–path	—	Max. 4 axes per path

Series 21*i*, Series 210*i*

Item		M series	T series
No. of basic controlled axes	1–path	3 axes	2 axes
Controlled axes expansion (total)	1–path	Max. 4 axes (Including the Cs axis)	Max. 4 axes (Including the Cs axis)
Basic simultaneously controlled axes	1–path	2 axes	2 axes
Simultaneously controlled axes expansion (total)	1–path	Max. 4 axes	Max. 4 axes

Parameter

1010	Number of CNC–controlled axes

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

[Data type] Byte

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

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

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

X, Y, Z, and A axes: Controlled by the CNC and PMC
B and C axes: Controlled by the PMC

Then set this parameter to 4 (total 4: X, Y, Z, and A)

Alarm and message

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

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 with 2–path control having nine or more axes.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.2.1	Controlled Axes
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.2.1	Controlled Axes
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.2.1	Controlled Axes
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.2.1	Controlled Axes

1.2 SETTING EACH AXIS

1.2.1 Name of Axes

General

Each axis that is controlled by the CNC (including those controlled by the PMC) must be named. Select and set names from among X, Y, Z, A, B, C, U, V, and W (with parameter 1020).
The names of the basic axes, however, are fixed (X, Y, and Z for the M series and X and Z for the T series). The names of additional axes can be selected, as desired, from the names other than those for the basic axes. The same name cannot be assigned to more than one axis.
With 2–path control, the name of basic axis for one path is fixed; the names of additional axes can be optionally selected from axes names except axes names of basic axes by using parameter No. 1020. For one path, the same axis name cannot be assigned to multiple axes, but the same axis name can be used with the other path.

Parameter

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

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

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

NOTE
1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
2 The same axis name cannot be assigned to more than one axis.
3 When the secondary auxiliary function is provided, address used for the secondary auxiliary function (address B in the T series, address set in parameter No. 3460 in the M series) cannot be used as an axis name. In the T series, when CCR, #4 of parameter 3405, is set to 1, address A and C may not be used with functions such as chamfering, corner R, or direct drawing dimensions programming.

Note

NOTE

With 2–path control, 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 an 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.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.2.2	NAMES OF AXES
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.2.2	NAMES OF AXES
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.2.2	NAMES OF AXES
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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)). Select IS–B or IS–C using bit 1 (ISC) of parameter 1004. 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 T series. Whether diameter programming or radius programming is used is selected by parameter DIAx (No. 1006#3) on each axis. Also, parameter IPR (No. 1004#7) can make the least input increment of IS–B and IS–C ten times the least command increment on each axis.

Parameter

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

Setting entry is acceptable.

[Data type] Bit

INI Unit of input

0 : In mm
1 : In inches

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

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

[Data type] Bit

INM Least command increment on the linear axis

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

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

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

[Data type] Bit

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

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

> **NOTE**
> IS–A cannot be used at present.

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

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

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

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

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

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

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

[Data type] Bit axis

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

0 : Radius
1 : Diameter

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.2.3	Increment System
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.2.3	Increment System
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.2.3	Increment System
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.2.3	Increment System

1.2.3 Specifying the Rotation Axis

General

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

Parameter

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

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

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis.

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

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

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

[Data type] Bit axis

ROAx The roll–over function of a rotation axis is

0 : Invalid

1 : Valid

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

RABx In the absolute commands, the axis rotates in the direction

0 : In which the distance to the target is shorter.
1 : Specified by the sign of command value.

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

RRLx Relative coordinates are

0 : Not rounded by the amount of the shift per one rotation
1 : Rounded by the amount of the shift per one rotation

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

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

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

[Data type] Two–word axis

[Unit of data]

Increment system	Unit of data	Standard value
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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.2	Rotary Axis Roll–over
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.19.2	Rotary Axis Roll–over
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.20.2	Rotary Axis Roll–over
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.19.2	Rotary Axis Roll–over

1.2.4 Controlled Axes Detach

General

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

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

Signal

Controlled axis detach signals DTCH1 – DTCH8 <G124>

[Classification] Input signal

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

DTCH 1

1 ..... The 1st axis is detached.
2 ..... The 2nd axis is detached.
: :
: :
8 ..... The 8th axis is detached.

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

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

Controlled axis detach status signals MDTCH1 – MDTCH8 <F110>

[Classification] Output signal

[Function] These signals notify the PMC that the corresponding axes have been released from control.

These signals are provided for each control axis; the affixed number of the signal name shows the control axis number.

MDTCH 1

- 1 ..... The 1st axis is detached.
- 2 ..... The 2nd axis is detached.
- : :
- : :
- 8 ..... The 8th axis is detached.

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

– When the corresponding axes are released from control

These signals are 0 in the following case:

– When the corresponding axes are under control

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G124	DTCH8	DTCH7	DTCH6	DTCH5	DTCH4	DTCH3	DTCH2	DTCH1

	#7	#6	#5	#4	#3	#2	#1	#0
F110	MDTCH8	MDTCH7	MDTCH6	MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1

Parameter

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

Setting entry is acceptable.

[Data type] Bit axis

RMVx Releasing the assignment of the control axis for each axis

0 : Not released

1 : Released

> **NOTE**
> RMVx is valid when RMBx in parameter 1005 is 1.

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

[Data type] Bit axis

RMBx Releasing the assignment of the control axis for each axis (signal input and setting input)

0 : Invalid

1 : Valid

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

0 : Disabled
1 : Enabled

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

Caution

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

Note

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

1.2.5 Outputting the Movement State of an Axis

General

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

Signal

Axis moving signals MV1 – MV8 <F102>

[Classification] Output signal

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

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

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

- The corresponding axis has started moving.
- In manual handle feed mode, the handle feed axis of the corresponding axis has been selected.

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

- When the move command for the corresponding axis has been distributed (when bit 6 (MVX) of parameter 3003 is 0)
- When deceleration for the corresponding axis has been completed and the axis is set to the in–position condition. If in–position check is not performed, when the deceleration for the corresponding axis is completed. (When bit 6 (MVX) of parameter 3003 is 1)

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

Axis moving direction signals MVD1 – MVD8 <F106>

[Classification] Output signal

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

MVD 1

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

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

> **CAUTION**
> These signals maintain their existing condition during a stop, indicating the direction of the axes’ movement before being stopped.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F102	MV8	MV7	MV6	MV5	MV4	MV3	MV2	MV1
F106	MVD8	MVD7	MVD6	MVD5	MVD4	MVD3	MVD2	MVD1

Parameter

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

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

[Data type] Bit

MVX The axis moving signal is set to 0 when:

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

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

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

0: Output
1: Not output

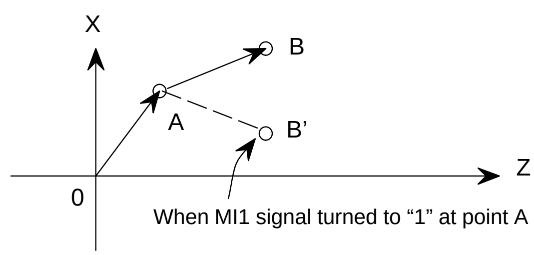
Caution

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

1.2.6 Mirror Image

General

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



Mirror image (Example for T series)

However, the following directions are not reversed:

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

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

Signal

Mirror image signal MI1 – MI8 <G106>

[Classification] Input signal

[Function] Apply mirror image to the specified axes.

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

MI 1

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

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

a) During offset cancel;

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

Mirror image check signal MMI1 – MMI8<F108>

[Classification] Output signal

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

MMI 1

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

[Output condition] These signals turn to "1" when:

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

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

These signals turn to "0" when:

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

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1

	#7	#6	#5	#4	#3	#2	#1	#0
F108	MMI8	MMI7	MMI6	MMI5	MMI4	MMI3	MMI2	MMI1

Parameter

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

Setting entry is acceptable.

[Data type] Bit axis

MIRx Mirror image for each axis
0 : Mirror image is off.
1 : Mirror image is on.

Warning

WARNING
1 When programmable mirror image 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 MMI8 <F108>.

Caution

CAUTION
Even when the mirror image is applied, commands which do not actuate mirror image (such as automatic reference position return and manual operation) do not affect mirror image check signals MMI1 to MMI8 <F108>.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.4.9	Mirror Image
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.4.7	Mirror Image
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.4.8	Mirror Image
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.4.7	Mirror Image

1.2.7 Follow–up

General

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

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

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

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

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

Signal

Follow–up signal *FLWU <G007#5>

[Classification] Input signal

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

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

Signal address

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

Parameter

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

[Data type] Bit axis

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

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

1 : Follow–up is not performed.

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

Reference item

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

1.2.8 Servo Off (Mechanical Handle)

General

Place the controlled axes in the servo off state; 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 SVF1 – SVF8 <G126>

[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 1

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

[Operation] These signals 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

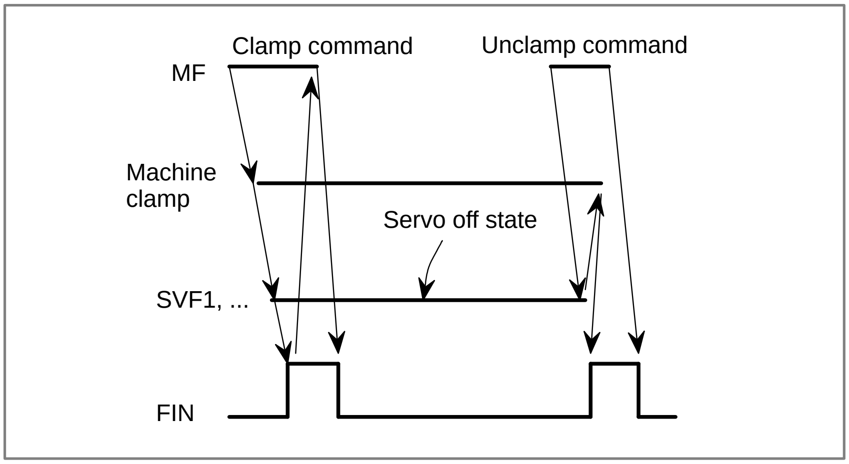
	#7	#6	#5	#4	#3	#2	#1	#0
G126	SVF8	SVF7	SVF6	SVF5	SVF4	SVF3	SVF2	SVF1

Caution

CAUTION

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

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



Reference item

CONNECTION MANUAL (This manual)	1.2.7	Follow–up

1.2.9 Position Switch

General

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

Signal

Position switch signal PSW01 – PSW10 <F070#0 – F071#1>

[Classification] Output signal

[Function] Notifies that the machine coordinates along the controlled axes specified by parameters (6910 to 6919) are within the ranges specified by parameters (6930 to 6939 and 6950 to 6959). 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
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071							PSW10	PSW09

Parameter

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

6910	Axis corresponding to the first position switch
6911	Axis corresponding to the second position switch
6912	Axis corresponding to the third position switch
6913	Axis corresponding to the fourth position switch
6914	Axis corresponding to the fifth position switch
6915	Axis corresponding to the sixth position switch
6916	Axis corresponding to the seventh position switch
6917	Axis corresponding to the eighth position switch
6918	Axis corresponding to the ninth position switch
6919	Axis corresponding to the tenth position switch

[Data type] Byte

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

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**

6930	Maximum operation range of the first position switch
6931	Maximum operation range of the second position switch
6932	Maximum operation range of the third position switch
6933	Maximum operation range of the fourth position switch
6934	Maximum operation range of the fifth position switch
6935	Maximum operation range of the sixth position switch
6936	Maximum operation range of the seventh position switch
6937	Maximum operation range of the eighth position switch
6938	Maximum operation range of the ninth position switch
6939	Maximum operation range of the tenth position switch

[Data type] Two–word

[Unit of data]

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

[Valid data range] –99999999 to +99999999

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

- **Minimum operation range**

6950	Minimum operation range of the first position switch
6951	Minimum operation range of the second position switch
6952	Minimum operation range of the third position switch
6953	Minimum operation range of the fourth position switch
6954	Minimum operation range of the fifth position switch
6955	Minimum operation range of the sixth position switch
6956	Minimum operation range of the seventh position switch
6957	Minimum operation range of the eighth position switch
6958	Minimum operation range of the ninth position switch
6959	Minimum operation range of the tenth position switch

[Data type] Two–word

[Unit of data]

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

[Valid data range] –99999999 to +99999999

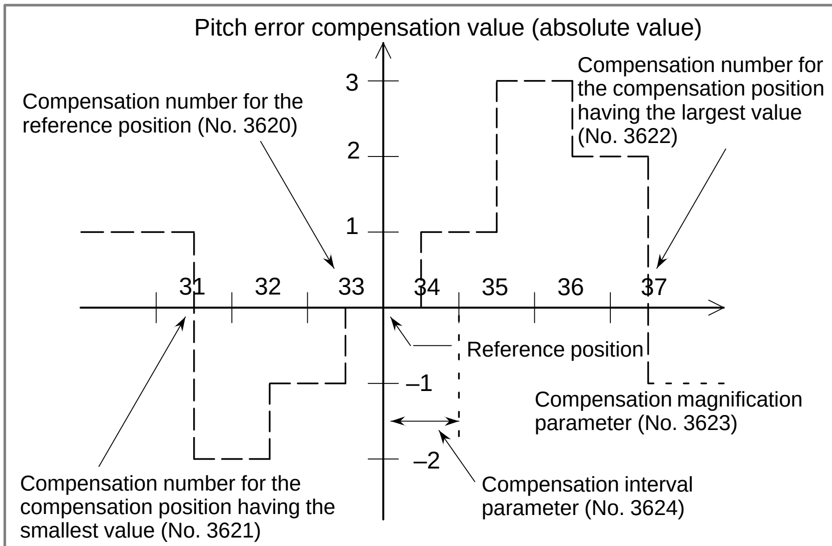
These parameters set the minimum operation range of the first through tenth position switches.

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

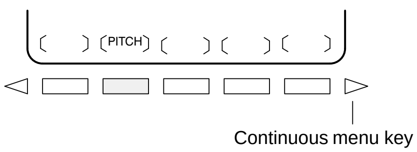
- Number of the pitch error compensation position at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation position having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation position having the largest value (for each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

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

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:

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



2 Press function key SYSTEM.

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

```
PIT–ERROR SETTING                  O0000 N00000

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

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

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

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

Explanations

- **Specifying the compensation position**

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

- **Compensation position number**

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

Examples

- **For linear axis**

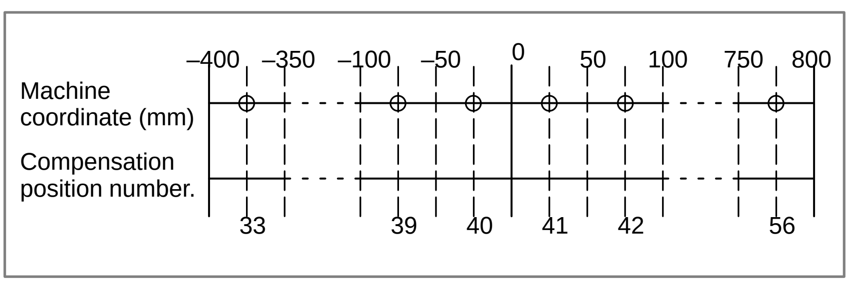
· Machine stroke: –400 mm to +800 mm
· Interval between the pitch error compensation positions: 50 mm
· No. of the compensation position of the reference position: 40
If the above is specified, the No. of the farthest compensation position in the negative direction is as follows:

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

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

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

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



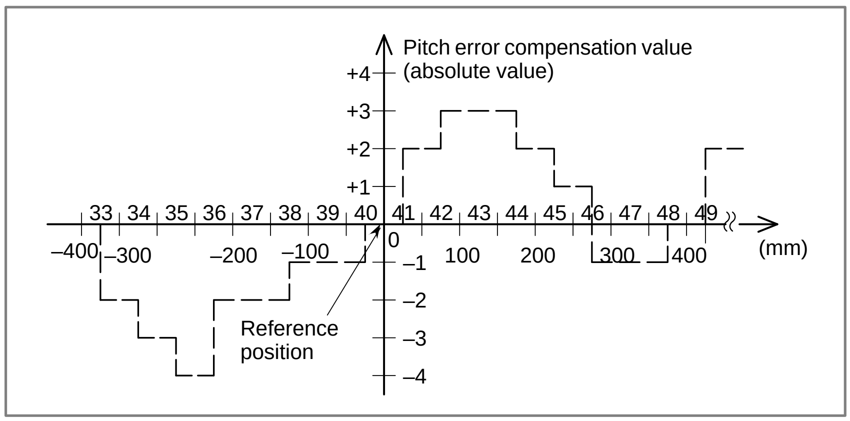
Compensation values are output at the positions indicated by ○.

Therefore, set the parameters as follows:

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

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

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

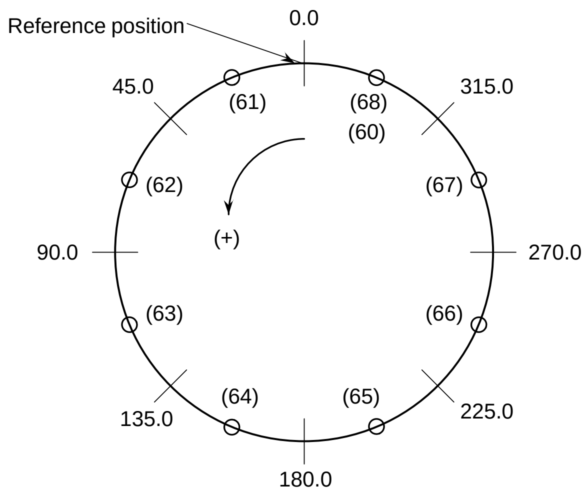


- **For rotary axis**

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

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

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



Compensation values are output at the positions indicated by ○ .

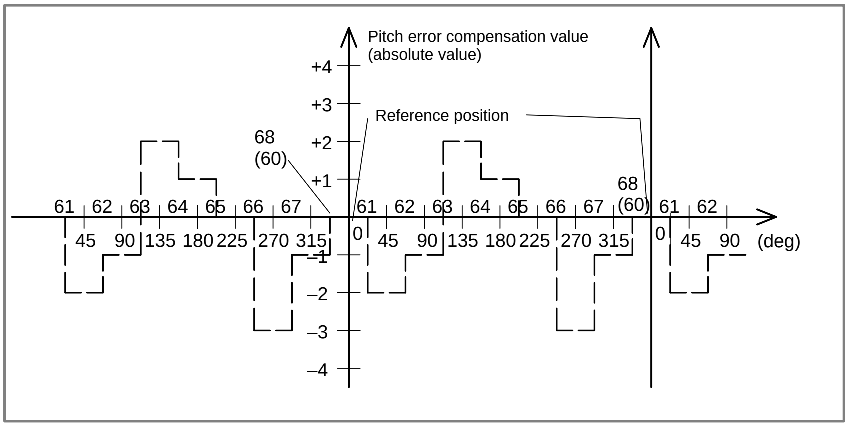
Therefore, set the parameters as follows:

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

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

The following is an example of compensation amounts.

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



Parameter

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

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

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

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

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

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

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

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

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

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

[Data type] Word axis

[Unit of data] Number

[Valid data range] 0 to 1023

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

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

3623	Magnification for pitch error compensation for each axis

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

[Data type] Byte axis

[Unit of data] 1

[Valid data range] 0 to 100

Set the magnification for pitch error compensation for each axis.

If the magnification is set to 1, the same unit as the detection unit is used for the compensation data.

3624	Interval between pitch error compensation positions for each axis

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] 0 to 99999999

The pitch error compensation positions are 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)/3750

Unit:

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

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

[Example] When the maximum rapid traverse rate is 15000 mm/min, the minimum interval between pitch error compensation positions is 4 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 0 to 100 in parameter 3623.

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. 3624).

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.

· If the compensation position Nos. on the positive or negative direction do not fall within the range of 0 to 1023.

· If the compensation position Nos. do not conform to the following relationship:
Negative side ≦ Reference position < Positive side

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III 8.6.3	Inputting pitch error compensation data
		III 8.6.4	Outputting pitch error compensation data
		III 11.5.2	Displaying and setting pitch error compensation data
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III 8.6.3	Inputting pitch error compensation data
		III 8.6.4	Outputting pitch error compensation data
		III 11.5.2	Displaying and setting pitch error compensation data
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III 8.6.3	Inputting pitch error compensation data
		III 8.6.4	Outputting pitch error compensation data
		III 11.5.2	Displaying and setting pitch error compensation data
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III 8.6.3	Inputting pitch error compensation data
		III 8.6.4	Outputting pitch error compensation data
		III 11.5.2	Displaying and setting pitch error compensation data

1.3.2 Backlash Compensation

General

- **Backlash compensation**

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

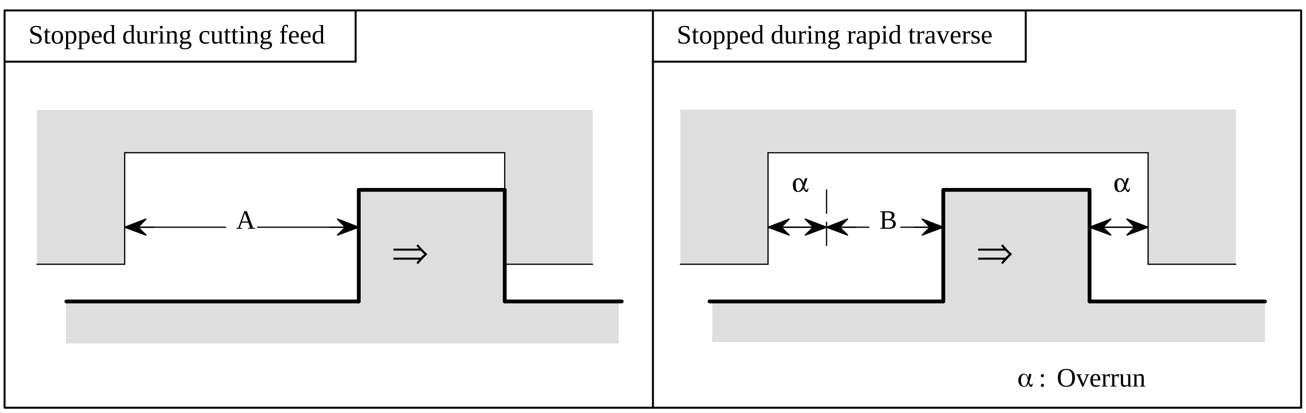
- **Backlash compensation for each rapid traverse and cutting feed**

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

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

Change of feedrate / Change of direction of movement	Cutting feed to cutting feed	Rapid traverse to rapid traverse	Rapid traverse to cutting feed	Cutting feed to rapid traverse
Same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
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 No. 1851 and that at rapid traverse (B) in parameter No. 1852.

Parameter

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

[Data type] Bit

RBK Backlash compensation applied separately for cutting feed and rapid traverse

0 : Not performed
1 : Performed

1851	Backlash compensating value for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensating value for each axis.

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

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

1852	Backlash compensating value used for rapid traverse for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

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

This parameter is valid when RBK is set to 1.

Caution

CAUTION
The backlash compensation for 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 No. 1851 irrespective of a rapid traverse and a cutting feed.

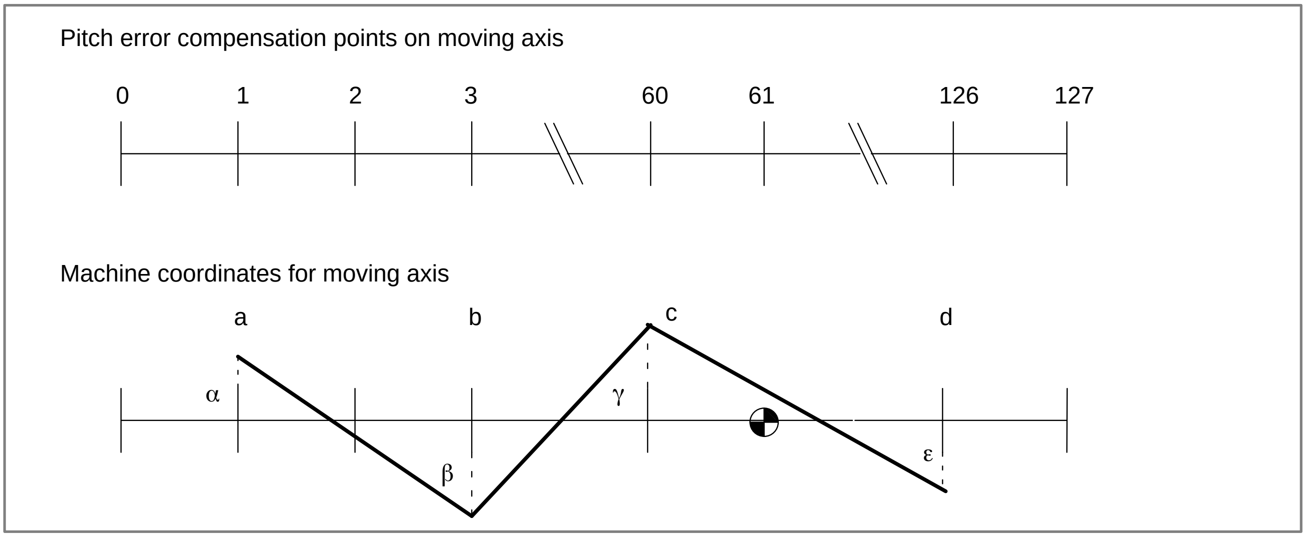
Note

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

1.3.3 Straightness Compensation

General

For a machine tool with a long stroke, deviations in straightness between axes may deteriorate machining accuracy. For this reason, when an axis moves, other axes are compensated in detection units to improve straightness. This improvement results in better machining accuracy.
When an axis (parameter Nos. 5711 to 5713) moves, the corresponding compensation axis (parameter Nos. 5721 to 5723) is compensated.
That is, the compensation axis is compensated at the pitch error compensation position (See 1.3.1) of the moving axis.



a, b, c, d — Compensation position numbers of the moving axis (parameter No. 5501 to 5524)
α, β, γ, ε — Compensation for compensation axis (parameter No. 5551 to 5574)

The compensation from point a to point b is calculated from the formula: (β–α)/(b–a).

Parameter

5711	Axis number of moving axis 1
5712	Axis number of moving axis 2
5713	Axis number of moving axis 3

[Data type] Byte

[Unit of data] Axis number (When 0, compensation is not performed.)

[Valid data range] 1 to Number of controlled axes

Set the axis numbers of moving axes.

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

[Data type] Byte

[Unit of data] Axis number (When 0, compensation is not performed.)

[Valid data range] 1 to Number of controlled axes

Set the axis numbers of compensation axes.

5731	Compensation position number a of moving axis 1
5732	Compensation position number b of moving axis 1
5733	Compensation position number c of moving axis 1
5734	Compensation position number d of moving axis 1
5741	Compensation position number a of moving axis 2
5742	Compensation position number b of moving axis 2
5743	Compensation position number c of moving axis 2
5744	Compensation position number d of moving axis 2
5751	Compensation position number a of moving axis 3
5752	Compensation position number b of moving axis 3
5753	Compensation position number c of moving axis 3
5754	Compensation position number d of moving axis 3

[Data type] Word

[Unit of data] Number
(Compensation position numbers in stored pitch error compensation)

[Valid data range] 0 to 1023

Set four compensation positions for each moving axis.

5761	Compensation corresponding to compensation position number a of moving axis 1
5762	Compensation corresponding to compensation position number b of moving axis 1
5763	Compensation corresponding to compensation position number c of moving axis 1
5764	Compensation corresponding to compensation position number d of moving axis 1
5771	Compensation corresponding to compensation position number a of moving axis 2
5772	Compensation corresponding to compensation position number b of moving axis 2
5773	Compensation corresponding to compensation position number c of moving axis 2
5774	Compensation corresponding to compensation position number d of moving axis 2
5781	Compensation corresponding to compensation position number a of moving axis 3
5782	Compensation corresponding to compensation position number b of moving axis 3
5783	Compensation corresponding to compensation position number c of moving axis 3
5784	Compensation corresponding to compensation position number d of moving axis 3

[Data type] Word

[Unit of data] Detection unit

[Valid data range] –32768 to +32767

Set compensation for each compensation position.

Alarm and message

Number	Message	Description
5046	ILLEGAL PARAMETER (ST. COMP)	Parameters related to straightness compensation have been erroneously specified. Possible causes are as follows: 1. Invalid axis numbers have been assigned to move or compensation axes. 2. The number of pitch–error compensation positions between the maximum positive and maximum negative positions exceeds 128. 3. Straightness compensation position numbers have been assigned in other than ascending order. 4. Straightness compensation positions could not be located between the maximum positive and maximum negative pitch–error compensation positions. 5. The amount of compensation per compensation position is too large or too small.

Note

NOTE

1 The straightness compensation function can be used after a moving axis and its compensation axis have returned to the reference position.
2 After setting parameters for straightness compensation, be sure to turn off the NC power.
3 Set parameters for straightness compensation according to the following conditions:
   - The compensation at a compensation position must be within the range –128 to 127.
   - Compensation positions must be set so that "a≦b≦c≦d" is satisfied.
   - Compensation positions must exist between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data for each axis. Four compensation positions can be set to 0 at a time. In this case, compensation is not performed.
4 To add the straightness compensation function option, the stored pitch error compensation option is needed.
   In this case, the number of compensation positions of each axis between the compensation position with the largest positive value and that with the largest negative value in the stored pitch error compensation data must be equal to or less than 128.
5 Straightness compensation data is superposed on stored pitch error compensation data and output.

1.4 SETTINGS RELATED TO SERVO–CONTROLLED AXES

The servo interface of the Series 16 features the following:
Digitally controlled AC servo motor
Motor feedback with serial pulse coders
(1) Absolute pulse coder with a resolution of 1,000,000 pulses/rev
(2) Absolute pulse coder with a resolution of 65,536 pulses/rev
(3) Incremental pulse coder with a resolution of 10,000 pulses/rev
Scale feedback with A/B/Z signal interface

1.4.1 Parameters Related to Servo

General

Explanation of terms frequently used in CNC

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

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

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

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

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

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

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

Parameter

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

[Data type] Bit

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

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

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

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

[Data type] Bit axis

OPTx Position detector

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

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

0 : Not corresponding
1 : Corresponding

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

APCx Position detector

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

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

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

[Data type] Bit axis

DM1x to DM3x Setting of detection multiplier

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

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

1820	Command multiplier for each axis (CMR)

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

[Data type] Byte axis

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

Least command increment = detection unit x command multiplier

Relationship between the increment system and the least command increment

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

The value set in the parameter is obtained as follows:

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

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

Valid data range: 102 to 127

(2) When command multiplier is 1 to 48

Set value = 2 × command multiplier
Valid data range: 2 to 96

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

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

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

1825	Servo loop gain for each axis

[Data type] Word axis

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

[Valid data range] 1 to 9999

Set the loop gain for position control for each axis.

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

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

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

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

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

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

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

If, in the stopped state, 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).

1832	Feed stop positioning deviation for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the feed stop positioning deviation for each axis.

If the positioning deviation exceeds the feed stop positioning deviation during movement, pulse distribution and acceleration/deceleration control are stopped temporarily. When the positioning deviation drops to the feed stop positioning deviation or below, pulse distribution and acceleration/deceleration control are resumed.

The feed stop function is used to reduce overshoot in acceleration/deceleration mainly by large servo motors.
Generally, set the middle value between the positioning deviation limit during movement and the positioning deviation at rapid traverse as the feed stop positioning deviation.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to +99999999

A grid shift is set for each axis.

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

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

1.4.2 Absolute Position Detection

General

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

Parameter

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

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

[Data type] Bit axis

APCx Position detector

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

1.4.3 FSSB Setting

Overview

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

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

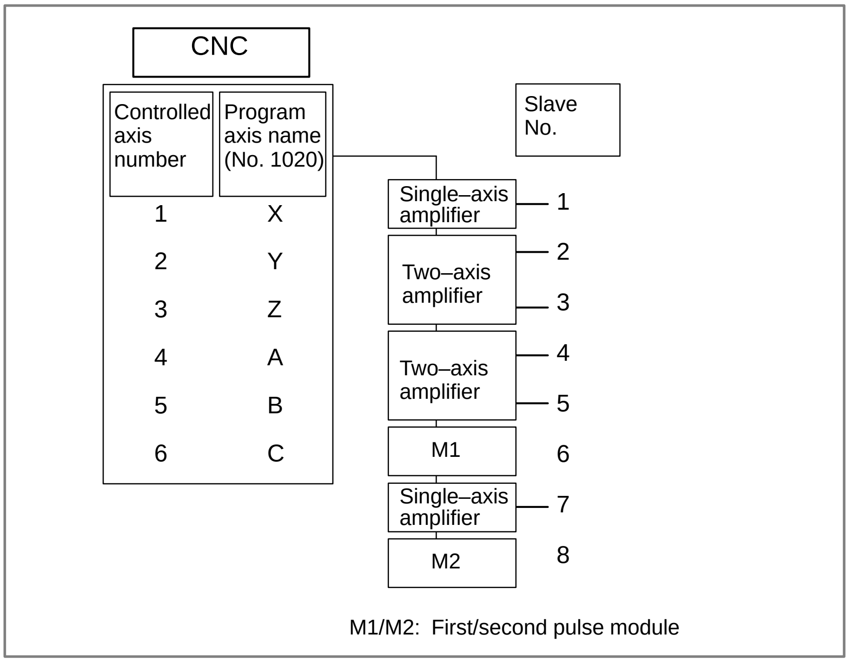
- Parameter No. 1023
- Parameter No. 1905
- Parameter Nos. 1910 to 1919
- Parameter Nos. 1936 and 1937

These parameters can be specified using the following methods:

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

Slave

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

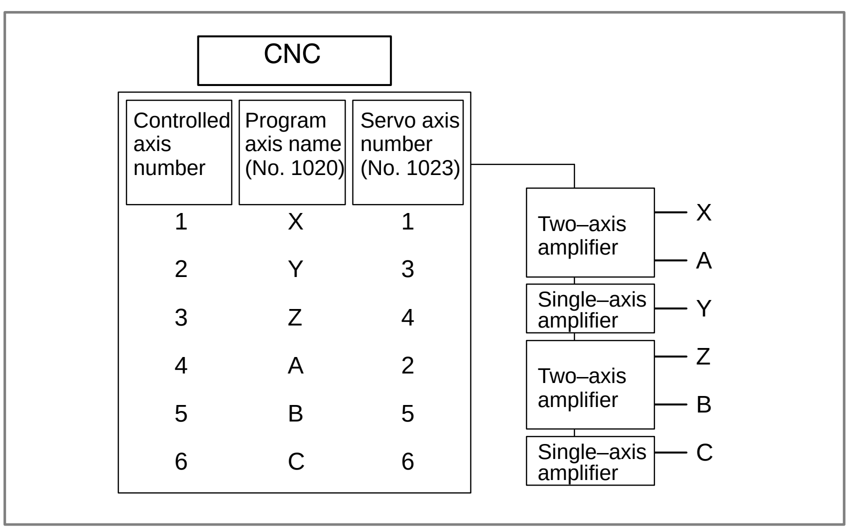


Manual setting 1

The manual setting 1 is usable when the following parameter specifications made:

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

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



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

- No pulse module can be used; hence, no separate position detectors can be used.
- No number can be skipped in parameter No. 1023; for example, number 3 cannot be used for any axis unless number 2 is used.
- The following servo functions cannot be used:
  - ❑ Learning control
  - ❑ High–speed current loop
  - ❑ High–speed interface axis
  - ❑ Simple electronic gearbox (EGB)

Automatic setting

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

Bit 0 of parameter No. 1902 = 0

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

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

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

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

NOTE
To use the simple electronic gearbox (EGB), perform EGB axis setting (parameter No. 7771) before automatic setting on the FSSB setting screen. Otherwise, automatic setting cannot be performed correctly.

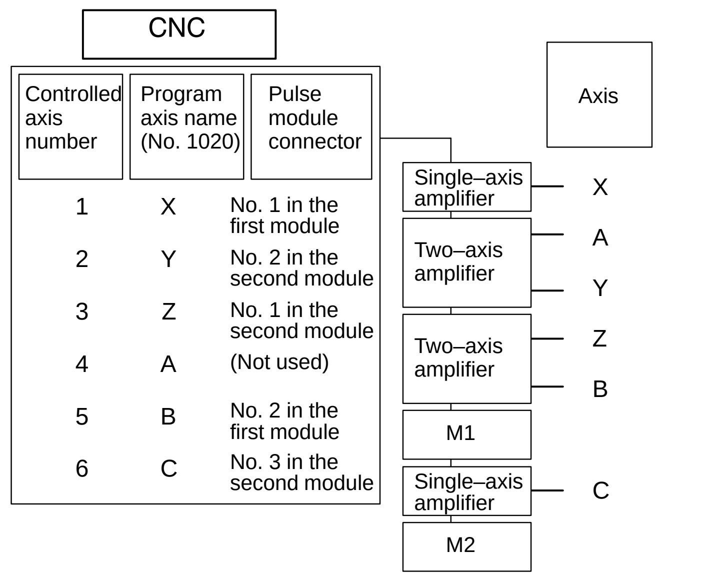
Manual setting 2

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

Bit 0 of parameter No. 1902 = 1

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

Example of axis configuration and manual setting 2
(Refer to the corresponding description for the meaning of each parameter.)



M1/M2: First/second pulse module

No.	1902#0 FMD
	1

No.	1910	1911	1912	1913	1914	1915	1916	1917	1918	1919
	0	1	2	3	4	16	5	48	40	40

No.	1023	1905#0 FSL	1905#6 PM1	1905#7 PM2	1936	1937
X	1	0	1	0	0	0
Y	3	0	0	1	0	1
Z	4	1	0	1	0	0
A	2	1	0	0	0	0
B	5	0	1	0	1	0
C	6	1	0	1	0	2

FSSB display and setting procedure

• Display

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

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

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

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

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

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

```
AMPLIFIER SETTING                        O1000 N00001
  NO.  AMP  SERIES    UNIT    CUR.   AXIS NAME
    1  A1–L    α     SVM–HV    40AL     1    X
    2  A1–M    α        SVM    12A      2    Y
    3  A2–L    β        SVM    40A      3    Z
    4  A3–L    α        SVM    20A      4    A
    5  A3–M    α        SVM    40A      5    B
    7  A4–L    α        SVU   240A      6    C


  NO.  EXTRA  TYPE   PCB ID
    6    M1     A    0000 DETECTOR(8AXES)
    8    M2     B    12AB
>_
MDI **** *** ***     13:11:56
[  AMP  ][  AXIS  ][ MAINTE ][      ][ (OPRT) ]
```

The amplifier setting screen consists of the following items:

- NO. . . . slave number
  The numbers of up to ten slaves (up to eight amplifiers and up to two pulse modules) connected via the FSSB are displayed sequentially, with the one nearest the CNC being number 1.

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

(2) Axis setting screen
The axis setting screen displays the information shown below:

```
AXIS SETTING                           O1000 N00001
  AXIS   NAME   AMP     M1  M2  1-DSP   CS   TNDM
    1      X    A1-L    0   0     0     0     1
    2      Y    A1-M    1   0     1     0     0
    3      Z    A2-L    0   0     0     1     0
    4      A    A3-L    0   0     0     0     2
    5      B    A3-M    0   0     0     0     0
    6      C    A4-L    0   0     0     0     0






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

The axis setting screen displays the following items:

- AXIS ..... controlled axis number
  This item is the placing of the NC controlled axis.
- NAME .... controlled axis name
- AMP ..... type of the amplifier connected to each axis
- M1 .... connector number for pulse module 1
  This item is the number of the connector for pulse module 1, specified in parameter No. 1931.
- M2 .... connector number for pulse module 2
  This item is the number of the connector for pulse module 2, specified in parameter No. 1932.
- 1–DSP
  This item is the value specified in bit 0 (parameter 1DSP) of parameter No. 1904. It is 1 for an axis (such as a learning control axis, high–speed current loop axis, or high–speed interface axis) that exclusively uses a DSP, which is usually shared by two axes.
- CS ..... Cs contour controlled axis
  This item is the value specified in parameter No. 1933. It is 1 for the Cs contour controlled axis.
- TNDM
  This item is the number specified in parameter No. 1934. Consecutive odd and even numbers are displayed for the master and slave axes for tandem control.

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

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





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

```
AMPLIFIER MAINTENANCE          O1000 N00001
  AXIS   NAME    EDITION     TEST    MEINTE-NO.
    1     X        01A      970123       01
    2     Y        01A      970123       01
    3     Z        01A      970123       01
    4     A        02B      970123       01
    5     B        02B      970123       01
    6     C        02B      970123       01




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

The amplifier maintenance screen displays the following items:

- AXIS . . . . controlled axis number
- NAME . . . . controlled axis name
- AMP . . . . type of amplifier connected to each axis
- SERIES . . . . servo amplifier series of an amplifier connected to each axis
- UNIT . . . . unit type of a servo amplifier connected to each axis
- AXES . . . . maximum number of axes controlled by an amplifier connected to each axis
- CUR. . . . . maximum rating current for amplifiers connected to each axis
- EDITION . . . . . . unit version number of an amplifier connected to each axis
- TEST . . . . date of test performed on an amplifier connected to each axis
  Example) 970123 = January 23, 1997
- MEINTE–No. . . engineering change number for an amplifier connected to each axis

- **Setting**

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

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

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

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

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

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

> **NOTE**
> 1 For the parameters to be specified on the FSSB setting screen, do not attempt to enter values on the parameter screen using the MDI or a G10 command. Use only the FSSB screen to enter values for these parameters.
> 2 If pressing the [SETING] key results in a warning message being displayed, retry data entry, or press the [CANCEL] key to clear the warning message. Note that pressing the reset key does not clear the warning message.

(1) Amplifier setting screen

```
AMPLIFIER SETING                         O1000 N00001
  NO.  AMP  SERIES   UNIT    CUR.   AXIS NAME
    1  A1–L    α    SVM–HV   40AL    1     X
    2  A1–M    α       SVM    12A    2     Y
    3  A2–L    β       SVM    40A    3     Z
    4  A3–L    α       SVM    20A    4     A
    5  A3–M    α       SVM    40A    5     B
    7  A4–L    α       SVU   240A    6     C


  NO.  EXTRA  TYPE   PCB ID
    6   M1     A    0000 DETECTOR(8AXES)
    8   M2     B    12AB
>_
MDI **** *** ***     13:11:56
[  AMP  ][  AXIS  ][ MAINTE ][      ][ (OPRT) ]
```

The amplifier setting screen displays the following items:

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

(2) Axis setting screen

```
AXIS SETTING                               O1000 N00001
  AXIS   NAME   AMP     M1  M2  1-DSP CS  TNDM
    1      X    A1-L     0   0    0    0    1
    2      Y    A1-M     1   0    1    0    0
    3      Z    A2-L     0   0    0    1    0
    4      A    A3-L     0   0    0    0    2
    5      B    A3-M     0   0    0    0    0
    6      C    A4-L     0   0    0    0    0

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

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

- M1 . . . . connector number for pulse module 1
  For an axis that uses pulse module 1, enter a connector number using a number in the range of between 1 and the maximum number of axes for pulse module 1. When pulse module 1 need not be used, enter 0. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- M2 . . . . connector number for pulse module 2
  For an axis that uses pulse module 2, enter a connector number using a number in the range of between 1 and the maximum number of axes for pulse module 2. When pulse module 2 need not be used, enter 0. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.
- 1–DSP
  Enter 1 for the following axes, each of which exclusively uses a DSP, which is usually shared by two axes. If a number other than 0 or 1 is entered, the message ***F.54–8*** is displayed.
  - Learning control axis
  - High–speed current loop axis
  - High–speed interface axis
- CS . . Cs contour controlled axis
  Enter 1 for the Cs contour controlled axis. If a number other than 0 or 1 is entered, the message FORMAT ERROR is displayed.
- TNDM
  Enter odd and even numbers for the master and slave axes for tandem control. These numbers must be consecutive and in the range of between 1 and 8. If a number that falls outside the valid range is entered, the message FORMAT ERROR is displayed.

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

- Both M1 and M2 are nonzero for an axis.
- Any two of the 1–DSP, CS, and TNDM are nonzero for an axis.

- A duplicate value is specified for M1.
- A duplicate value is specified for M2.
- A duplicate value is specified for CS.
- A duplicate value is specified for TNDM.
- An invalid master/slave axis pair is specified for TNDM.

Parameter

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

[Data type] Bit

FMD Specifies the FSSB setting mode.

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

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

0 : Incomplete.
1 : Complete.

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

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

[Data type] Bit axis

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

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

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

[Data type] Bit axis

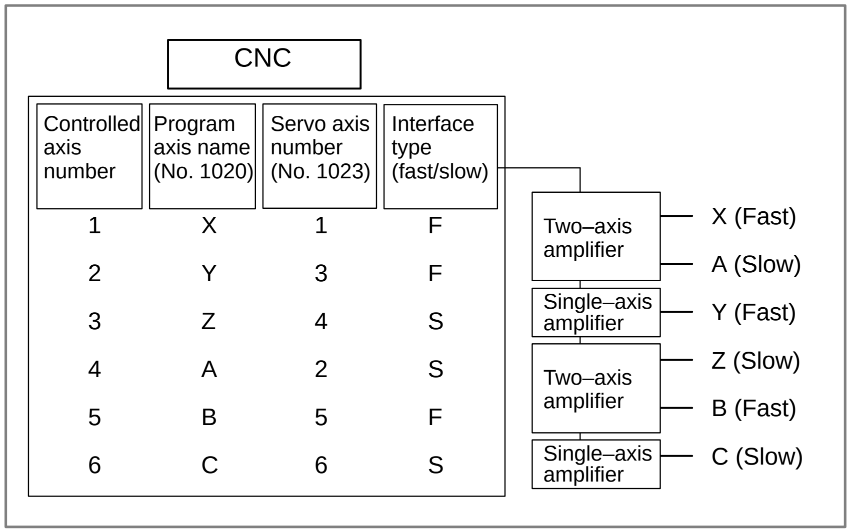
FSL Specifies whether to use a fast or slow interface between a servo amplifier and the servo software.

0 : Fast type
1 : Slow type

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

- Both types are usable for single–axis amplifiers.

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



PM1 Specifies whether the first pulse module is to be used.

0 : Not used.
1 : Used.

PM2 Specifies whether the second pulse module is to be used.

0 : Not used.
1 : Used.

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

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

[Data type] Byte

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

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

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

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

○ When there is no slave:
The parameter is set to 40 except when the simple electronic gearbox (EGB) is used, in which case the following should be observed.

● When the simple electronic gearbox (EGB) is used:
The EGB axis (that axis specified with parameter No. 7771) requires no amplifier. It should be regarded as being connected to a dummy amplifier; that is, the address conversion table value for one of the non–existing slaves should be set to the "value set in parameter No. 1023 for the EGB axis," minus 1, rather than 40.

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

○ Example of axis configuration and parameter setting

CNC

Controlled axis number	Program axis name (No. 1020)	Servo axis number (No. 1023)
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

	Slave number	ATR No.1910 to 1919	Axis
Single–axis amplifier	1	0	X
Two–axis amplifier	2	1	A
	3	2	Y
Two–axis amplifier	4	3	Z
	5	4	B
M1	6	16	(M1)
Single–axis amplifier	7	5	C
M2	8	48	(M2)
	9	40	(None)
	10	40	(None)

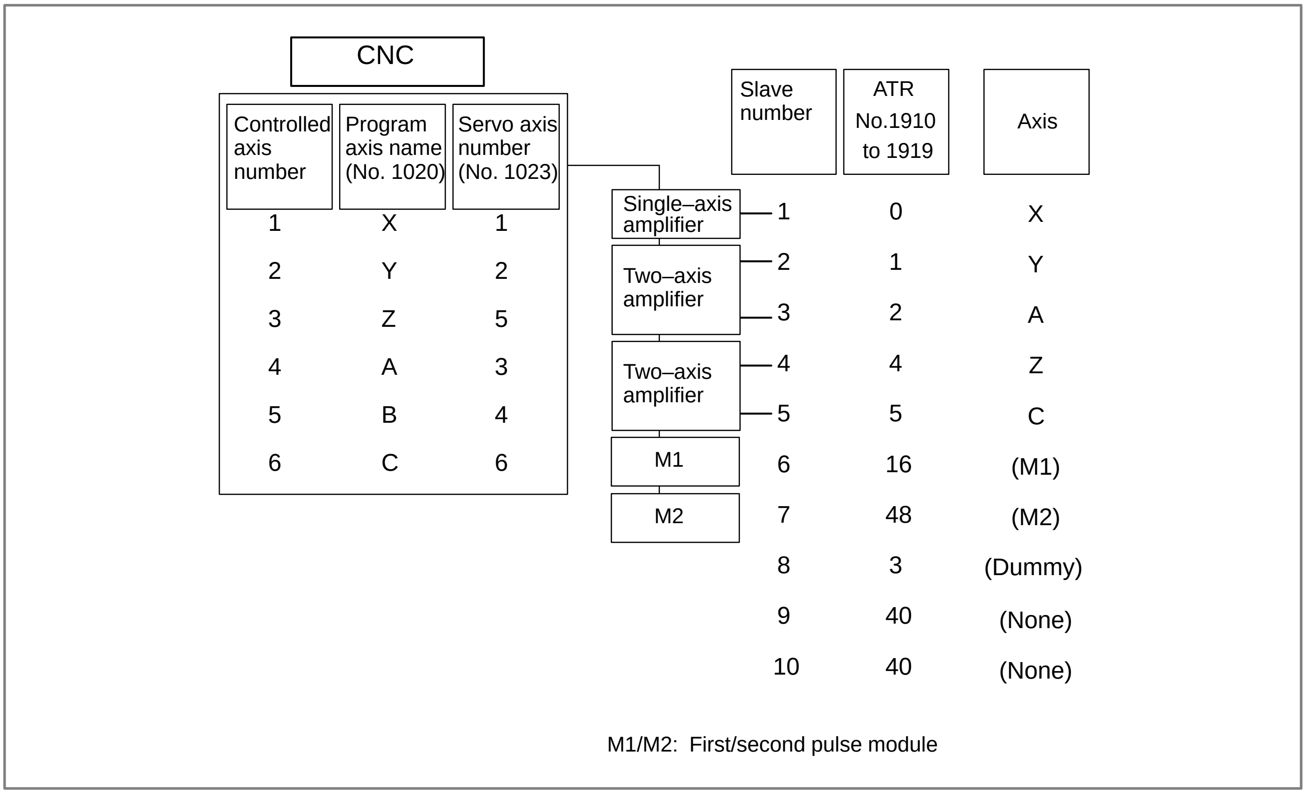
CNC

Controlled axis number	Program axis name (No. 1020)	Servo axis number (No. 1023)
1	X	1
2	Y	3
3	Z	4
4	A	2
5	B	5
6	C	6

	Slave number	ATR No.1910 to 1919	Axis
Single–axis amplifier	1	0	X
Two–axis amplifier	2	2	Y
	3	3	Z
Two–axis amplifier	4	1	A
	5	4	B
M1	6	16	(M1)
Single–axis amplifier	7	5	C
M2	8	48	(M2)
	9	40	(None)
	10	40	(None)

M1/M2: First/second pulse module

○ Example of axis configuration and parameter setting when the simple electronic gearbox (EGB) function is used
(EGB workpiece axis = A–axis; EGB axis = B–axis (parameter No. 7771 = 5))



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

[Data type] Byte

[Valid data range] 0 to 8

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

1931	Connector number for first pulse module (FSSB setting screen only)
1932	Connector number for second pulse module (FSSB setting screen only)

[Data type] Byte axis

[Valid data range] 0 to the number of connectors in each pulse module

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

1933	Cs contour controlled axis (FSSB setting screen only)

[Data type] Byte axis

[Valid data range] 0 and 1

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

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

[Data type] Byte axis

[Valid data range] 0 to 8

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

1936	Connector number for first pulse module
1937	Connector number for second pulse module

[Data type] Byte axis

[Valid data range] 0 to 7

To use a pulse module, this parameter must be set to “connection number for the pulse module,” minus 1; that is, 0 for connector number 1, 1 for connector number 2, and so on. It is also necessary to set up bits 6 and 7 of parameter No. 1905. For an axis that does not use a pulse module, 0 is specified for the parameter. Basically, each axis can be combined with any connector, but the smaller connector numbers must be used first. For example, connector number 4 cannot be used unless connector number 3 is in use.

Example)

Controlled axis	Connector number for first pulse module	Connector number for second pulse module	No.1936	No.1937	No.1905 (#7, #6)
X	1	Not used	0	0	0,1
Y	Not used	2	0	1	1,0
Z	Not used	1	0	0	1,0
A	Not used	Not used	0	0	0,0
B	2	Not used	1	0	0,1
C	Not used	3	0	2	1,0

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

Alarm and message

• Servo alarms

Number	Message	Description
460	n AXIS : FSSB DISCONNECT	FSSB communication was interrupted. The most likely causes are: 1. The FSSB communication cable is disconnected or has a broken conductor. 2. The amplifier power supply was turned off. 3. A low–voltage alarm condition occurred in the amplifier.
461	n AXIS : ILLEGAL AMP INTERFACE	The fast type interface was assigned to both axes of a two–axis amplifier.
462	n AXIS : SEND CNC DATA FAILED	The slave could not receive data correctly because of an FSSB communication error.
463	n AXIS : SEND SLAVE DATA FAILED	The servo section failed to receive data correctly because of an FSSB communication error.
464	n AXIS : WRITE ID DATA FAILED	An attempt to write maintenance information to the amplifier maintenance screen failed.
465	n AXIS : READ ID DATA FAILED	An attempt to read the initial ID information for the amplifier failed when the power was switched on.

Number	Message	Description
466	n AXIS : MOTOR/AMP COMBINATION	The maximum current rating for the amplifier does not match that for the motor.
467	n AXIS : ILLEGAL SETTING OF AXIS	The servo function for the following has not been enabled when an axis occupying a single DSP is specified on the axis setting screen. 1. Learning control (bit 5 of parameter No. 2008 = 1) 2. High–speed current loop (bit 0 of parameter No. 2004 = 1) 3. High–speed interface axis (bit 4 of parameter No. 2005 = 1)

- **P/S alarms**

Number	Message	Description
5134	FSSB : OPEN READY TIME OUT	The FSSB did not become ready to open during initialization.
5135	FSSB : ERROR MODE	The FSSB entered an error mode.
5136	FSSB : NUMBER OF AMPS IS SMALL	The number of amplifiers recognized by the FSSB is insufficient, compared with the number of controlled axes.
5137	FSSB : CONFIGURATION ERROR	The FSSB detected a configuration error.
5138	FSSB : AXIS SETTING NOT COMPLETE	Axis setting has not been performed in automatic setting mode. Perform axis setting using the FSSB setting screen.
5197	FSSB : OPEN TIME OUT	The FSSB did not open when the CNC had allowed the FSSB to open.
5198	FSSB : ID DATA NOT READ	The initial ID information for the amplifier cannot be read because of a failure in the temporary assignment.

1.5 SETTINGS RELATED WITH COORDINATE SYSTEMS

1.5.1 Machine Coordinate System

General

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

Parameter

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

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

Warning

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

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.7.1	MACHINE COORDINATE SYSTEM
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.7.1	MACHINE COORDINATE SYSTEM
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.7.1	MACHINE COORDINATE SYSTEM
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.7.1	MACHINE COORDINATE SYSTEM

1.5.2 Workpiece Coordinate System/Addition of Workpiece Coordinate System Pair

General

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

Setting a workpiece coordinate system

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

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

(2) Automatic setting
If bit 0 (ZPR) of parameter No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed.

This method can be used when no option is specified for the workpiece coordinate system.

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

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.

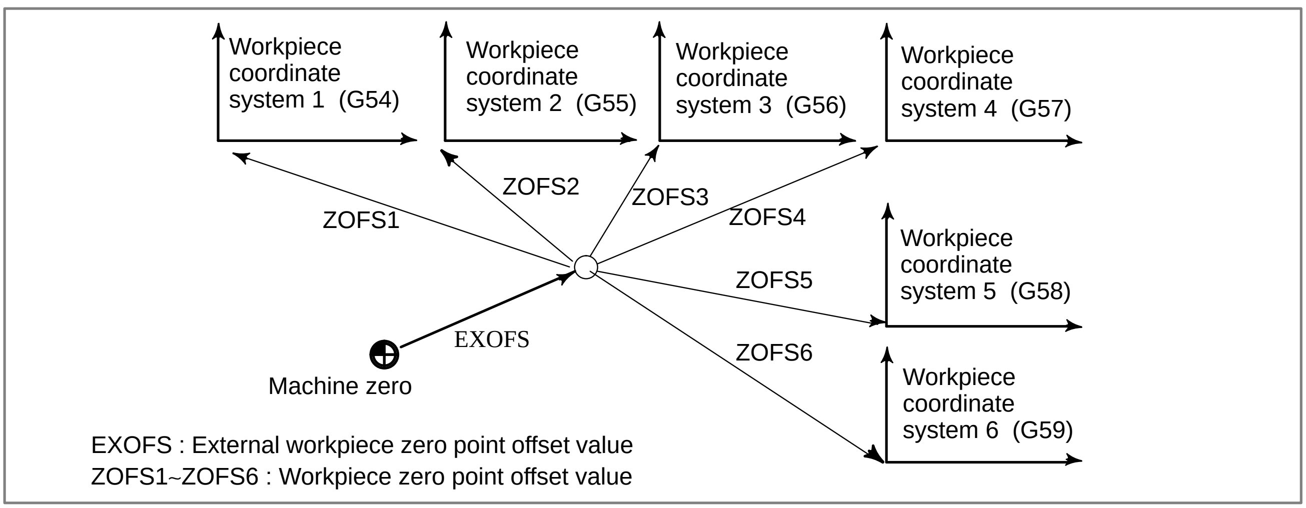
(2) Choosing from six workpiece coordinate systems set using the MDI panel
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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

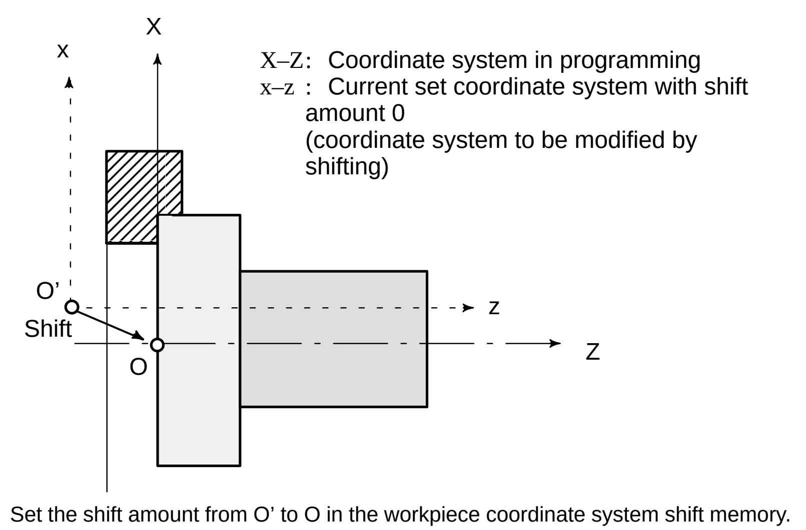
The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.
(1) Inputting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



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

Workpiece coordinate system shift (T series)

When the coordinate system actually set by the G92 (G50) command 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 or 300 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Parameter

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed

0 : Not set automatically
1 : Set automatically

NOTE
If a workpiece coordinate system option is available, automatic coordinate system setting is not performed. Whenever manual reference position return is performed, the workpiece coordinate system is established according to the workpiece origin offset (parameter Nos. 1220 to 1226).

ZPI Coordinates at the reference position when a coordinate system is set automatically

0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

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

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

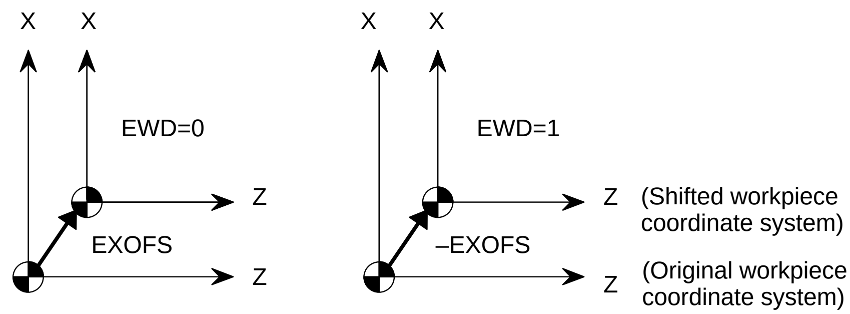
> **NOTE**
> In either case, the change becomes effective when the next block is buffered.

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

[Data type] Bit

EWD The shift direction of the workpiece coordinate system is:

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



EXOFS : External workpiece zero point offset value

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

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

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

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

1220	External workpiece zero point offset value

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

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

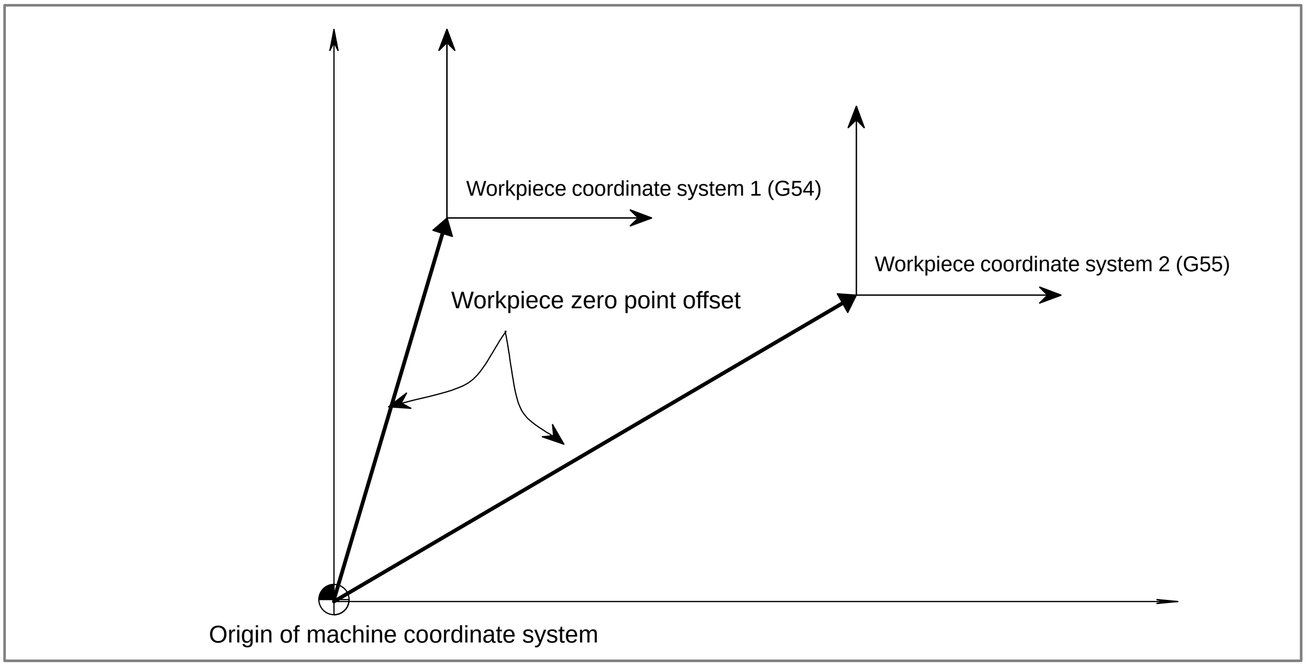
[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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



1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

1251	Coordinate value of the reference position used when automatic coordinate system setting is performed with inch input

[Data type] Two–word axis

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 ZPI in parameter 1201 is set to 1.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.7.2	WORK COORDINATE SYSTEM
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.7.2	WORK COORDINATE SYSTEM
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.7.2	WORK COORDINATE SYSTEM
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.7.2	WORK COORDINATE SYSTEM

1.5.3 Rotary Axis Roll Over

General

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

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

Parameter

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

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

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

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

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

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

[Data type] Bit axis

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

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

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

NOTE
RABx is valid only when ROAx is 1.

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

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

1260	Move amount per rotation of rotary axis

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] 1000 to 9999999
Set move amount per rotation of rotation axis.

Note

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

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.2	ROTARY AXIS ROLL–OVER
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.19.2	ROTARY AXIS ROLL–OVER
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.20.2	ROTARY AXIS ROLL–OVER
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.19.2	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.

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

• Simple synchronous control for the M series and T series

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

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

<M series> The function can synchronize both automatic and manual operations.

The following functions are provided only for the M series:

- Synchronization error check function
- Synchronization compensation function

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

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

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

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

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

WARNING

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

NOTE

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

- **Synchronization compensation function**

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

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

WARNING

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

CAUTION

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

NOTE

1 To use the synchronization compensation function, set the SOF bit, bit 7 of parameter 8301 (if only one master/slave axis pair is in sync) or SOFx bit, bit 7 of parameter 8303 (if more than one master/slave axis pair is in sync), to 1.

2 The synchronization compensation function is also enabled when emergency stop is canceled.

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

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

[Operating procedure]

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

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

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

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

4 Switch the power off then back on.

5 Perform ordinary reference position return.

NOTE

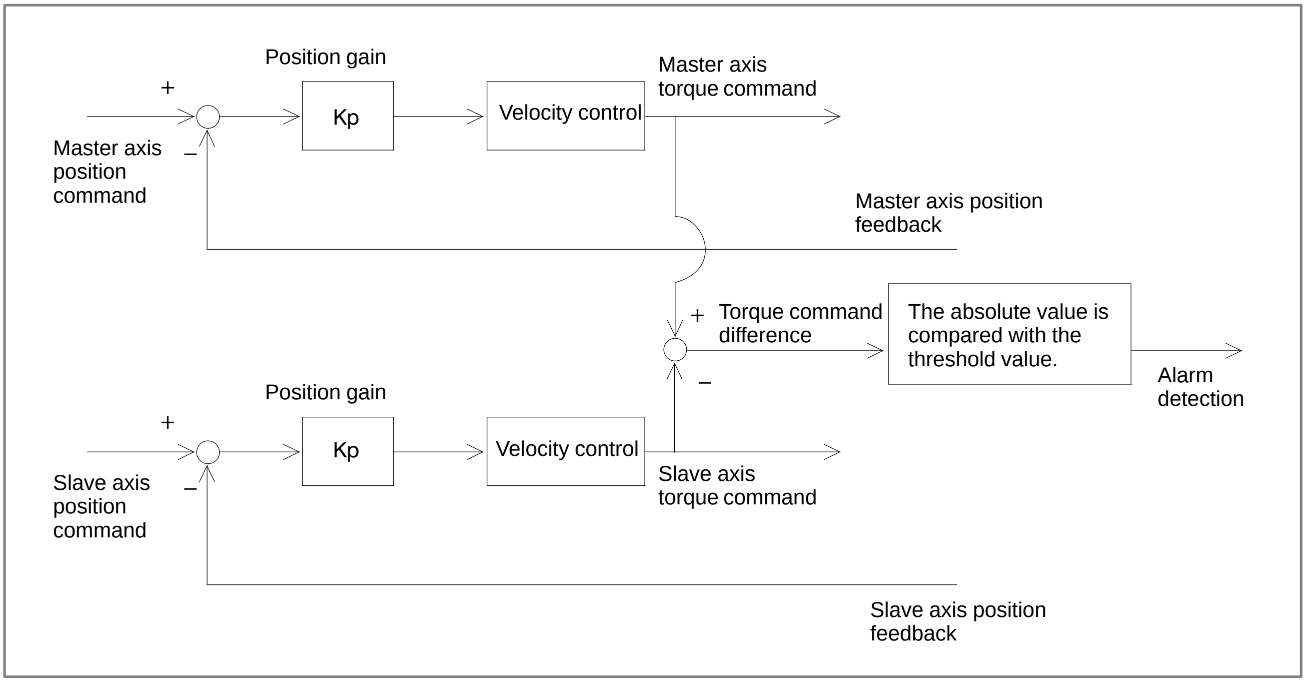
Parameter setting

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

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

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

[System configuration]



[How to use]

Determine the threshold parameter using the following procedure.

1 Set up the following parameters:
Parameter No. 2031 = 0 : Disable torque difference alarm detection.
Bit 0 of parameter No. 8950 = 1 : Display the memory screen.
Parameter No. 8960 : Specify the following values as data addresses.
1304 if the servo axis numbers for two axes in sync are 1 and 2
1312 if the servo axis numbers for two axes in sync are 3 and 4
1800 if the servo axis numbers for two axes in sync are 5 and 6
1808 if the servo axis numbers for two axes in sync are 7 and 8
Parameter No. 2115 : Display, on the memory screen, the absolute value of the torque difference between the two axes that are in sync. This parameter No. must be set to the same value for both axes.
Value: 22160

2 Display the memory screen using:
[SYSTEM] function key → [▷] soft key → [MEMORY] soft key
The absolute value of the torque difference between the two axes that are in sync is displayed as No.1C0 on the memory screen.

3 Read the maximum torque difference absolute value during ordinary operation, and convert it to decimal. The threshold parameter should be specified with a margin.

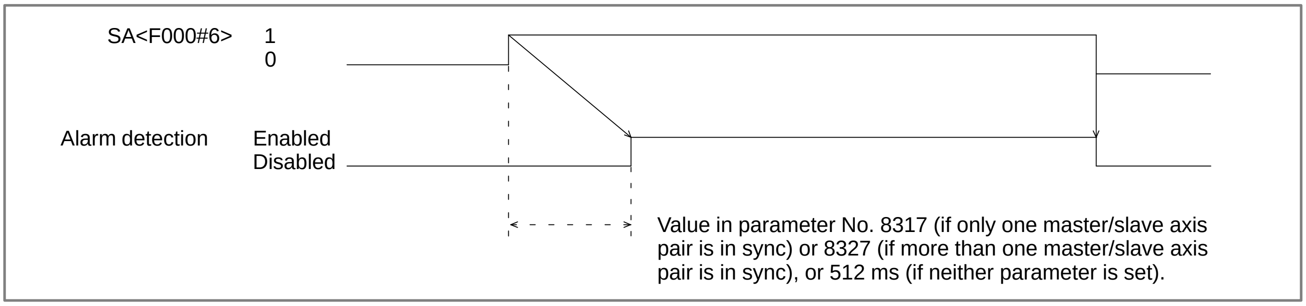
If it is difficult to read the absolute torque difference value on the memory screen it can be observed, using an oscilloscope by means of the following procedure:

4 Set parameter No. 2115 to 22160.

5 Connect a check board, and observe a signal at CH7 with the rotary switch on the check board set to 1.

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

[Timing chart]



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

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

Signal

<T series and M series>

Signals to select the slave axis for simple synchronous control SYNC1 to SYNC8 <G138>

[Classification] Input signal

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

SYNC 1

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

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

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

The master axis is specified with a parameter.

<M series>

Signals for selecting the manual feed axis for simple synchronous control SYNCJ1 to SYNCJ8 <G140>

[Classification] Input signal

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

SYNCJ 1

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

:
:

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

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

The master axis is specified with a parameter.

Signal address

T series

	#7	#6	#5	#4	#3	#2	#1	#0
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1

M series

	#7	#6	#5	#4	#3	#2	#1	#0
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
G140	SYNCJ8	SYNCJ7	SYNCJ6	SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1

Parameter

T series

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 to 7

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

Units digit in the parameter for the first axis
→ Set the master axis number for the first axis.
Tens digit in the parameter for the first axis
→ Set the master axis number for the second axis.
Units digit in the parameter for the second axis
→ Set the master axis number for the third axis.
Tens digit in the parameter for the second axis
→ Set the master axis number for the fourth axis.
Units digit in the parameter for the third axis
→ Set the master axis number for the fifth axis.
Tens digit in the parameter for the third axis
→ Set the master axis number for the sixth axis.
Units digit in the parameter for the fourth axis
→ Set the master axis number for the seventh axis.
Tens digit in the parameter for the fourth axis
→ Set the master axis number for the eighth axis.

Number	Tens digit	Units digit
First	Second axis	First axis
Second	Fourth axis	Third axis
Third	Sixth axis	Fifth axis
Fourth	Eighth axis	Seventh axis

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

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

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

8312	Enabling/disabling mirror image in synchronous control

[Data type] Byte axis

[Valid data range] –128 to +127

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

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

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

M series

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

[Data type] Bit

SMF During simple synchronous contorol, movement along a slave axis is:

0 : Included in the actual speed display
1 : Not included in the actual speed display

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

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

[Data type] Bit

SOF The synchronization compensation funciton under simple synchronous control (one master/slave axis pair) is:

0 : Not used.
1 : Used.

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

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

[Data type] Bit

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

ATS Specify whether to start the automatic setting of grid positioning under simple synchronous control (one master/slave axis pair)
0 : Not started
1 : Started

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

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

[Data type] Bit axis

ATEx Specify whether to enable the automatic setting of grid positioning under simple synchronous control (more than one master/slave axis pair)
0 : Disabled
1 : Enabled

ASTx Specify whether to start the automatic setting of grid positioning under simple synchronous control (more than one master/slave axis pair)
0 : Not started
1 : Started

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

SOFx Specify whether to enable synchronization compensation during simple synchronous control (more than one master/slave axis pair)
0 : Disabled
1 : Enabled

8311	Axis number of master axis in synchronous control

[Data type] Byte axis

[Valid data range] 0 to 7

Select a master axis and slave axis for simple synchronous control. Set a master axis number for the slave axis side.

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

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

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

To specify the:
master axis as the 1st axis, and the slave axis as the 6th axis
master axis as the 2nd axis, and the slave axis as the 5th axis
master axis as the 3rd axis, and the slave axis as the 4th axis,
set the following:
Parameter No. 8311 X(1st axis) = 0
Y(2nd axis) = 0
Z (3rd axis) = 0
(4th axis) = 3
(5th axis) = 2
(6th axis) = 1

NOTE

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

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

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

8314	Allowable error in synchronization error check

[Data type] Word axis

[Unit of data]

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

[Valid data range] 0 to 32767

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

8315	Maximum compensation value for synchronization compensation

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

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

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

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

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

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

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

[Data type] Word

[Unit of data] ms

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

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

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

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

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

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

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

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

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

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

8327	Torque difference alarm detection timer (more than one master/slave axis pair under synchronous control)

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

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

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

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

Alarm and message

T series

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

M series

Number	Message	Description
213	ILLEGAL COMMAND IN SYNCHRO–MODE	One of the following errors occurred during synchronous operation (simple synchronous 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 8313 or 8323.

Servo alarm

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

Diagnostic data

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

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.
3 If control of more than one master/slave axis pair is specified in parameter No. 8311, parameters to perform setting for, and display diagnostic information about, only one master/slave axis pair are invalid, and if control of only one master/slave axis pair is specified, parameters to perform setting for, and display diagnostic information about, only one master/slave axis pair are valid (except for common parameter Nos. 8311 and 8314).

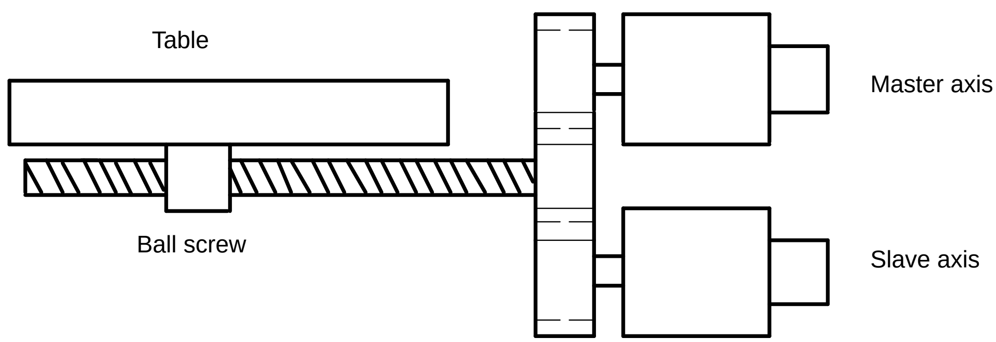
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.1	Simple synchronous control
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.19.3	Simple synchronous control
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.20.1	Simple synchronous control
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.19.3	Simple synchronous control

1.7 TANDEM CONTROL

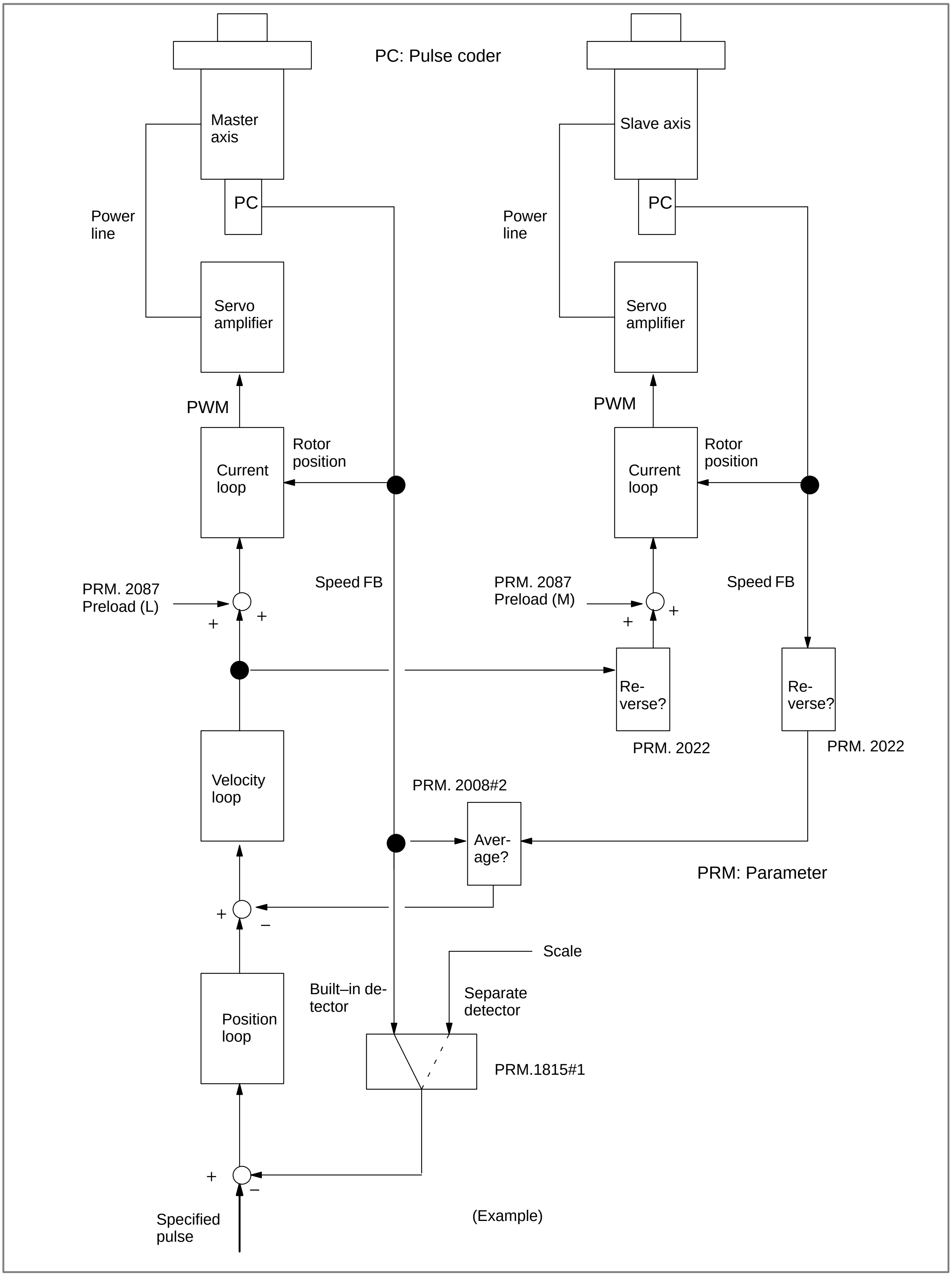
General

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



Sample application

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



Block Diagram of Tandem Control

Explanations

• Axis configuration in tandem control

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

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

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

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

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

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

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

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

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

The following sample axis configuration is for a machine with six axes X, Y, Z, A, B (PMC axis), and C. The X–axis and Y–axis are the master axes of tandem control.

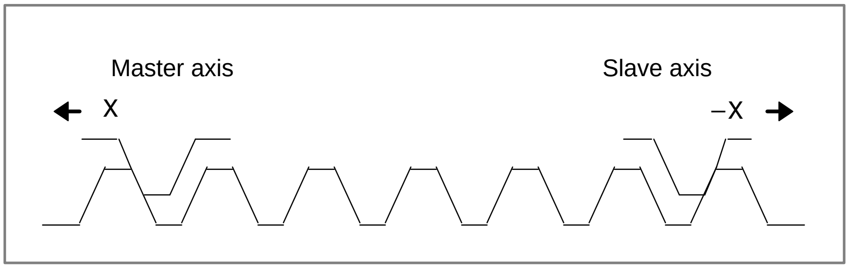
Number of controlled axes (optional parameter): 6

Number of CNC controlled axes (parameter 1010): 3

Axis number	Axis name	Servo axis number (PRM. 1023)	
1	X	3	CNC axis (master axis of tandem control)
2	Y	1	CNC axis (master axis of tandem control)
3	Z	6	CNC axis
4	A	4	Slave axis of tandem control (master axis: X–axis)
5	B	5	PMC axis
6	C	2	Slave axis of tandem control (master axis: Y–axis)

- **Preload function**

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



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

- **Velocity feedback average function**

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

- **Improved stability of a closed–loop system**

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

· Dual position feedback function
· Machine velocity feedback function

For details of these functions, refer to FANUC AC SERVO MOTOR α series PARAMETER MANUAL (B–65150E).

- **Notes on stability of tandem control**

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

- **Connection of axis signals**

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

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

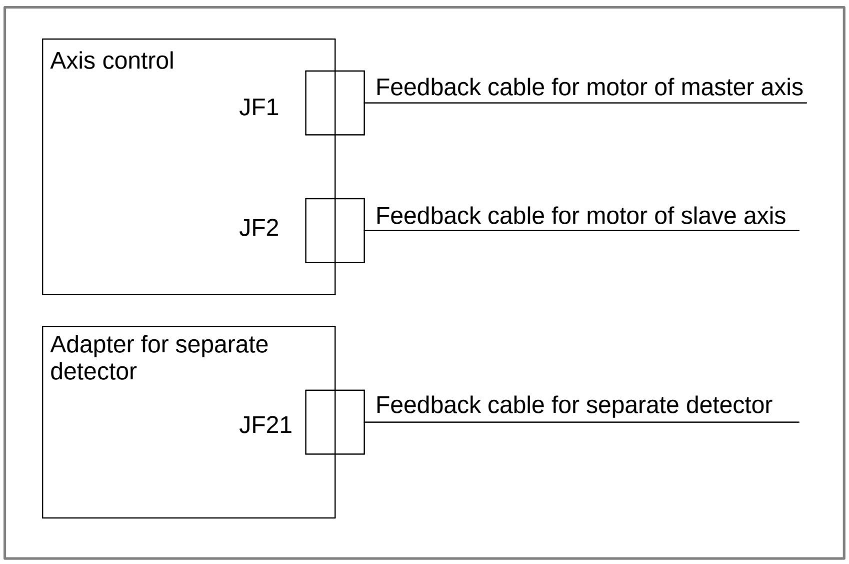
ii) Overtravel limit signal
Connect the signal so that 1 is always output as the overtravel limit signal for the slave axis.

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

- **Connecting motors**

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

(Sample connection for position feedback cable)



- **Servo alarms**

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

Parameter

Setting data (parameters)

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

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

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

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

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

WARNING

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

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

Parameter 2022: Direction of rotation of the motor

Parameter 2087: Preload value

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

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

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

Bit 1 of parameter 1815: Separate pulse coder

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

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

Parameter 1821: Capacity of an optional reference counter

Parameter 2084: Numerator of flexible feed gear ratio

Parameter 2085: Denominator of flexible feed gear ratio

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

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

Parameters that should be set to only the master axes

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

Parameters that should be set different values between the master and slave axes

Parameter No.	Meaning of parameters
1020	Axis name
1023	Servo axis number
2022	Motor rotation direction
2087	Preload value
3115	Current position display
1816#6 to #4	Detection multiplier (DMR)
1821	Arbitrary reference counter capacity
2024	Position detection feedback pulses (PPLS)
2084	Numerator of flexible feed gear ratio
2085	Denominator of flexible feed gear ratio

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

Parameter No.	Meaning of parameters
1005#0	Movement before reference position return
1005#1	Dogless reference position setting
1006#0	Rotary axis
1006#1	Machine coordinate of rotary axis is rotary type
1006#3	Diameter/radius specification
1006#5	Direction of reference position return
1240	Reference position as viewed from machine zero
1241	Coordinate of 2nd reference position
1242	Coordinate of 3rd reference position
1243	Coordinate of 4th reference position
1260	Move distance per rotation of rotary axis
1310#0	Soft OT2
1310#1	Soft OT3
1320	1st stroke limit of plus side
1321	1st stroke limit of minus side
1322	2nd stroke limit of plus side
1323	2nd stroke limit of minus side
1420	Rapid traverse rate
1421	F0 of rapid traverse override
1620	Time constant of rapid traverse linear acceleration/deceleration
1621	Time constant of rapid traverse bell shaped acceleration/deceleration
1622	Time constant of feed exponential acceleration/deceleration
1623	FL of feed exponential acceleration/deceleration
1624	Time constant of manual continuous exponential acceleration/deceleration
1625	FL of manual continuous exponential acceleration/deceleration
1626	Time constant of exponential acceleration/deceleration during thread cutting cycle
1627	FL of exponential acceleration/deceleration during thread cutting cycle
1820	Command multiplier (CMR)
18XX	Digital servo parameters
20XX	Digital servo parameters

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

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

TAN Tandem control is
1 : ignored
0 : valid

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

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

VFBAVE Velocity feedback average function
1 : invalid
0 : valid

2087	Preload of each axis (Tcmd offset)

[Data type] Word axis

[Unit of data] (Preamplifier limit) /7282

[Valid data range] –1821 to 1821

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

[Example] To set a torque of 3A in the opposite direction under amplifier limit of 40A:
3 / (40 / 7282) = 546
Master side = 546
Slave side = –546

2021	Load inertia

[Data type] Word axis

Set the same value to the master and slave axes.

[Unit of data] (All load inertia) / (Motor inertia) × 256 / 2

2022	Direction of rotation of motor

[Data type] Word axis

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

Alarm and message

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

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.4	Tandem control

1.8 SYNCHRONOUS CONTROL (T SERIES)

General

This function enables synchronous control, in which an axis can be synchronized with another axis.

An axis can be moved in synchronization with another axis. This is done by issuing a move command for one axis (synchronous master axis) to another axis so that both axes behave in the same way. When this function is used in conjunction with the parking function, which keeps an axis at a rest by ignoring a move command for it, the following operations can be performed.

(1) Two axes move in synchronization. (Both master and slave axes move.)

(2) One axis moves according to a move command originally issued to another axis, but the latter is kept at a stop. (The master parks, but the salve moves.)

(3) The coordinate values for one axis are updated according to the amount of movement specified for another axis, but the former is kept at a rest. (The master moves, but the slave parks.)

CAUTION

1 The term synchronous control used here only refers to an operation in which issuing the same move command to two different servo systems at one time. Note that synchronous control does not involve out–of–synchronization compensation, in which the deviation between two servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation.

2 The term parking means to issue no move command to a servo system. No coordinate values are updated during parking. However appropriate parameter setting can change absolute and relative coordinate systems.

3 If synchronous control is terminated during automatic operation, a move command or coordinate system setting may not be specified for the synchronous slave axis in the current block and two blocks that follows it (or three blocks for tool–nose radius compensation).

Automatic setting of workpiece coordinate system

• Explanation

When synchronous control is started for a workpiece coordinate system, it is possible to specify the workpiece coordinate system automatically. When synchronous control for a workpiece coordinate system is terminated, it is possible to return the workpiece coordinate system to ordinary machining (not synchronous control). The explanation of the workpiece coordinate system used during synchronous control follows: For example, when synchronous control is used to move an axis differently from the way originally specified, the master axis may be placed in a parking state, while the slave axis is allowed to move. In such a case, it will be convenient if a coordinate system used to represent the current position of the slave axis is used as a workpiece coordinate system for the master axis. Conventionally, this workpiece coordinate system must be specified by program when synchronous control is started, because the workpiece coordinate system does not originally belong to the master axis. This automatic workpiece coordinate system setting function for synchronous control sets up this workpiece coordinate system automatically. This function can also resume the original workpiece coordinate system for the master axis automatically. This function does not work for the slave axis.

• Setting and command

In addition to setting ordinary synchronous control, parameters must be specified as follows:

(1) To set up a workpiece coordinate system for synchronous control automatically when starting synchronous control
Set parameter No. 8163 (SPMx) to “1”. (Master axis parameter only)
Set parameter No. 8185 with the coordinates of the master axis when both master and slave axes are at the reference position.

(2) To resume the ordinary workpiece coordinate system automatically when synchronous control is terminated
Set parameter No. 8163 (SPSx) to “1”. (Master axis parameter only)
Set parameter No. 1250 with the workpiece coordinates of the master axis when the master axis is at the reference position.

This synchronous control can be specified using the G0138 signal similarly to the ordinary synchronous control. When the signal is raised to start synchronous control, the workpiece coordinate system for the master axis is automatically set up. Likewise, when the signal is dropped to terminate synchronous control, the original workpiece coordinate system for the master axis is set automatically.

• Workpiece coordinate value calculation method

(1) Workpiece coordinate system for synchronous control

(Master axis workpiece coordinate value)
= (parameter No. 8185 for the master axis)
± (slave axis machine coordinate value) . $\boxed{1}$
+ (master axis machine coordinate value) $\boxed{2}$

$\boxed{1}$ + :Master axis parameter SCDx = 0
− :Master axis parameter SCDx = 1
$\boxed{2}$ Master axis parameter SCMx = 1 only

(2) Workpiece coordinate system for ordinary operation

(Master axis workpiece coordinate value)
= (parameter No. 1250 for the master axis)
+ (master axis machine coordinate value)

- **Others**

- If many slave axes are synchronized with one master axis, the master axis is set with the workpiece coordinate system that corresponds to the current position of the first slave axis that is synchronized with the master axis.
- The same least command and input increments must be applied to both master and slave axes.
- The tool offset is taken into consideration when the coordinate system is set up. So, the coordinate system is set up normally even when tool geometry compensation is applied.
- If synchronous control that involves automatic coordinate system setting is started or terminated during automatic operation, a move command or other commands that reference the current position of an axis may not be specified in the current block and two blocks that follows it (or three blocks for tool–nose radius compensation), except for an M code in which the current block is not buffered; for the M code, a move command can be executed in the block next to the block that involves synchronous control.

Signal

Synchronous control axis selection signals SYNC1 to SYNC8 <G138>

[Classification] Input signal

[Function] These signals perform synchronous control.

[Operation] When one of these signals becomes “1”, the control unit:

- Begins synchronous control in such a way that the corresponding axis becomes a slave axis.

The axis with which the slave axis is synchronized is determined by parameter No. 8180.

Parking signals PK1 to PK8 <G122>

[Classification] Input signal

[Function] These signals place each axis in a parking state.

[Operation] When one of these signals becomes “1”, the control unit:

- Places the corresponding axis in a parking state.

If the corresponding axis is under synchronous control, it enters a parking state immediately regardless of whether the axis is moving. If a parking signal is set to “1” without specifying synchronous control, it is ignored.

Synchronous control under way signals SYN1O to SYN8O <F118>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous control.

[Operation] These signals become “1” under the following condition:

- The corresponding axis is under synchronous control.

These signals become “0” under the following condition:

- The corresponding axis is not under synchronous control.

CAUTION

Whether each axis is under synchronous control does not always match whether the corresponding selection signal (synchronous control axis selection signal) has been issued or not. For example, if these signals are set to “1” during an alarm, they are ignored. If a servo alarm occurs during synchronous control, it is terminated automatically. Before attempting to perform synchronous control, always check the state of these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0138					SYNC4	SYNC3	SYNC2	SYNC1
G0122	PK8	PK7	PK6	PK5	PK4	PK3	PK2	PK1

	#7	#6	#5	#4	#3	#2	#1	#0
F0118					SYN4O	SYN3O	SYN2O	SYN1O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS							

[Data type] Bit

NRS When the system is reset, synchronous, composite, or superimposed control is:

0 : Released.
1 : Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8162						PKUx		SMRx

[Data type] Bit axis

SMRx Synchronous mirror–image control is:

0 : Not applied. (The master and slave axes move in the same direction.)
1 : Applied. (The master and slave axes move in opposite directions.)

PKUx In the parking sate,

0 : The absolute, relative, and machine coordinates are not updated.
1 : The absolute and relative coordinates are updated. The machine coordinates are not updated.

	#7	#6	#5	#4	#3	#2	#1	#0
8163				SCDx	SCMx	SPSx	SPMx	

[Data type] Bit axis

Set the parameters SPMx, SPSx, SCMx, and SCDx for the master axis. These settings are referenced during automatic workpiece coordinate setting for the master axis at the start of synchronous control.

SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is

0 : Not performed.
1 : Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the start of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates of each axis at the reference position set in parameter No. 8185.

SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master axis is:

0 : Not performed.
1 : Performed.

> **NOTE**
> When a workpiece coordinate system is automatically set at the end of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates for each axis at the reference position set in parameter No. 1250.

SCMx When workpiece coordinates are calculated in synchronous control:

0 : The workpiece coordinates are calculated from the machine coordinates of the slave axis.
1 : The workpiece coordinates are calculated from the machine coordinates of the master axis and slave axis.

SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:

0 : Identical.
1 : Opposite.

8180	Master axis with which an axis is synchronized under synchronous control

[Data type] Byte axis

[Valid data range] 201, 202, 203, ... to 200 plus the maximum number of control axes

This parameter specifies the number of the master axis with which an axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more axes, one master axis has two or more slave axes.

In the parameter of a slave axis, specify 200 plus the number of the master axis with which the slave axis is to be synchronized.

Setting: 201 to 208

The value specified here must not exceed 200 plus the maximum number of control axes.

[Example] Synchronizing the Y–axis with the Z–axis

Parameter No. 8180x 0
Parameter No. 8180z 0
Parameter No. 8180c 0
Parameter No. 8180y 202

8185	Workpiece coordinates on each axis at the reference position

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

This parameter sets the workpiece coordinates on each master axis, subject to synchronous control, when the master and slave axes are at the reference position. This parameter is enabled when SPMx of bit 1 parameter No. 8163 is set to 1. Set this parameter for the master axis.

Alarm and message

Number	Message	Description
225	Axis recomposition error	This alarm occurs under either of the following conditions. (It is detected when synchronous control is specified.) (1) There is an error in axis number parameter setting. (2) There is an error in a control command.
226	A move command was issued to a synchronous axis.	A move command was issued to an axis to be synchronized with another axis.

Warning, Caution, and Note for synchronous control

WARNING

1 When synchronous control is started or terminated, the target axes must be at a stop.
2 All axes subjected to synchronous control must have the same least command increment, detection unit, and diameter/radius specification. Otherwise, the amount of movement will differ from one axis to another.
3 During synchronous control, do not change the parameters related to synchronous control.
4 Before starting synchronous control after an emergency stop, servo–off, or servo alarm is released, be sure to make a return to the reference position and set up the necessary coordinate system.
5 Before starting synchronous control, make sure that for the target axes, a reference position return after power–on has been made and a reference position has been set up according to the absolute pulse coder.
6 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are performed independently of synchronization control.
7 The program cannot issue a move command to the slave axis under synchronization control.
8 The same acceleration/deceleration time constants and servo parameters should be used for the axes subjected to synchronous control as much as possible.
9 The workpiece coordinate system of a synchronous slave is not affected by the synchronous master axis operations that affect the workpiece coordinate system but do not cause the machines to move, such as workpiece coordinate system set/shift and geometry offset commands.
10 If a wear offset command or tool–tip radius compensation is performed for a synchronous master axis, the travel path of the slave axis is shifted by the offset, but the offset is not set (no offset vector is created).
11 When using parking signal PK7 or PK8 while both spindle synchronous control and simple spindle synchronous control are being applied, set the SPK bit (bit 7 of parameter No. 4800) to 1. This sets parking signals PKESS1 and PKESS2, used for simple spindle synchronous control, to #6 and #7 of G031.

CAUTION

1 Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are carried out regardless of synchronous control.
2 A move command should not be issued to a synchronous slave axis during synchronous control.

NOTE

1 More than one axis can be placed in synchronous state at the same time. However, an axis cannot be synchronized with more than one axis simultaneously.
2 Synchronous control and simplified synchronous control cannot be specified simultaneously.

Restrictions imposed during synchronous control

Function	During synchronous control
Acceleration/deceleration control	The same type of acceleration/deceleration control is performed for the synchronous axes, but different time constants are used.
Linear acceleration/deceleration after cutting feed interpolation	Not carried out for any axes during synchronous control.
Feedrate clamping	The axes are clamped at the feedrate of the master axis.
Reference position return	A reference position return is possible for the master axis unless it is in a parking state. In a parking state, only automatic reference position return (G28) is possible for the master axis (*1).
Reference position return check	Possible (*2)
PMC axis control	Possible for other than synchronous slave axes.
Polar coordinate interpolation and cylindrical interpolation	Possible
Handle interrupt	Performed regardless of synchronous control
Axial mirror image	Each signal originally belonging to a particular axis is effective for that axis.
Machine lock	Each signal originally belonging to a particular axis is effective for that axis.
Interlock	The signals for the master axis are effective for the slave axes.
Override	The signals for the master axis are effective for the slave axes.
External deceleration	The signals for the master axis are effective for the slave axes.
Skip function	Ineffective for slave axes.
Automatic tool compensation	Ineffective for slave axes.
Tool setter	Ineffective for slave axes.
Follow–up	Impossible during synchronous control.
Program restart	Impossible for a program involving synchronous control.
Cs axis	Synchronous control is impossible.

WARNING

If a reference position return command is issued for a synchronous master axis during synchronous control, it is executed normally for the master axis, but the slave axis does not return to their reference position (the slave axis only moves in synchronization with the reference position return of the master axis) except for an automatic reference position return (G28) issued when the master axis is in a parking state, in which case the amount of movement is calculated so that the slave axis returns to its reference position provided that a reference position return has been carried out for the slave axis. If more than one slave axis belongs to one master axis, a reference position return command is executed so that the lowest–numbered slave axis returns to its reference position. A return to the second (third or fourth) reference position by the G30 command works in the same way as G28.

CAUTION

If a reference position return check (G27) is specified during synchronous control, the master and slave axes move to the specified position. Upon completion of axis movement, a check is made to see whether the master axis is at its reference position (no check is made for the slave axes) unless the master axis is in a parking state, in which case a check is made to see whether the lowest–numbered slave axis is at its reference position after completion of positioning.

Reading the coordinate values during synchronous control

The following list summarizes how positional information such as custom macro system variables and current coordinates from the PMC window are read during synchronous control.

Positional information	During synchronous control
Absolute coordinate	Readable
Machine coordinate	Readable
End of each block	Readable only for the master axis
Skip signal position	Readable only for the master axis

Terminating synchronous control

Synchronous control is terminated not only when the corresponding synchronization signal becomes off but also when one of the following conditions occurs.

(1) Emergency stop
(2) Reset
(3) Servo alarm
(4) Servo off
(5) Overtravel
(6) Alarm related to synchronous control
(7) P/S000 alarm

The above conditions terminate synchronous control for all axes.

Reference item

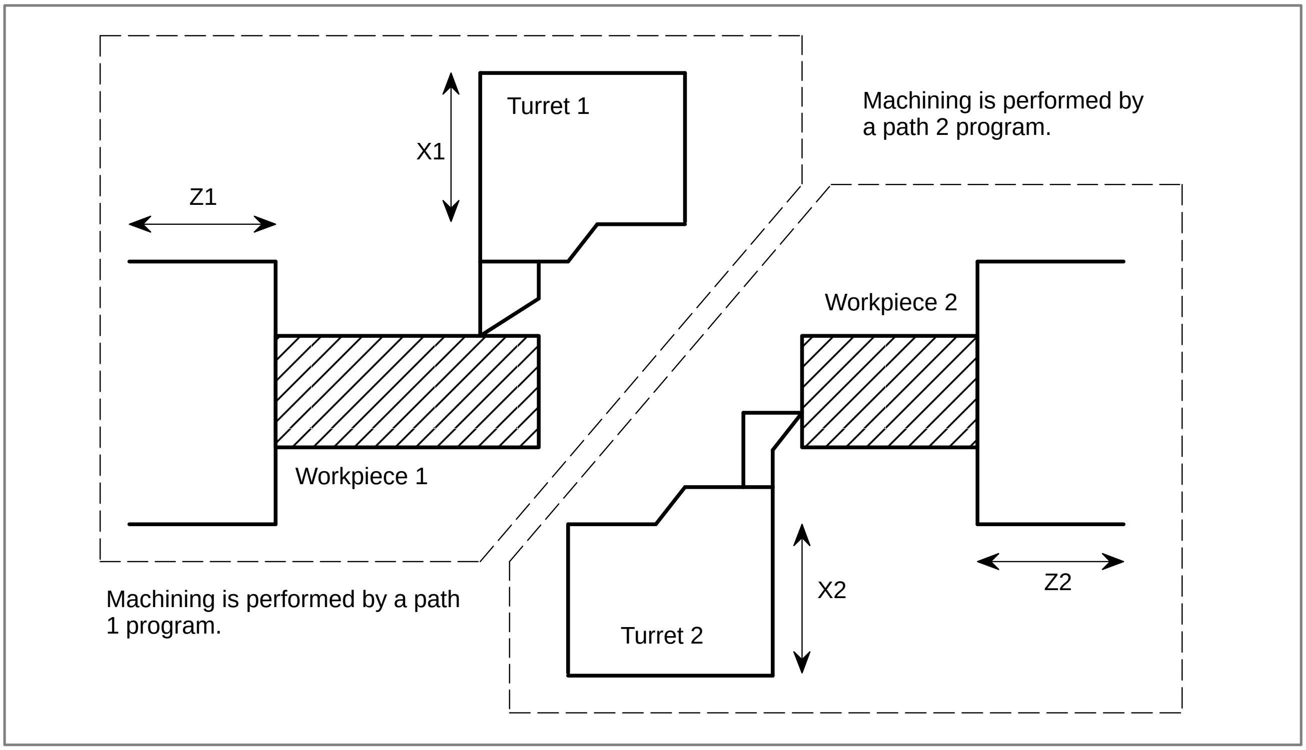
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.19.4	Synchronous control

1.9 SYNCHRONOUS CONTROL AND COMPOSITE CONTROL (T SERIES (TWO–PATH CONTROL))

1.9.1 Overview

The T series CNC has two independent control paths. For example, it can be used to control two turrets of a multiple–turret lathe independently. The axes (such as X1–and Z1–axes) belonging to path 1 are controlled by commands in path 1, and the axes (such as X2– and Z2–axes) belonging to path 2 are controlled by commands in path 2.

- **Independent control in each path**

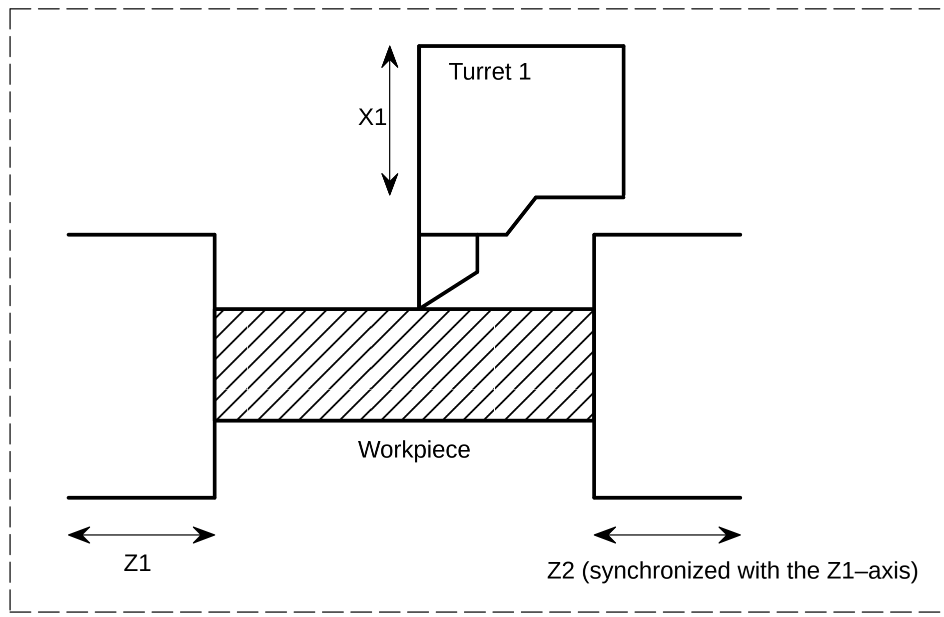


This function enables synchronous control between paths or within a path, composite control between paths, and superimposed control between paths, as explained below.

(1) Synchronous control

- Synchronization of an axis in one path with an axis in the other path

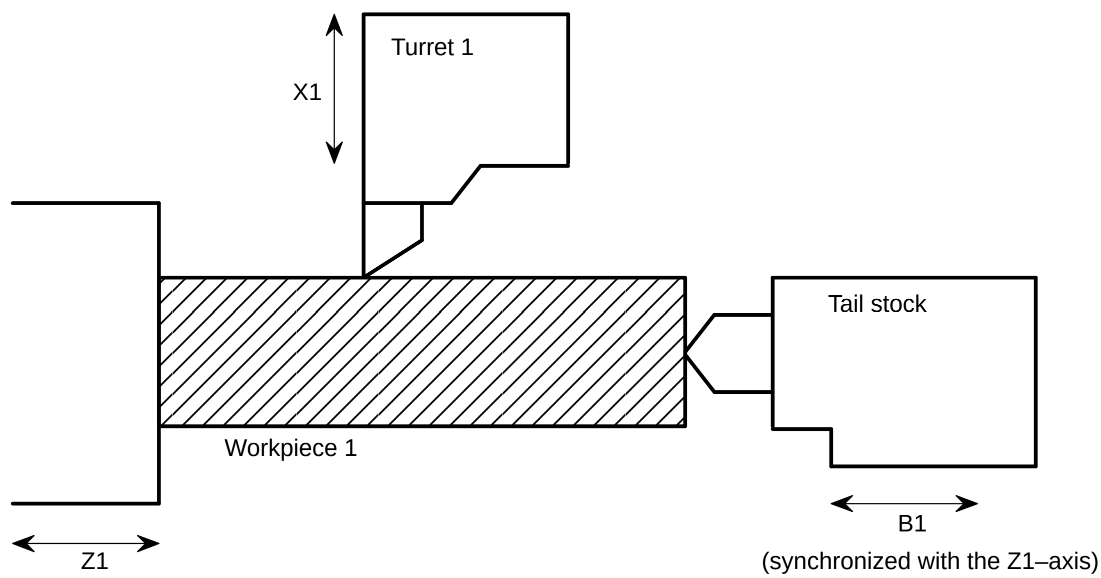
(Example) Synchronization of the Z1–axis with the Z2–axis



Machining is performed by a path 1 program.

- Synchronization of an axis in one path with another axis in the same path

(Example) Synchronization of the Z1–axis with the B1–axis

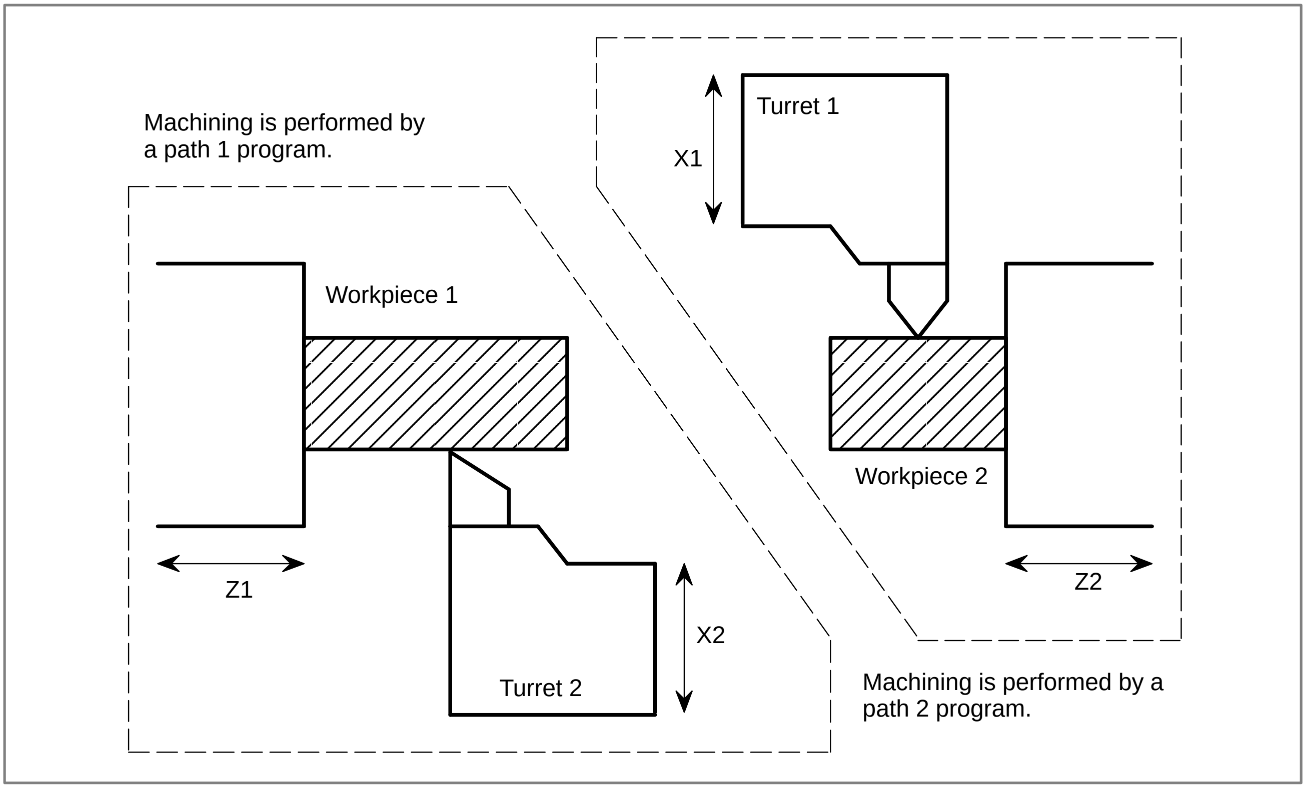


(2) Composite control

- Interchanging move commands for an axis in one path with those for an axis in the other path.

(Example) Interchanging commands between the X1– and X2–axes

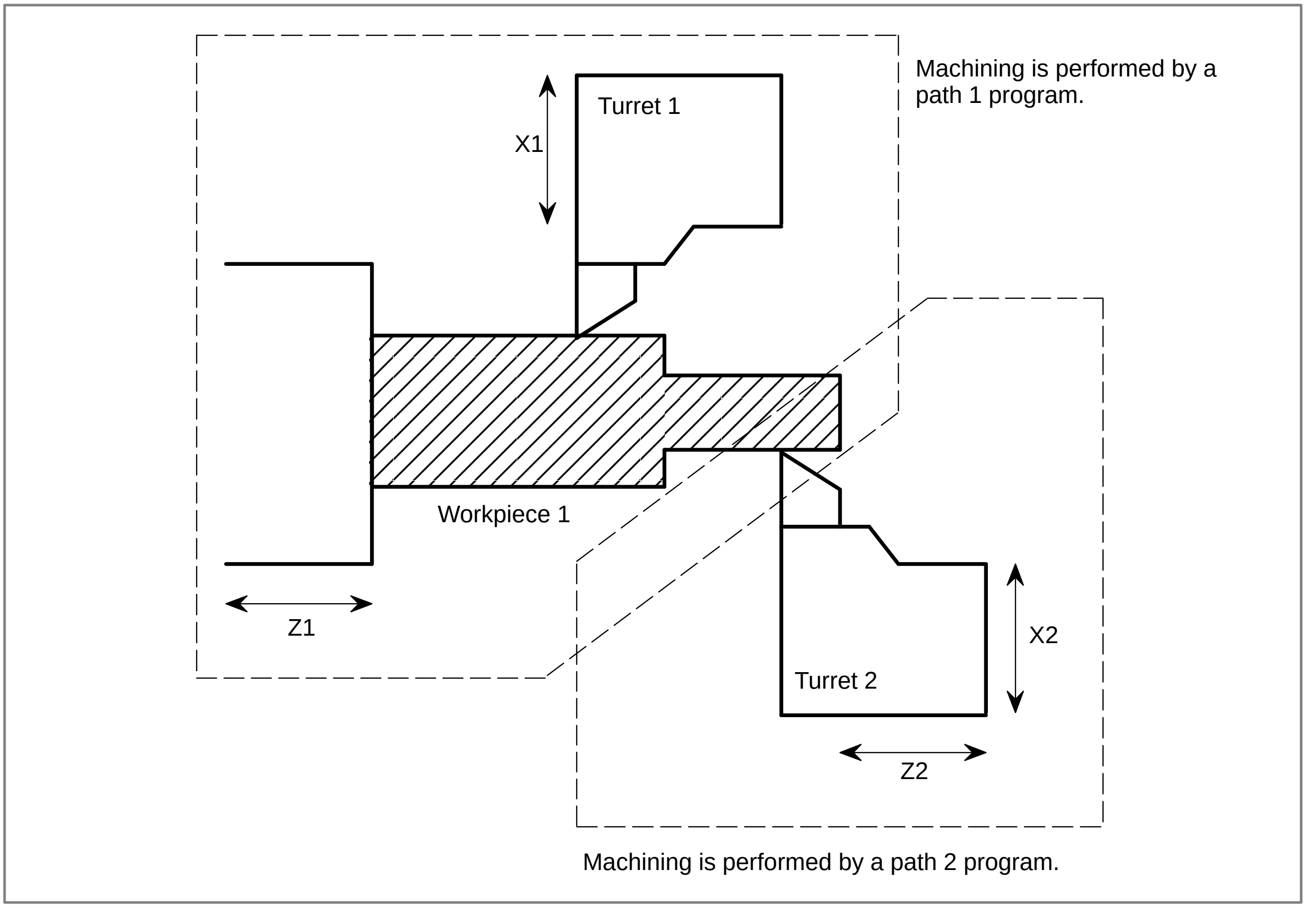
→ Control both X2– and Z1–axes by commands in a path 1 program
Control both X1– and Z2–axes by commands in a path 2 program



(3) Superimposed control

- Superimposing move commands for an axis in one path on an axis in the other path

(Example) Superimposing the movement of the Z1–axis on the Z2–axis



1.9.2 Synchronous Control

An axis in one path can be synchronized with another axis in the same path or an axis in another path. This is done by issuing the same move commands for one axis (synchronous master axis) to another axis (synchronous slave axis). Using parameter SMRx (bit 0 of parameter No. 8162) can cause the slave axis to move in the direction opposite to that of the master axis. It is possible to place either the master or slave axis in a parking state. The term parking here means to discontinue giving move commands to a servo system. No coordinates are updated in the parking state. Note however that absolute and relative coordinates can be updated using parameter PKUx (bit 2 of parameter No. 8162).

1.9.2.1 Setting

Parameter No. 8180 specifies which axis is to be synchronized with which axis.

(Example)
To synchronize the Z1–axis with the Z2–axis:
Parameter No. 8180z of path 1 = 2
To synchronize the Y2–axis with the X1–axis:
Parameter No. 8180y of path 2 = 1
To synchronize the Y1–axis with the X1–axis:
Parameter No. 8180y of path 1 = 201

1.9.2.2 Programming

Use M codes for wait, beginning, and terminating synchronization in a machining program in the stated order. It is also possible to begin and terminate synchronous control without using M codes.

1.9.2.3 Signal operation

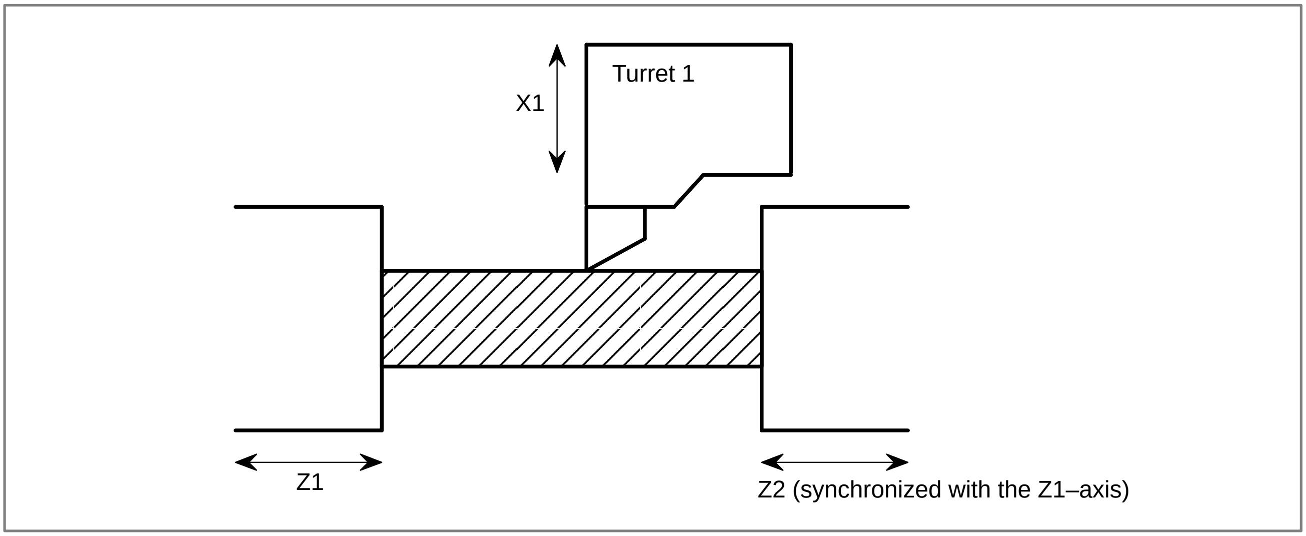
When synchronization begins or ends (when an M code is issued, for example), the synchronous control axis selection signals SYNC1 to SYNC7 for the slave axis (from the PMC to the CNC) are changed from “0” to “1” (to begin synchronization) or from “1” to “0” (to terminate synchronization). To place an axis in a parking state, a parking signal PK1 to PK7 is set to “1” for the target axis.

1.9.2.4 Examples of applications

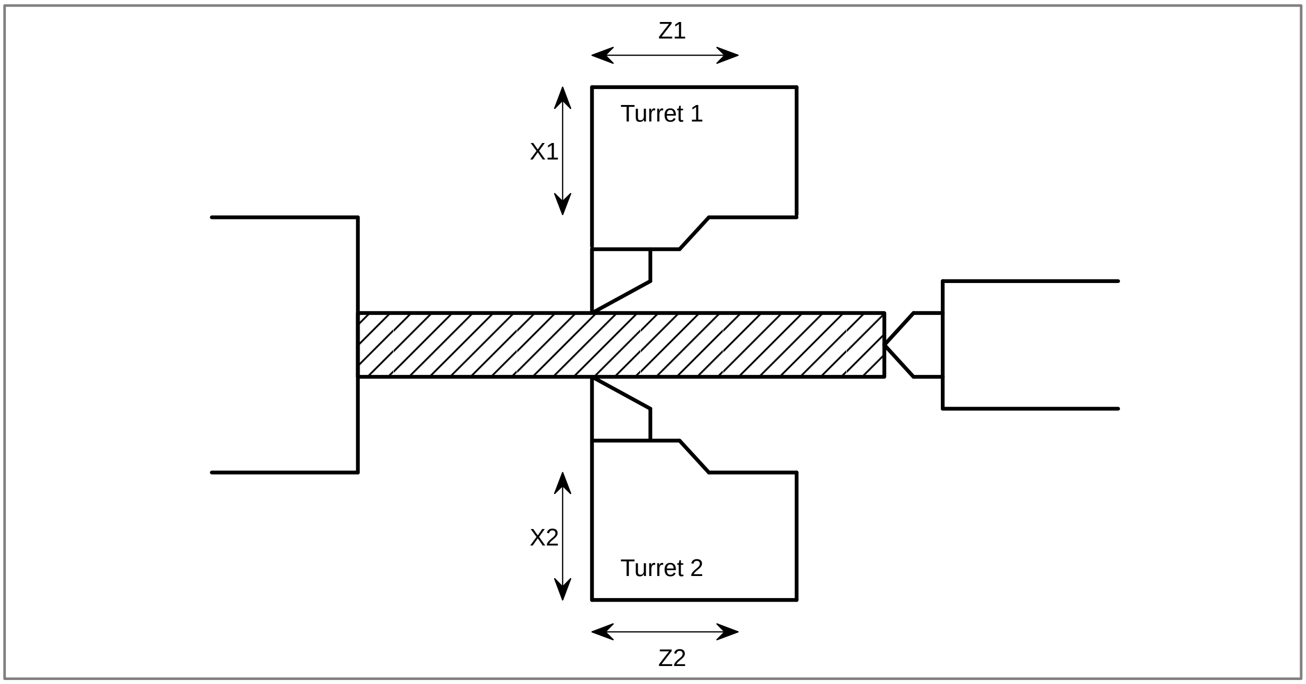
The following operations can be performed by using the synchronization functions together with the parking function, which causes move commands for an axis to be ignored and keeps the axis at a rest.

(1) Moving an axis in one path in synchronization with an axis in the other path (Both master and slave axes move.)

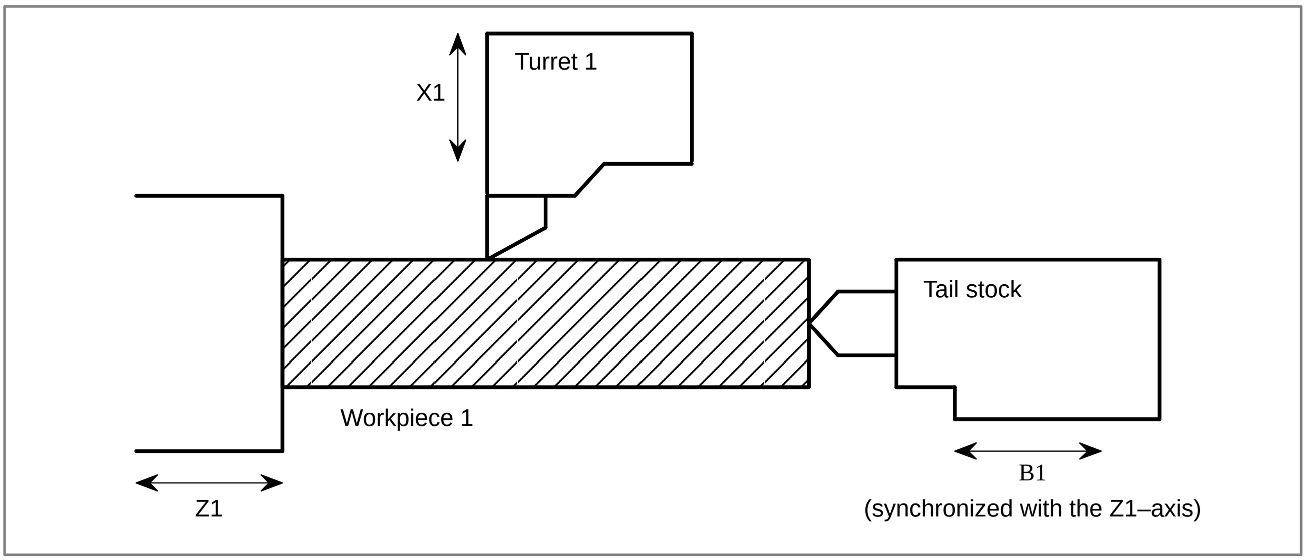
(Example 1) Synchronizing the Z2–axis with the Z1–axis (machining with both ends of a workpiece chucked)



(Example 2) Synchronizing the X2– and Z2–axes with the X1– and Z1–axes (balanced cutting)



(Example 3) Synchronizing the B1–axis (tail stock axis) with the Z1–axis



(2) Moving an axis in one path using move commands for an axis in the other path (The master axis parks and the slave axis moves.)

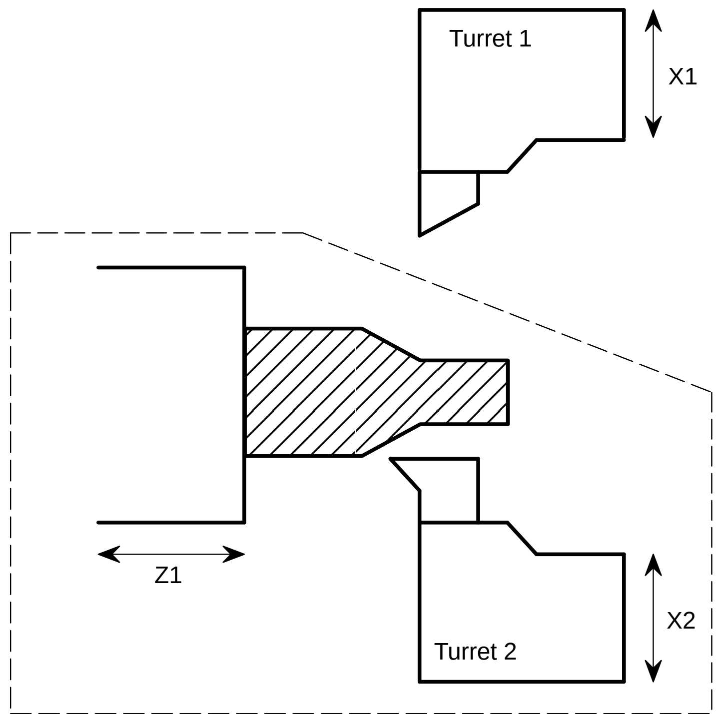
(3) Updating the coordinates of an axis in one path by the amount of movement for an axis in the other path (The master axis moves and the slave axis parks.)

Using method (2) or (3) can control one motor from both paths.

(Example 4) Sharing one motor with the Z1– and Z2–axes (assuming that the motor is linked to the Z1–axis)

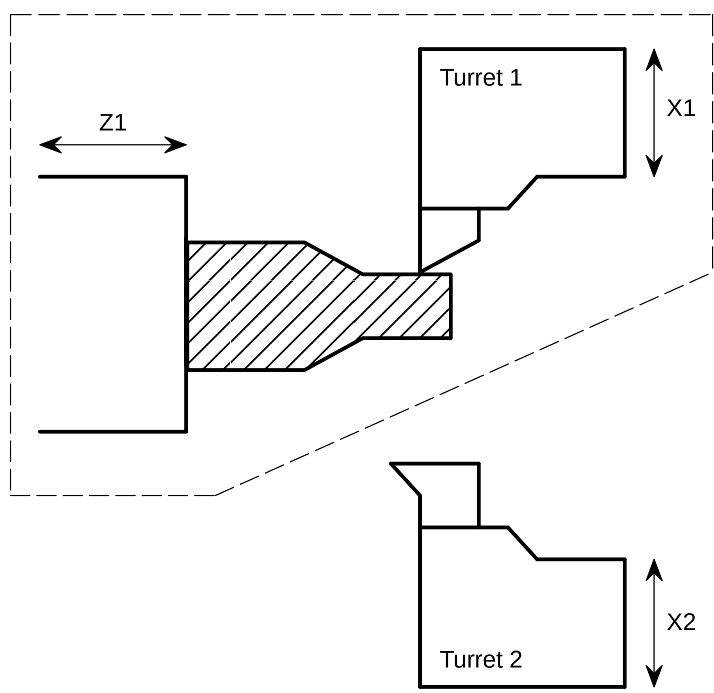
- **Master axis parking**

The X2– and Z1–axes are moved by commands in a path 2 program (by synchronizing the Z1–axis with the Z2–axis). At this point, the Z2–axis is kept in a parking state. The coordinates of both Z1– and Z2–axes are updated.



- **Slave axis parking**

The X1– and Z1–axes are moved by commands in a path 1 program (by synchronizing the Z2–axis with the Z1–axis). At this point, the Z2–axis is kept in a parking state. For the Z2–axis, only the coordinates are updated.



Because the coordinates of both Z1– and Z2–axes are updated, move commands can be executed immediately, without resetting up the coordinate system, when the synchronization state is switched.

1.9.2.5 Reference position return and its check during synchronous control

If a reference position return command is issued for a synchronous master axis during synchronous control, it is executed normally for the master axis, but the slave axis does not return to its reference position (the slave axis only moves in synchronization with the reference position return of the master axis) except for an automatic reference position return (G28) issued when the master axis is in a parking state, in which case the amount of movement is calculated so that the slave axis returns to its reference position provided that a reference position return has been carried for the slave axis. If more than one slave axis belongs to one master axis, a reference position return command is executed so that the lowest–numbered slave axis returns to its reference position. If the master axis in one path is subjected to both synchronization with an axis in the same path and synchronization with an axis in the other path simultaneously, the lowest–numbered slave axis in the two paths is moved to the reference position.

A return to the second (third or fourth) reference position by the G30 command works in the same way as G28. In other words, usually only the master axis moves to the second (third or fourth) reference position. If the master axis is parking, the lowest–numbered axis is caused to move to its second (third or fourth) reference position.

If a reference position return check (G27) is specified during synchronous control, the master and slave axes move to the specified position. Upon completion of movement, a check is made to see whether the master axis is at its reference position (no check is made for the salve axes) unless the master axis is in a parking state, in which case a check is made upon completion of positioning to see whether the lowest–numbered slave axis is at its reference position.

1.9.2.6 Out–of–synchronization detection

The term synchronous control used here only refers to an operation in which the same move command is issued to two different servo systems at one time. Note that synchronous control does not involve out–of–synchronization compensation, in which the positional deviation between two servo motors is constantly checked and one of the servo motors is subjected to compensation to reduce the deviation. However, using parameter SERx (bit 1 of parameter No. 8162) can specify detection of out–of–synchronization.
If out–of–synchronization is detected, synchronous control is immediately terminated, turning off the servo ready signal.

CAUTION
Out–of–synchronization detection is not applied to synchronous control within one path.

1.9.2.7 Move command after switching between independent control and synchronous control

If synchronous control is terminated during automatic operation, do not issue a move command or coordinate system setting for the synchronous slave axis in the current block and one or two (during tool–nose radius compensation) subsequent blocks. This restriction is intended to reflect the coordinates changed during synchronous control in the preprocessing for the subsequent blocks.

(Example) Terminating synchronous control of the Z–axis (slave axis) in block N200

N190 ..... ;
N200 M55 ; (This M code terminates synchronous control.)
N210 ..... ;
N220 ..... ;
N230 ..... ;

In this example, block N210 (and N220 during tool–nose radius compensation) cannot issue a move command to the Z–axis. However, if the M55 code does not involve buffering, it can be issued in block N210 to move the Z–axis or update its coordinates. For other than the Z–axis, block N210 can issue move commands. These restrictions do not apply to the synchronous master axis.

1.9.2.8 Automatic setting of a workpiece coordinate system

When synchronous control is started in a workpiece coordinate system, it is possible to specify the workpiece coordinate system automatically. When synchronous control for a workpiece coordinate system is terminated, it is possible to return the workpiece coordinate system to ordinary machining (not synchronous control). The explanation of the workpiece coordinate system used during synchronous control follows. When synchronous control is used to move an axis differently from the way originally specified, for example, the master axis may be placed in a parking state, while the slave axis is allowed to move. In such a case, it will be convenient if a coordinate system that indicates the current position of the slave axis is used as a workpiece coordinate system for the master axis. Conventionally, this workpiece coordinate system must be specified by program when synchronous control is started, because the workpiece coordinate system does not originally belong to the master axis. This automatic workpiece coordinate system setting function for synchronous control sets up this workpiece coordinate system automatically. This function can also resume the original workpiece coordinate system for the master axis automatically. This function does not work for the slave axis.

- **Setting and commands**

In addition to setting ordinary synchronous control, parameters must be specified as follows:

(1) To set up a workpiece coordinate system for synchronous control automatically when starting synchronous control
Set parameter SPMx (bit 1 of parameter No. 8163) to “1”.
Set parameter No. 8185 with the coordinates of the slave axis reference position relative to the coordinates of the master axis when the master axis is at the reference position.

(2) To resume the ordinary workpiece coordinate system automatically when terminating synchronous control

Set parameter SPSx (parameter No. 8163) to “1”.
Set parameter No. 1250 with the master axis coordinates in the workpiece coordinate system when the master axis is at the reference position.

This synchronous control can be specified using the synchronous control axis selection signal (SYNC1 to SYNC7) similarly to the ordinary synchronous control. When the signal is raised to start synchronous control, a workpiece coordinate system for the master axis is automatically set up. When the signal is dropped to terminate synchronous control, the original workpiece coordinate system for the master axis is resumed automatically.

- **Workpiece coordinate calculation method**

(1) Workpiece coordinate system for synchronous control

(Master axis workpiece coordinate value)
= (parameter No. 8185 for the master axis)
± (slave axis machine coordinate value) .... $\boxed{1}$
+ (master axis machine coordinate value) ...... $\boxed{2}$

$\boxed{1}$ +: Master axis parameter SCDx (bit 4 of parameter No. 8163) = 0
–: Master axis parameter SCDx (bit 3 of parameter No. 8163) = 1

$\boxed{2}$ Master axis parameter SCMx (bit 3 of parameter No. 8163) = 1 only

(2) Workpiece coordinate system for ordinary operation

(Master axis workpiece coordinate value)
= (parameter No. 1250 for the master axis)
+ (master axis machine coordinate value)

Caution

CAUTION

1 The same least command and input increments must apply to both master and slave axes.
2 If synchronous control that involves automatic coordinate system setting is started or terminated during automatic operation, a move command or other commands that reference the current position of an axis may not be specified in the current block and one or two (during tool–nose radius compensation) blocks, except when the M code in the current block does not involve buffering.

Note

NOTE

1 If more than one slave axis is synchronized with one master axis, the master axis is set with the workpiece coordinate system that corresponds to the current position of the first slave axis that is synchronized with the master axis.

2 The tool offset is taken into account when the coordinate system is set up. So, the coordinate system is set up normally even when tool geometry compensation is applied.

1.9.3 Composite Control

Move commands can be interchanged between an axis in one path and an axis in the other path. In other words, when a machining program is executed for one path, actual machining can be performed with an axis in the other path. Coordinate systems can also be switched automatically between independent control and composite control.

1.9.3.1 Setting

Parameter No. 8183 of path 2 specifies the axes between which commands are to be interchanged.

(Example) Between the X1– and X2–axes:
Parameter No. 8183x of path 2 = 1
Between the Y1– and X2–axes:
Parameter No. 8183x of path 2 = 4
Between The X1– and Y2–axes:
Parameter No. 8183y of path 2 = 1

To set up coordinate systems automatically when composite control begins or ends, set parameters MPMx and MPSx (bits 4 and 5 of parameter No. 8162) to ”1”, and specify the positional relationship between the coordinate systems in parameter No. 8184.

1.9.3.2 Programming

Use M codes for wait, beginning, and terminating composite control in a machining program in the stated order. It is also possible to begin and terminate composite control without using M codes.

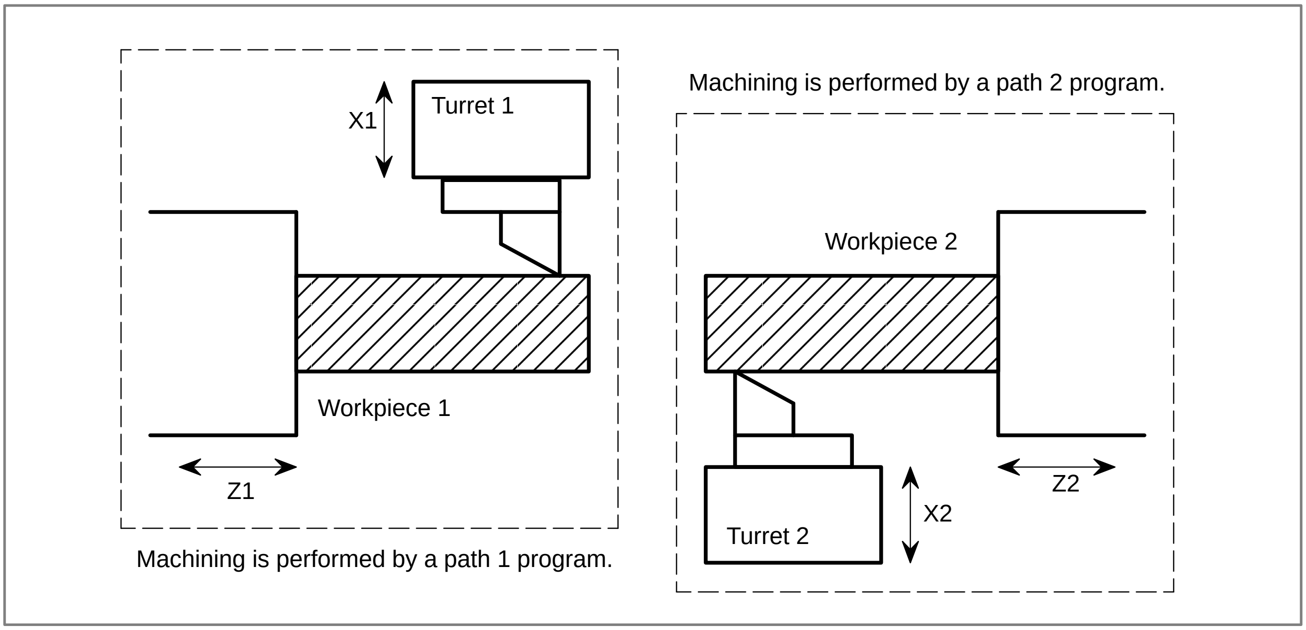
1.9.3.3 Signal operation

When composite control begins or ends (when an M code is issued), the composite control axis selection signals MIX1 to MIX7 for the target axis in path 1 (from the PMC to the CNC) are changed from “0” to “1” (to begin composite control) or from “1” to “0” (to end composite control).

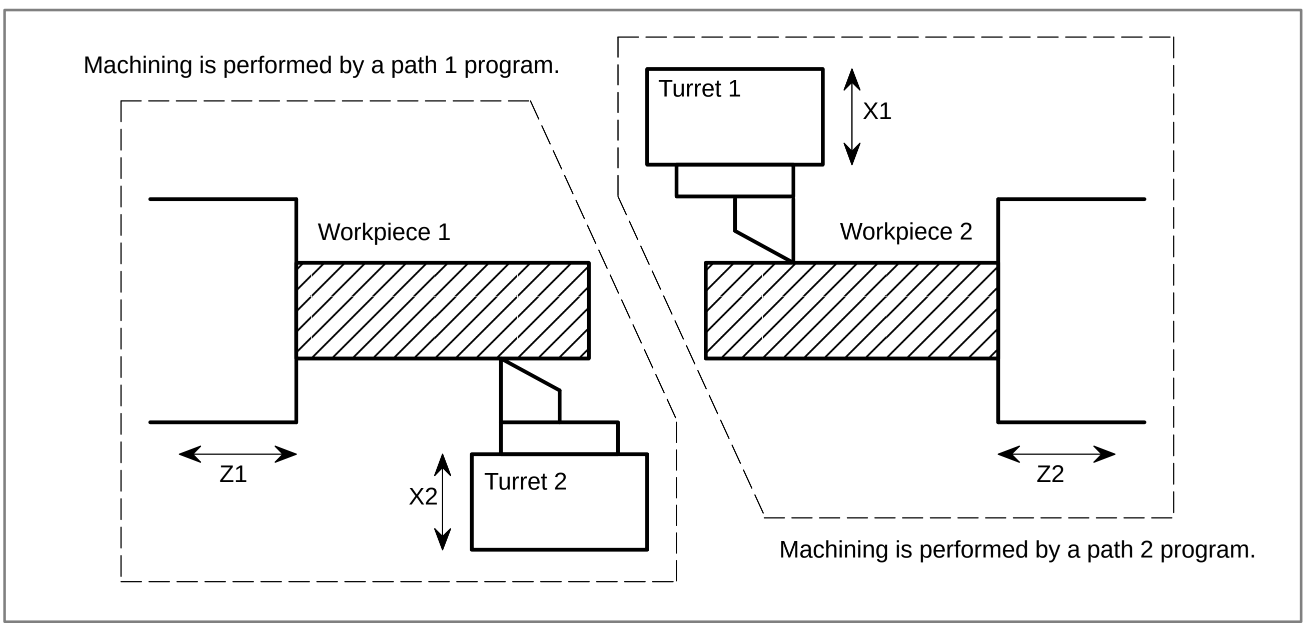
1.9.3.4 Examples of applications

Suppose that a machine has the X1– and Z1–axes belonging to path 1 and the X2– and Z2–axes belonging to path 2 and that a workpiece moves along the Z1– and Z2–axes as directed by move commands. The following examples interchange commands between the X1– and X2–axes.

(1) Independent control



(2) Composite control



During composite control, the X2– and Z1–axes are moved by a path 1 program, and the workpiece coordinates of the X–axis in path 1 indicates the position of turret 2. Similarly, the X1– and Z2–axes are moved by a path 2 program, and the workpiece coordinates of the X–axis in path 2 indicates the position of turret 1.

1.9.3.5 Spindle control

The composite control function does not switch the spindle speed command or the feed per rotation command based on feedback pulses from the position coder. Therefore, the spindle speed command and feedback pulses should be switched using the following signals. (See Section 9.4.2 for details.)

- Spindle command selection signal SLSPA <G063#2> and SPSPB <G063#3>
- Spindle feedback selection signal SLPCA <G064#2> and SLPCB <G064#3>

1.9.3.6 Tool offset during composite control

A preset offset or tool–nose radius compensation is not changed when the control mode is switched between independent control and composite control. It is necessary to reset the offset using a T code after the control mode is switched.

1.9.3.7 Reference position return during composite control

If G28 is issued to specify an automatic reference position return for an axis in one path during composite control, an amount of movement is calculated so that the associated axis in the other path can move to the reference position. In this case, the reference position for that axis must have already been established. A manual reference position return is not allowed.

1.9.3.8 Move commands after the control mode is switched between independent control and composite control

If the control mode is switched between independent control and composite control during automatic operation, do not issue a move command or coodinate system setting for the switched axis in the current block and one or two (during tool–nose radius compensation) subsequent blocks. This restriction is intended to reflect the coordinates changed due to coordinate system setting during control mode switching in the preprocessing for the subsequent blocks.

(Example) Starting composite control to switch between the X1– and X2–axes in block N200

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

In this example, block N210 (and N220 during tool–nose radius compensation) cannot issue a move command to the X–axis. However, if the M55 code does not involve buffering, it can be issued in block N210 to move the X–axis or update its coordinates. For other than the X–axis, block N210 can issue move commands.

1.9.4 Superimposed Control

The superimposed control function adds the amount of movement of an axis (superimposed control master axis) in one path to an axis (superimposed control slave axis) on the other path for which ordinary move commands are being executed. This function is similar to synchronous control but differs from it in that move commands can be issued not only for the master axis but also for the slave axis. The slave axis moves by the sum of the amount of movement specified by its own move commands and the amount of movement specified by move commands for the master axis. Appropriate setting of parameter OMRx (bit 3 of parameter No. 8162) can reverse the direction in which the master and slave axes move.

1.9.4.1 Setting

Parameter No. 8186 specifies between which axes move commands are to be superimposed.

(Example) To superimpose the amount of movement of the Z2–axis to that of the Z1–axis: Parameter No. 8186z of path 1 = 2
To superimpose the amount of movement of the X1–axis to that of the Y2–axis: Parameter No. 8186y of path 2 = 1

1.9.4.2 Programming

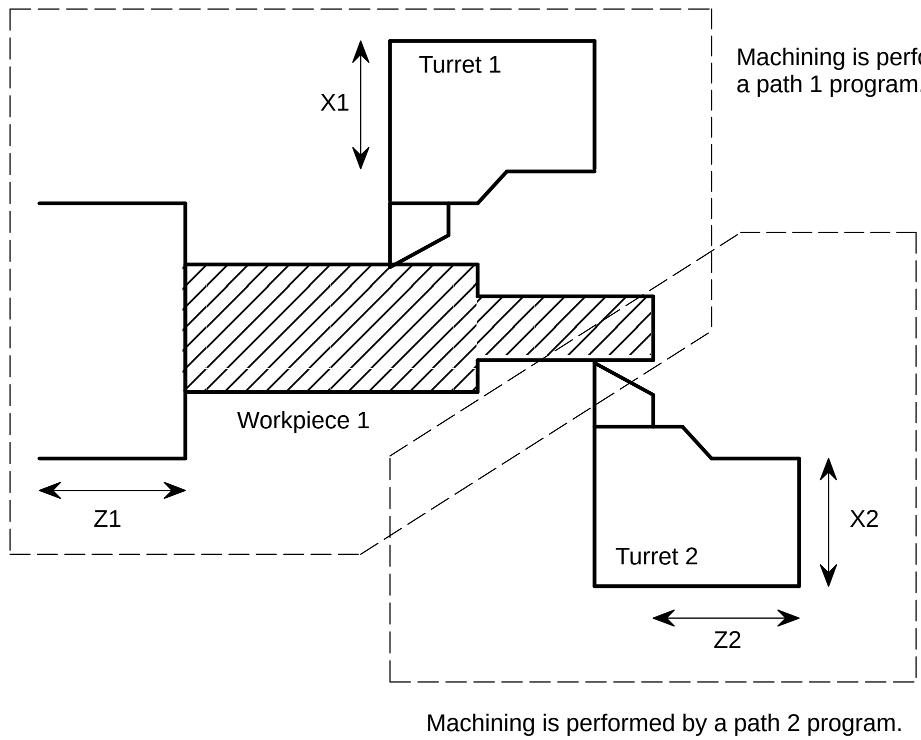
Use M codes for wait, beginning, and terminating superimposed control in a machining program in the stated order. It is also possible to begin and terminate superimposed control without using M codes.

1.9.4.3 Signal operation

When superimposed control begins or ends (when an M code is issued), the superimposed control axis selection signals OVLS1 to OVLS7 for the target slave axis (from the PMC to the CNC) are changed from “0” to “1” (to begin superimposed control) or from “1” to “0” (to terminate superimposed control).

1.9.4.4 Examples of applications

Suppose that a workpiece on the spindle (Z1–axis) that moves along the axis is to be cut with a tool in path 1 and a tool in path 2 simultaneously. This example superimposes the amount of movement of the Z1–axis on that of the Z2–axis.



1.9.4.5 Feedrate

Because the amount of movement of the master axis is added to that of the slave axis, the resulting speed of the slave axis may become much more larger than a normal speed (such as rapid traverse speed specified in a parameter). To solve this problem, it is necessary to set feedrates that are used only during superimposed control. The feedrates and time constants that are used only during superimposed control include:

- Rapid traverse rate: Parameter No. 8190
- Rapid traverse override F0 rate: parameter No. 8191
- Rapid traverse linear acceleration/deceleration time constant: Parameter No. 8192
- Maximum cutting feedrate: Parameter No. 8193
- (• Manual rapid traverse rate: Parameter No. 8190 or 1424 whichever is smaller)

These special parameters are used for both master and slave axes during superimposed control. Appropriate values should be specified with the resulting feedrate taken into account. When superimposed control begins or ends during automatic operation, it is impossible to switch the maximum cutting feedrate in the current block and the next block. If an M code that does not involve buffering is used to direct superimposed control to begin or end, the maximum cutting feedrate is switched in a block next to the current block. The rates other than the maximum cutting feedrate are switched immediately when superimposed control begins or ends.

1.9.4.6 Differences between superimposed control and ordinary synchronous control

- Neither out–of–synchronization compensation or detection is performed between the master and slave axes during superimposed control.
- A parking signal is ineffective for axes under superimposed control.
- When superimposed control is terminated during automatic operation, move commands and coordinate system setting can be executed for the slave axis immediately. Unlike synchronous control, superimposed control does not inhibit move commands in two or three blocks including the current block.
- A reference position return cannot be specified for the salve axis under superimposed control.

1.9.5 Signal

Synchronous control axis selection signals SYNC1 to SYNC7 <G138#0 to G138#6>

[Classification] Input signal

[Function] These signals perform synchronous control.

[Operation] When one of these signals becomes “1”, the control unit:

- Begins synchronous control in such a way that the corresponding axis becomes a slave axis.

The axis with which the slave axis is synchronized is determined by parameter No. 8180.

Parking signals PK1 to PK7 <G122#0 to G122#6>

[Classification] Input signal

[Function] These signals place each axis in a parking state.

[Operation] When one of these signals becomes “1”, the control unit:

- Places the corresponding axis in a parking state.

If the corresponding axis is under synchronous control, it enters a parking state immediately regardless of whether the axis is moving. If a parking signal is set to “1” without specifying synchronous control, it is ignored.

Composite control axis selection signal MIX1 to MIX7 <G128#0 to G128#6>

[Classification] Input signal

[Function] These signals perform composite control.

[Operation] When one of these signals becomes “1”, the control unit:

- Begins composite control over the corresponding axis.

The axis with which the corresponding axis is controlled together is determined by parameter No. 8183.

> **NOTE**
> These signals are available only for path 1.

Superimposed control axis selection signals OVLS1 to OVLS7 <G190#0 to G190#6>

[Classification] Input signal

[Function] These signals perform superimposed control.

[Operation] When one of these signals becomes “1”, the control unit:

- Begins superimposed control over the corresponding axis.

The master axis is selected according to parameter No. 8186.

Synchronous/composite/superimposed control under way signals SYN1O to SYN7O <F118#0 to F118#6>

[Classification] Output signal

[Function] These signals indicate each axis is being subjected to synchronous/composite/superimposed control.

[Output condition] These signals become “1” under the following condition:

- The corresponding axis is under synchronous, composite, or superimposed control.

These signals become “0” under the following condition:

- The corresponding axis is not under synchronous, composite, or superimposed control.

> **CAUTION**
> Whether each axis is under synchronous, composite, or superimposed control does not always match whether the corresponding selection signal (synchronous control axis selection, composite control axis selection, or superimposed control axis selection signal) has been issued or not. For example, if these signals are set to “1” during an alarm, they are ignored. If a servo alarm occurs during these types of control, they are terminated automatically. Before attempting to perform these types of control, always check the state of these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G122		PK7	PK6	PK5	PK4	PK3	PK2	PK1
G128		MIX7	MIX6	MIX5	MIX4	MIX3	MIX2	MIX1
G138		SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
G190		OVLS7	OVLS6	OVLS5	OVLS4	OVLS3	OVLS2	OVLS1

	#7	#6	#5	#4	#3	#2	#1	#0
F118		SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O

1.9.6 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8160	NRS	SPE				ZSI	XSI	MXC

[Data type] Bit

MXC During mixed control of the X– or Z–axis, measurement direct input of tool offset value measured B performs calculation based on:

0 : Machine coordinates for the path being controlled
1 : Machine coordinates for another path subject to mixed control

> **NOTE**
> 1 This parameter is valid for setting tool compensation values for the X– or Z axis and setting shift of the workpiece coordinate system for the Z–axis in direct input of tool offset value measured B.
> 2 This parameter cannot be used when mixed control is applied to paths for which different minimum command increments (metric or inch) are specified.

XSI When MXC = 1, the machine coordinates along the X–axis for the other path subject to mixed control are fetched:

0 : With the sign as is
1 : With the sign inverted

ZSI When MXC = 1, machine coordinates along the Z–axis for the other path subject to mixed control are fetched:

0 : With the sign as is
1 : With the sign inverted

SPE The synchronization deviation is:

0 : The difference between the positioning deviation of the master axis and that of the slave axis.
1 : The difference between the positioning deviation of the master axis and that of the slave axis plus the acceleration/deceleration delay.

NOTE
When the master and slave axes have different acceleration/deceleration time constants, set 1.

NRS When the system is reset, synchronous, composite, or superimposed control is:

0 : Released.
1 : Not released.

	#7	#6	#5	#4	#3	#2	#1	#0
8161							CZM	NMR

[Data type] Bit

NMR When an axis subject to mixed control is placed in servo–off state:

0 : Mixed control is stopped.
1 : Mixed control is not stopped, provided bit 0 (FUP) of parameter No. 1819 is set to 1 to disable follow–up for the axis.

NOTE
Mixed control is not stopped only when bit 0 (FUP) of parameter No. 1819 is set to 1. If follow–up is disabled with the follow–up signal (*FLWU <G007 bit 5> =1), mixed control is stopped.

CZM When two Cs contour axes are subject to mixed control, the function for mixing reference position return commands for Cs contour axes is:

0 : Not used
1 : Used

	#7	#6	#5	#4	#3	#2	#1	#0
8162	MUMx	MCDx	MPSx	MPMx	OMRx	PKUx	SERx	SMRx

[Data type] Bit axis

SMRx Synchronous mirror–image control is:

0 : Not applied. (The master and slave axes move in the same direction.)
1 : Applied. (The master and slave axes move in opposite directions.)

SERx The synchronization deviation is:

0 : Not detected.
1 : Detected.

> **NOTE**
> When both master and slave axes move in synchronization, the positioning deviations of the corresponding axes are compared with each other. If the difference is larger than or equal to the value specified in parameter No. 8181, an alarm occurs. When either axis is in the parking or machine–locked state, however, the synchronization deviation is not detected.

PKUx In the parking state,

0 : The absolute, relative, and machine coordinates are not updated.
1 : The absolute and relative coordinates are updated. The machine coordinates are not updated.

> **WARNING**
> Set the parameter to 1 for any axes for which polar coordinate interpolation will be specified. Otherwise, coordinates may shift when single block stop or feed hold is specified in polar coordinate interpolation mode.

OMRx Superimposed mirror–image control is:

0 : Not applied. (The superimposed pulse is simply added.)
1 : Applied. (The inverted superimposed pulse is added.)

MPMx When composite control is started, the workpiece coordinate system is:

0 : Not set automatically.
1 : Set automatically.

> **NOTE**
> When the workpiece coordinate system is automatically set at the start of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis (parameter No. 8184).

MPSx When composite control is terminated, the workpiece coordinate system is:

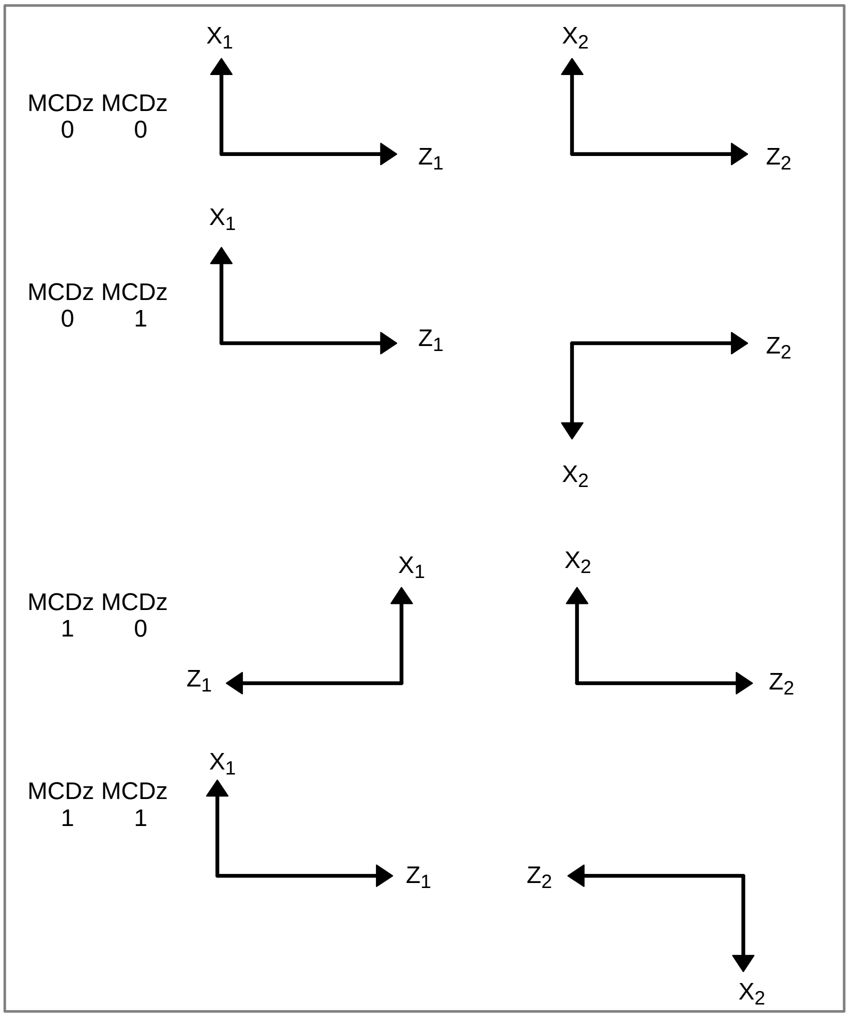
0 : Not set automatically.
1 : Set automatically.

> **NOTE**
> When the workpiece coordinate system is automatically set at the end of composite control, it is calculated from the following: Current machine coordinates and the workpiece coordinates at the reference point of each axis under composite control (parameter No. 1250)

MCDx The axes to be replaced with each other under composite control have the coordinate systems placed:

0 : In the same direction. Simple composite control is applied. (The axes of paths 1 and 2 move in the same direction.)
1 : In opposite directions. Mirror–image composite control is applied. (The axes of paths 1 and 2 move in opposite directions.)

This parameter determines the direction in which an axis moves. The parameter is also used to automatically set the coordinate system when composite control is started or terminated.



MUMx In mixed control, a move command for the axis:

0 : Can be specified.
1 : Cannot be specified.

> **NOTE**
> Upon the execution of a move command along an axis for which MUMx is set to 1 during mixed control, alarm P/S 226 is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
8163				SCDx	SCMx	SPSx	SPMx	MDXx

NOTE
Set the parameters SPMx, SPSx, SCMx, and SCDx for the master axis. These settings are referenced during automatic workpiece coordinate setting for the master axis at the start of synchronous control.

[Data type] Bit axis

MDXx In mixed control, the current position (absolute/relative coordinates) display indicates:

0 : Coordinates in the local system.
1 : Coordinates in the other system under mixed control.

SPMx When synchronous control is started, automatic workpiece coordinate system setting for the master axis is

0 : Not Performed.
1 : Performed.

NOTE
When a workpiece coordinate system is automatically set at the start of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates of each axis at the reference position set in parameter No. 8185.

SPSx When synchronous control terminates, automatic workpiece coordinate system setting for the master axis is:

0 : Not performed.
1 : Performed.

NOTE
When a workpiece coordinate system is automatically set at the end of synchronous control, the workpiece coordinate system is calculated from the current machine coordinates and the workpiece coordinates for each axis at the reference position set in parameter No. 1250.

SCMx When workpiece coordinates are calculated in synchronous control:

0 : The workpiece coordinates are calculated from the machine coordinates of the slave axis.
1 : The workpiece coordinates are calculated from the machine coordinates of the master axis and slave axis.

SCDx The positive (+) directions of the master axis and slave axis in the coordinate system in synchronous control are:

0 : Identical.
1 : Opposite.

Parameters SPMx, SPSx, SCMx, and SCDx must be specified for the master axis. These parameter settings are referenced in automatic setting of the workpiece coordinates for the master axis when synchronization control begins.

8180	Master axis with which an axis is synchronized under synchronous control

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to the maximum number of control axes, or 201, 202, 203, ... to 200 plus the maximum number of control axes

This parameter specifies the number of the master axis with which an axis is synchronized. When zero is specified, the axis does not become a slave axis and is not synchronized with another axis. When an identical number is specified in two or more axes, one master axis has two or more slave axes.

- Exercising synchronous control between two paths
  In the parameter of a slave axis, specify the axis number of the master axis with which the salve axis is to be synchronized.
  Setting: 1 to 8
  The value specified here must not exceed the maximum number of control axes.

(Example 1) Synchronizing the Z_2–axis with the Z_1–axis

Path 1		Path 2	
Parameter No. 8180x	0	Parameter No. 8180x	0
Parameter No. 8180z	0	Parameter No. 8180z	2
Parameter No. 8180c	0	Parameter No. 8180c	0
Parameter No. 8180y	0	Parameter No. 8180y	0

- Exercising synchronous control in a path
  In the parameter of a slave axis , specify 200 plus the number of the master axis with which the slave axis is to be synchronized.
  Setting: 201 to 208
  The value specified here must not exceed 200 plus the maximum number of control axes.

(Example 1) Synchronizing the Y_1–axis with the Z_1–axis

Path 1		Path 2	
Parameter No. 8180x	0	Parameter No. 8180x	0
Parameter No. 8180z	0	Parameter No. 8180z	0
Parameter No. 8180c	0	Parameter No. 8180c	0
Parameter No. 8180y	202	Parameter No. 8180y	0

8181	Synchronization error limit of each axis (Synchronous or composite control)

[Data type] Two–word axis

[Unit of data] Unit of detection

[Valid data range] 0 to 32767

When the synchronization deviation detected (SERx of Bit #1 parameter No. 8162 is set to 1), this parameter specifies the limit of the difference between the positioning deviation of the slave axis and that of the master axis. Set this parameter to the slave axis.

8182	Display of the synchronization error of an axis (synchronous or composite control)

[Data type] Two–word axis

[Unit of data] Unit of detection

[Valid data range] 0 or more

When the synchronization deviation is detected (SERx of Bit #1 parameter No. 8162 is set to 1), this parameter specifies the difference between the positioning deviation of the slave axis and that of the master axis. (The value is used for diagnosis.) The deviation is displayed on the slave side
The parameter is only of display. It should not be set.
The difference between the positioning deviation is:

(Positioning deviation of the master axis) ± (Positioning deviation of the slave axis)

Plus for a mirror–image synchronization command
Minus for a simple synchronization command

8183	Axis under composite control in path 1 corresponding to an axis of path 2

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to the maximum number of control axes

This parameter specifies an axis of path 1 to be placed under composite control with each axis of path 2. The value specified here must not exceed the maximum number of axes that can be used in path 1. When zero is specified, control of the axis is not replaced under composite control. An identical number can be specified in two or more axes, but composite control cannot be exercised for all of tem at a time.

> **NOTE**
> Specify this parameter only for path 2.

(Example 1) Exercising composite control to replace the X_1–axis with the X_2–axis

Path 1	Path 2
Parameter No. 8183x 0	Parameter No. 8183x 1
Parameter No. 8183z 0	Parameter No. 8183z 0
Parameter No. 8183c 0	Parameter No. 8183c 0
Parameter No. 8183y 0	Parameter No. 8183y 0

(Example 2) Exercising composite control to replace the Y_1–axis with the X_2–axis

Path 1	Path 2
Parameter No. 8183x 0	Parameter No. 8183x 4
Parameter No. 8183z 0	Parameter No. 8183z 0
Parameter No. 8183c 0	Parameter No. 8183c 0
Parameter No. 8183y 0	Parameter No. 8183y 0

8184	Coordinates of the reference point of an axis on the coordinate system of another axis under composite control

[Data type] Two–word axis

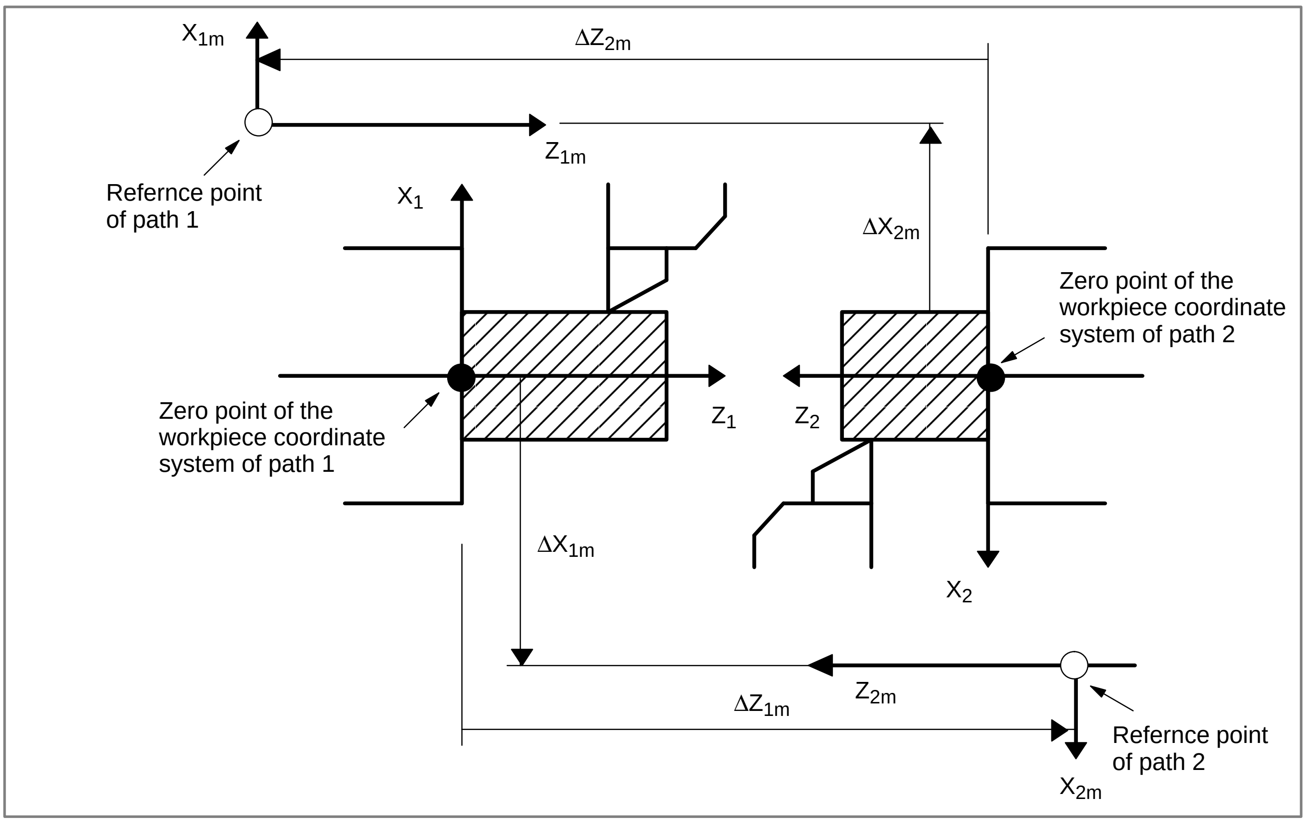
[Unit of data]

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

[Valid data range] 0 to ±99999999

This parameter specifies the coordinates of the reference point of an axis on the coordinate system of another axis under composite control.
The parameter is validated when MPMx of bit 4 parameter No. 8162 is set to 1.

(Example) Exercising composite control to replace the X_1–axis with the X_2–axis



(ΔX_{1m}, ΔZ_{1m}) are the coordinates of the reference point of path 2 on the workpiece coordinate system of path 1. (ΔX_{2m}, ΔZ_{2m}) are the coordinates of the reference point of path 1 on the workpiece coordinate system of path 2.

ΔX_{1m} is specified for the X–axis of path 1 and ΔX_{2m} for the X–axis of path 2.

If bit 4 of parameter No. 8162 MPMx is set to 1 when composite control is started, the workpiece coordinate system satisfying the following conditions is specified:

X_1 = (Value specified for the X–axis of path 1) ± (Machine coordinates of X_2)

Plus when parameter No. 8162#6 MCDx of path 1 is set to 0
Minus when parameter No. 8162#6 MCDx of path 1 is set to 1

X_2 = (Value specified for the X–axis of path 2) ± (Machine coordinates of X_1)

Plus when parameter No. 8162#6 MCDx of path 2 is set to 0
Minus when parameter No. 8162#6 MCDx of path 2 is set to 1

If bit 5 of parameter No. 8162 MPSx is set to 1 when composite control is terminated, the workpiece coordinate system satisfying the following conditions is specified:
X_1 = Parameter No. 1250 of path 1 + Machine coordinate of X_1
X_2 = Parameter No. 1250 of path 2 + Machine coordinate of X_2

8185	Workpiece coordinates on each axis at the reference position

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

This parameter specifies the reference position coordinates along the slave axes, according to the workpiece coordinate system for the master axis, when the tool is positioned to the reference position along the master axis. This parameter is enabled when SPMx of bit 1 parameter No. 8163 is set to 1. Set this parameter for the master axis.

8186	Master axis under superimposed control

[Data type] Byte axis

[Valid data range] 1, 2, 3, ... to number of control axes

This parameter specifies the axis number of the master axis under superimposed control.
When zero is specified , the axis does not become a slave axis under superimposed control and the move pulse of another axis is not super imposed.

8190	Rapid traverse rate of an axis under superimposed control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000
Rotaion axis	1 deg/min	30 to 240000	30 to 100000

Set a rapid traverse rate for each of the axes when the rapid traverse override of the axes (master and slave axes) under superimposed control is 100%. A manual rapid traverse rate depends on this parameter or No. 1424 (When No. 1424 is set to 0, No. 1420) whichever is smaller.

8191	F0 velocity of rapid traverse override of an axis under superimposed control

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 150000	6 to 12000

This parameter specifies the F0 velocity of rapid traverse override for each of the axes (master and slave axes) under superimposed control.

8192	Linear acceleration/deceleration time constant in rapid traverse of an axis under superimposed control

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

This parameter specifies the linear acceleration/deceleration time constant in rapid traverse for each of the axes (master and slave axes) under superimposed control.

8193	Maximum cutting feedrate under superimposed control

[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 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000
Rotaion axis	1 deg/min	30 to 240000	30 to 100000

This parameter specifies the maximum cutting feedrate under superimposed control.

8194	Maximum cutting feedrate of an axis under superimposed control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 48000
Rotaion axis	1 deg/min	6 to 240000	6 to 100000

This parameter specifies the maximum cutting feedrate for an axis under superimposed control.

1.9.7 Alarms and messages

If one of the alarms listed below occurs, it terminates synchronous, composite, and superimposed control for all axes.

1.9.7.1 P/S alarms

Number	Message	Description
225	Synchronous or composite control error	This alarm occurs under either of the following conditions (detected when synchronous, composite, or superimposed control is terminated). (1) There is an error in an axis number parameter (parameter No. 1023). (2) An invalid control command is issued. If this alarm occurs when synchronous, composite, or superimposed control is terminated, place the machine in an emergency stop state before resetting the alarm.
226	A move command was issued to a synchronous axis.	When an axis is in a synchronization mode, a move command was issued to that axis. (Only during synchronous control) During composite control, a move command was issued to an axis for which parameter MUMx (bit 7 of parameter No. 8162) is “1”. (Only during composite control) In a control mode other than synchronous or composite control, a move command was issued to an axis for which parameter NUMx (bit 7 of parameter No. 8163) was “1”.

Number	Message	Description
229	Synchronization cannot be maintained.	This alarm occurs under either of the following conditions. (1) Synchronous or composite control cannot be maintained because of system overload. (2) Synchronous or composite control cannot be maintained because of a hardware failure. (This alarm does occur during normal use.)
000	Turn the power off.	This message is issued if superimposed control is suspended because of an alarm that occurs when the axis is moving. Turn the power of the CNC off, then on gain.

1.9.7.2 Servo alarms

Number	Message	Description
407	Servo alarm: Excessive error	A positional deviation for a synchronous axis exceeded the specified value. (Only during synchronous control).

1.9.8 Definition of Warning, Caution, and Note

1.9.8.1 Items common to synchronous, composite, and superimposed control

WARNING

1 When synchronous, composite, or superimposed control begins or ends, the target axes must be at a stop.
2 All axes subjected to synchronous, composite, or superimposed control must have the same least command, detection increment, and diameter/radius specification. Otherwise, the amount of movement will differ from one axis to another.
3 When an axis is under synchronous, composite, or superimposed control, do not change the parameters related to that axis.
4 Before starting synchronous, composite, superimposed control, make sure that for the target axis, a reference position return after power–on has been made or a reference position has been set up according to the absolute pulse coder.
5 Before starting synchronous, composite, or superimposed control after an emergency stop, servo–off, or servo alarm is released, be sure to make a return to the reference position and set up the necessary coordinate system.

CAUTION

Acceleration/deceleration control, pitch error compensation, backlash compensation, and stored stroke limit check are carried out regardless of synchronous or composite control. During superimposed control, these operations except acceleration/deceleration are performed on the position where superimposed pulses have been added.

NOTE

1 More than one axis can be subjected to synchronous, composite, or superimposed control. On the other hand, an axis cannot be synchronized with more than one axis simultaneously. Moreover, an axis under composite control cannot be synchronized with another axis or cannot doubly be subjected to composite control.

2 Synchronous, composite, or superimposed control cannot be performed between a linear axis and a rotation axis.

3 Synchronous, composite, or superimposed control cannot be specified simultaneously with simplified synchronous control. Synchronous control within one path provides the same functions as simplified synchronous control.

1.9.8.2 Items related only to synchronous control

WARNING

1 The same acceleration/deceleration time constants and servo parameters should be used for axes subjected to synchronous control as much as possible. If there is a large difference in a set value between the axes, a deviation will occur in the actual movement of the machine.

2 The workpiece coordinate system of a synchronous slave axis is not affected by the synchronous master axis operations that affect workpiece coordinate systems but do not cause the machine to move, such as workpiece coordinate system set/shift and geometry offset commands.

3 If a wear offset command or tool–nose radius compensation is performed for the synchronous master axis, the travel path of the slave axis is shifted by the offset, but the shift is not set as an offset (no offset vector is created).

CAUTION

A move command should not be issued to a synchronous slave axis during synchronous control.

1.9.8.3 Restrictions imposed during synchronous, composite, and superimposed control

Function	During synchronous control	During composite control	During superimposed control
Acceleration/deceleration control	The acceleration/deceleration control for the master axis is performed also for the synchronous slave axes, but different time constants are used.	The acceleration/deceleration control originally specified for one path is used also for the other path, but different time constants are used (*3).	The move pulses that are effective after acceleration/deceleration for the superimposed control master axis are added to those for the slave axes.
Linear acceleration/deceleration after cutting feed interpolation	Possible	Possible	Possible
Feedrate clamping	The axes are clamped at the feedrate of the master axis.	The axes in both paths are clamped at the feedrate originally specified for one path (*4).	The axes are clamped to the feedrate specified for superimposed control.
Reference position return	A reference position return is possible for the master axis unless it is in a parking state. If the master axis is in a parking state, only automatic reference position return (G28) is possible for the master axis. (⇒ Section 1.9.2.5)	A reference position return is possible for axes not under composite control. For axes under composite control, only an automatic reference position return (G28) is possible.	Impossible for superimposed control slave axes.
Second–, third–, or fourth–reference position return	Possible (⇒ 1.9.2.5).	Possible	Impossible for superimposed control slave axes.
Reference position return check	Possible (⇒ 1.9.2.5).	Possible	Impossible for superimposed control slave axes.
PMC axis control	Possible for other than synchronous slave axes.	Possible	Possible
Polar coordinate interpolation and cylindrical interpolation	Possible	Switching between independent control and composite control should be carried out during cancel mode.	Possible
Handle interrupt	Performed regardless of synchronous control.	Possible for axes having nothing to do with composite control.	Performed regardless of superimposed control (*5).
Mirror image	Each signal originally belonging to a particular axis is effective for that axis (*1).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).
Machine lock	Each signal originally belonging to a particular axis is effective for that axis (*1).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).

Function	During synchronous control	During composite control	During superimposed control
Interlock	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5).
Override	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5)
External deceleration	The signals for the synchronous master axis are effective for the synchronous slave axes (*2).	Signals originally specified for one path are effective for the other path (*4).	Signals for the master axis are effective as superimposed control pulses (*5)
Skip function	Impossible for slave axes.	Possible for axes having nothing to do with composite control.	Possible.
Automatic tool compensation	Impossible for slave axes.	Possible for axes having nothing to do with composite control.	Impossible for superimposed control slave axes.
Direct tool compensation measurement input B	Impossible for slave axes.	Possible (*7)	Impossible for superimposed control slave axes.
Follow–up	Impossible during synchronous control.	Impossible during composite control.	Impossible during superimposed control.
Program restart	Impossible for a program involving synchronous control.	Impossible for a program involving composite control.	Impossible for a program involving superimposed control.
Cs contour control	Synchronous control is possible (*6).	Composite control is possible (*6).	Superimposed control is possible (*6).
Spindle positioning	Synchronous control is impossible.	Composite control is impossible.	Superimposed control is impossible.

(*1) Processed after synchronization pulses are sent to the slave axes.

(*2) After it is processed on the master side, synchronization pulses are sent.

(*3) Composite control pulses and acceleration/deceleration type are sent. The time constant for the slave axis is used.

(*4) Composite control pulses are sent after processed on the master side.

(*5) Performed normally for move commands originally intended to the master or slave axes, but not performed on the slave side for superimposed control pulses received from the master axis.

(*6) Restricted to a combination of Cs axes. Necessary signal operations and orientation should be performed for each axis separately. Also specify parameter CZM (bit 1 of parameter No. 8161).

(*7) Specify parameters MXC, XSI, and ZSI (bits 0, 1, and 2 of parameter No. 8160).

1.9.8.4
Reading the coordinates during synchronous, composite, or superimposed control

The following list summarizes how positional information such as custom macro system variables and current coordinates from the PMC window are read during synchronous, composite, or superimposed control.

Positional information	During synchronous control	During composite control	During superimposed control
Absolute coordinate	Readable	Readable (*1)	Readable (*2)
Machine coordinate	Readable	Readable	Readable
End of each block	Readable only for the master axis	Readable (*1)	Readable (*2)
Skip signal position	Readable only for the master axis	Unreadable	Readable (*2)

(*1) The coordinates are represented in the coordinate system that is effective during composite control. Their relationship with the machine coordinate system differs from the relationship that exists during independent control.

(*2) No superimposed control pulse is added.

1.9.8.5
Terminating synchronous, composite, or superimposed control

Synchronous, composite, or superimposed control is terminated not only when the corresponding synchronization signal becomes off but also when the following conditions occur.

(1) Emergency stop

(2) Reset

(3) Servo alarm

(4) Servo off (*1)

(5) Overtravel

(6) Alarm related to synchronous, composite, or superimposed control

(7) P/S000 alarm

If one of the above conditions occurs for either path, it terminates synchronous control, composite, and superimposed control for all axes. If one of the above conditions occurs for one path during synchronous, composite, or superimposed control, the other path is placed in a feed hold state (during automatic operation) or interlock state (during manual operation).

(*1) Setting parameter NMR (bit 0 of parameter No. 8161) specifies that synchronous, composite, or superimposed control be not terminated even when an axis under composite control enters a servo–off state. (If an axis under synchronous or superimposed control enters a servo–off state, synchronous, composite, and superimposed control is terminated.

1.9.8.6
Status output signals for an axis under synchronous, composite, or superimposed control

Status output signal	During synchronous control	During composite control	During superimposed control
Axis moving signal MVn F0102/F1102 (See Section 1.2.5.)	• The master axis moving signal becomes ”1” when the master or slave axis is moving. • The slave axis moving signal is always ”0” (*1).	• The moving signal for an axis to which a move command is originally issued becomes ”1”. The moving signal for the axis that is actually moving does not become ”1” (*1).	• The master axis moving signal works as usual. • The slave axis moving signal reflects the state of movement due to a command for the slave axis rather than superimposed control pulses.
Axis movement direction signal MVDn F0106/F1106 (See Section 1.2.5.)	• The master axis movement direction signal indicates the direction in which the master axis is moving. • The slave axis movement direction signal indicates the direction of movement after synchronous control mirror image processing.	• The axis movement direction signal indicates the actual movement direction (that is, direction after composite control mirror image processing).	• The master axis movement direction signal indicates the direction in which the master axis is moving. • The slave axis movement direction signal indicates the direction of movement after superimposed control pulses are added.
Axis in position signal INPn F0104/F1104 (See Section 7.2.6.1.)	• The master axis in position signal becomes “1” when both master and slave axes are in position. • The slave axis in position signal is always “1”.	• The in position signal for an axis in a path for which a move command is issued reflects the state of the axis that is driven by that move command.	• The master axis in position signal works as usual. • The slave axis in position signal is always “1”.

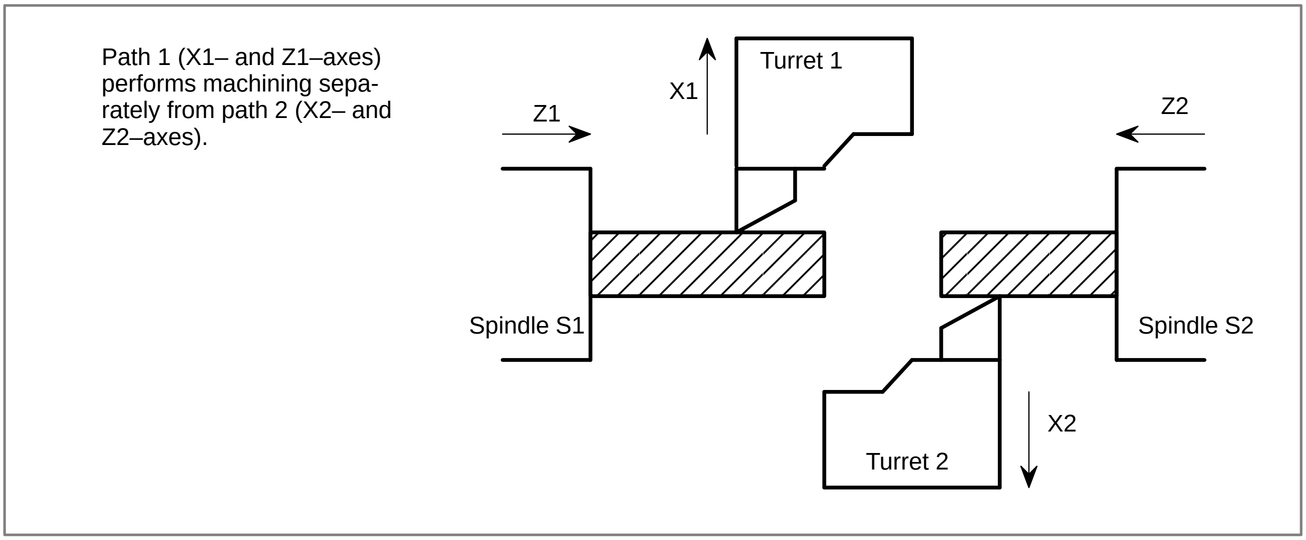
(*1) A positional deviation check does not depend on the state of this signal. If move command pulses have been output to a motor (either master or slave), parameter No. 1828 is used as a limit. Otherwise, parameter No. 1829 is used.

1.9.9 Examples of Applications

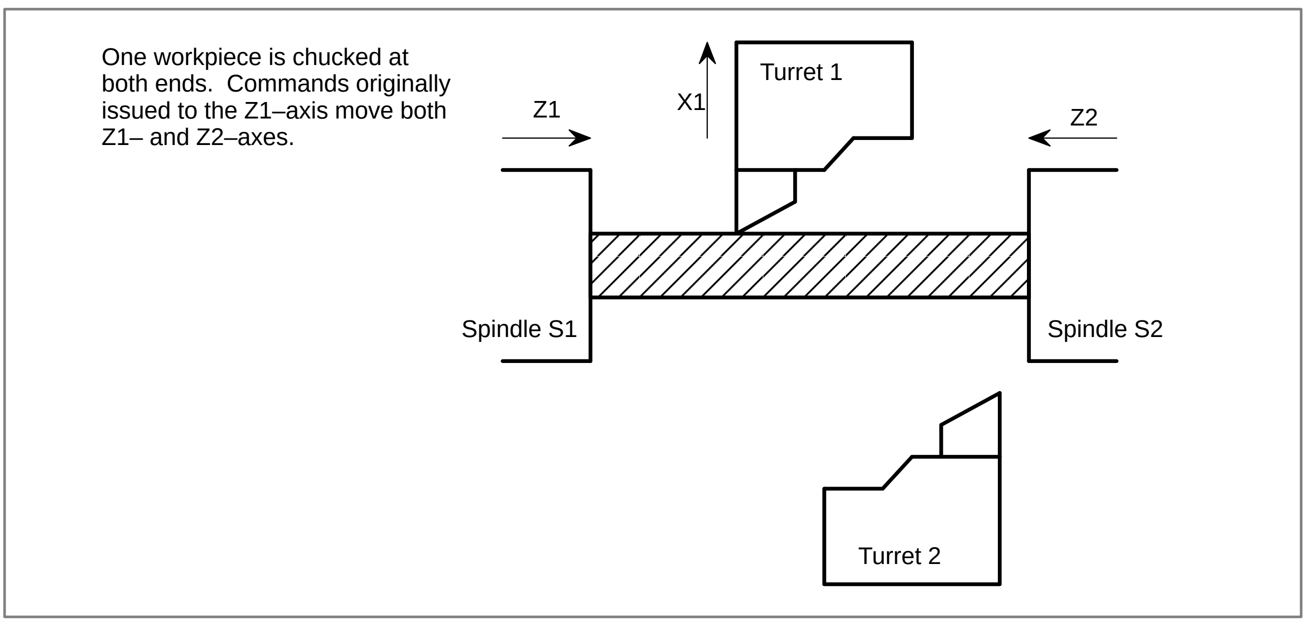
1.9.9.1 Independent control and synchronous control of the Z1– and Z2–axes

(1) Machine configuration

(a) Independent control



(b) Synchronous control of the Z1– and Z2–axes



(2) Parameter setting

- To synchronize the Z2–axis with the Z1–axis, set parameter No. 8180z of path 2 to “2”.

- To apply mirror–image synchronization (because initially the positive direction of one axis is opposite to that of the other axis), set SMRz (bit 0 of parameter No. 8162) of path 2 to “1”.
- To detect out–of–synchronization (because both axes should move by the same amount), set SERz (bit 1 of parameter No. 8162) to “1”. Set a value from 100 to 1000 as a limit to out–of–synchronization in parameter No. 8181z of path 2 (this limit varies from one machine to another).
- A difference in the positional deviation between the Z1– and Z2–axes is indicated in parameter No. 8182z during synchronization.

(3) Signal operation

- Set signal G1138#1 SYNC2 to “1” when the Z1– and Z2–axes start moving in synchronization.
- Reset signal G1138#1 SYNC2 to “0” when synchronization is terminated.
- Also reset signal G1138#1 SYNC2 to “0” if an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G1138#1 SYNC2 reset to “0”.

(4) Sample program

Path 1	Path 2	
N1000 ········	N2000 ········	Machining under control independent of the other path
N1010 Z80. ;	N2010 Z150. ;	Moves the workpiece and chuck to the specified position.
N1020 M200 ;	N2020 M200 ;	Waits for completion of movement.
N1030 M61 ;		Clamps the workpiece and begins synchronization
N1040 M3 S800 ;		Turns the spindle in normal direction.
N1050 Z- 25. ;		Moves the Z1–axis.
N1060 ········		Machining with the X1– and Z1–axes
N1070 M62 ;		Terminates synchronization and unclamps the workpiece.
N1080 M201 ;	N2080 M201 ;	Waits for synchronization to be terminated.
N1090 ;	N2090 ;	Dummy block (performing no move command)
N1100 ········	N2100 ·······	Machining under control independent of the other path

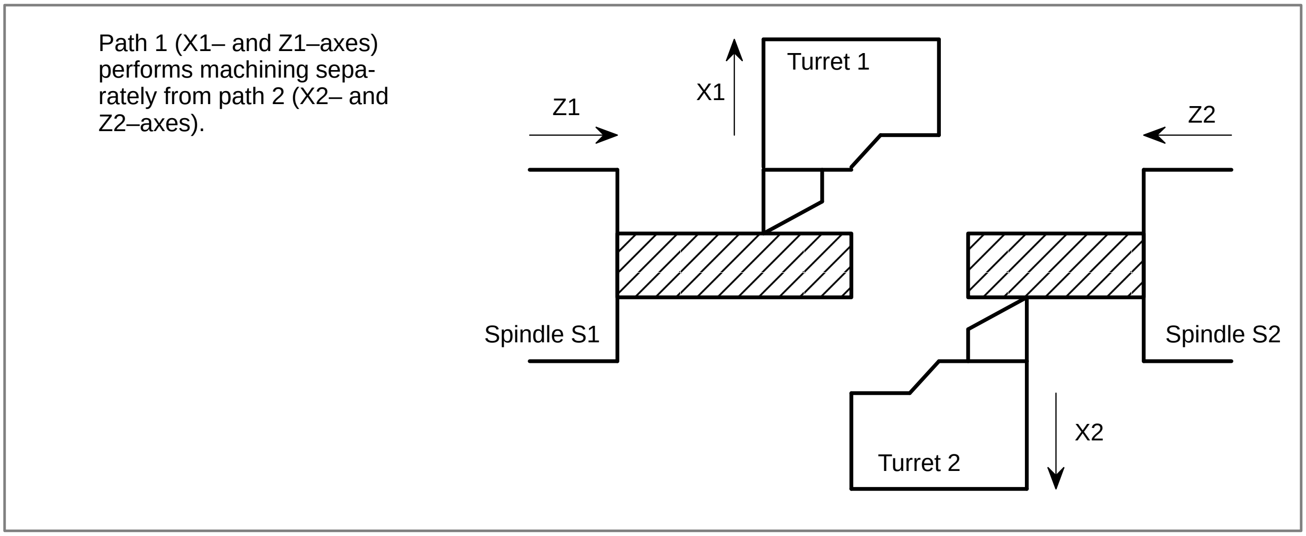
In this example, assume that M61 clamps the workpiece and sets signal G1138#1 SYNC2 to “1” and that M62 resets signal G1138#1 SYNC2 to “0” and unclamps the workpiece.

NOTE
It is necessary to make the speed of spindle S1 equal that of spindle S2. For example, issue spindle commands of path 1 to both S1 and S2.

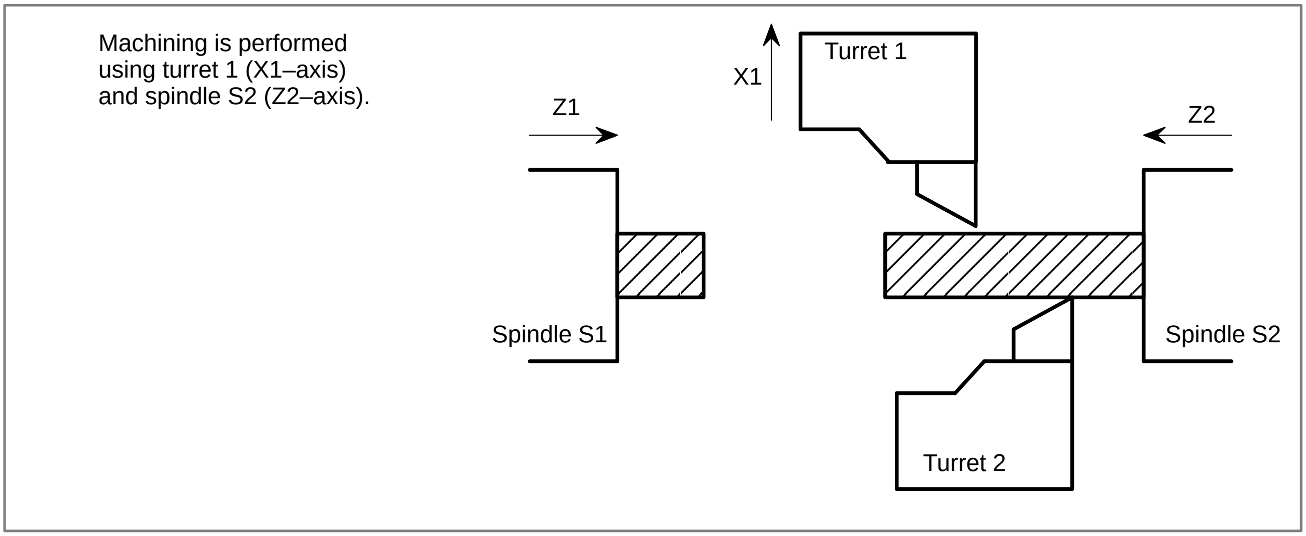
1.9.9.2 Independent control and interpolation for the X1– and Z2–axes

(1) Machine configuration

(a) Independent control



(b) Interpolation for the X1– and Z2–axes



Interpolation for the X1– and Z2–axes can be carried out by either of the following two methods.

1. The path 2 program directs the X2– and Z2–axes, synchronizes the X1–axis with the X2–axis, and causes the X2–axis to park. The path 1 program issues no move command.
2. Composite control is performed in which move commands are switched between the X1–axis in one path and the X2–axis in the other path. Path 1 does not issue move commands.

The following sections describe a case in which synchronous control is used and a case in which composite control is used, separately.

1.9.9.2.1 Using synchronous control

(1) Parameter setting

- To synchronize the X1–axis with the X2–axis, set parameter No. 8180x of path 1 to “1”.
- Do not specify mirror image, because for both X1– and X2–axes, the direction in which they go away from the workpiece center is defined as positive.
- Do not specify out–of–synchronization detection for the X2–axis because it is caused to park.
- Parameter No. 8182x indicates a difference in the positional deviation between the X2– and X1–axes during synchronous control.

(2) Signal operation

- Set signals G0138#0 SYNC1 and G1122#0 PK1 to “1” when synchronous control begins for the X2– and X1–axes.
- Reset signals G0138#0 SYNC1 and G1122#0 PK1 to “0” when synchronization is terminated.
- Also reset signals G0138#0 SYNC1 and G1122#0 PK1 to “0” if an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0138#0 SYNC1 or G1122#0 PK1 reset to “0”.

(3) Sample program

Path 1	Path 2	
N1000	N2000	Machining under control independent of the other path
N1010 Z0 ;	N2010 Z20. ;	Moves the workpiece to the specified position.
N1020 X120. ;	N2020 X120. ;	Moves each X–axis to their start position for synchronization (X1 = X2)
N1030 M200 ;	N2030 M200 ;	Waits for completion of movement.
	N2040 M55 ;	Synchronizes the X2– and X1–axes and causes the X2–axis to park.

	N2050 T0212 ;	Specifies an offset for turret 1.
	N2060 S1000 M4 ;	Reverses the spindle.
	N2070 G0 X30. Z55. ;	Performs machining using the X1– and Z2–axes.
	N2080 G1 F0. 2 W- 15. ;	
	N2090 ·······	
	N2100 M56 ;	Terminates synchronization and parking.
N1110 M201 ;	N2110 M201 ;	Waits for synchronization to be terminated.
N1120 ;	N2120 ;	Dummy block (performing no move command)
N1130 ······	N2130 ········	Machining under control independent of the other path

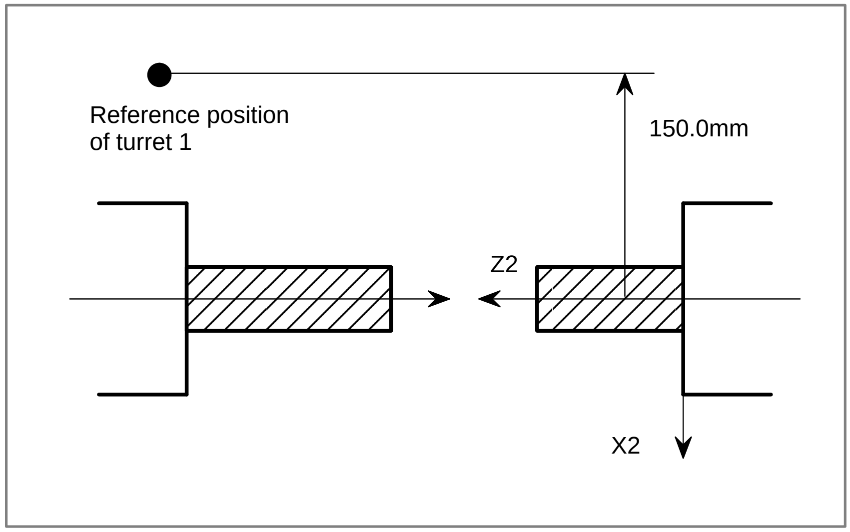
In this example, assume that M55 begins control of turret 1 in path 2 and that M56 terminates control of turret 1 in path 2.

NOTE
When the X–axis is under synchronous control, path 1 cannot issue a move command to the X1–axis, but can move the Z1–axis.

1.9.9.2.2 Using composite control

(1) Parameter setting

- To specify composite control in which commands for the X1–axis are interchanged with those of the X2–axis, set parameter No. 8183x of path 2 to “1”.
- Set MCDx (bit 6 of parameter No. 8162) of path 2 to “1”, because the direction of the X1–axis is opposite to that of the X2–axis.
- To cause the position of turret 1 to be specified automatically in the workpiece coordinate system in path 2 when composite control begins, set MPSx (bit 5 of parameter No. 8162) to “1”.
- To cause the position of turret 1 to be specified automatically in the workpiece coordinate system in path 1 when composite control ends, set MPMx (bit 4 of parameter No. 8162) to “1”.
- Assuming that the X–coordinate of the reference position of turret 1 in the workpiece coordinate system in path 2 is –150.0 mm as shown below, set “–150000” in parameter No. 8184x of path 2 for automatic coordinate system setting.



(2) Signal operation

- Set signal G0128#0 MIX1 to “1” when composite control begins for the X2– and X1–axes.
- Reset signal G0128#0 MIX1 to “0” when composite control ends.
- Also reset G0128#0 MIX1 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0128#0 MIX1 reset to “0”.

(3) Sample program

Path 1	Path 2	
N1000 ·········	N2000 ·········	Machining under control independent of the other path
N1010 Z0 ;	N2010 Z20. ;	Moves each workpiece to the specified position.
	N2020 X120. ;	Moves the X2–axis to a position where no interference occurs.
N1030 M200 ;	N2030 M200 ;	Waits for completion of movement.
	N2040 M55 ;	Begins composite control of the X2– and X1–axes (the position of turret 1 is set up as workpiece coordinates in path 2.)
	N2050 ;	Dummy block (performing no move command)
	N2060 T0212 ;	Specifies an offset for turret 1.
	N2070 S1000 M4 ; !	Performs machining using the X1– and Z2–axes.
	N2080 G0 U10. W- 20. ;	
	N2090 G1 F0. 2 W- 15. ;	
	N2100 ········	

	N2110 M56 ;	Terminates composite control (the position of turret 1 is set up as workpiece coordinates in path 1.)
N1120 M201;	N2120 M201 ;	Waits for composite control to be terminated.
N1130 ;	N2130 ;	Dummy block (performing no move command)
N1140 ········	N2140 ·······	Machining under control independent of the other path

In this example, assume that M55 begins control of turret 1 by a path 2 program and that M56 terminates control of turret 1 by a path 2 program.

NOTE

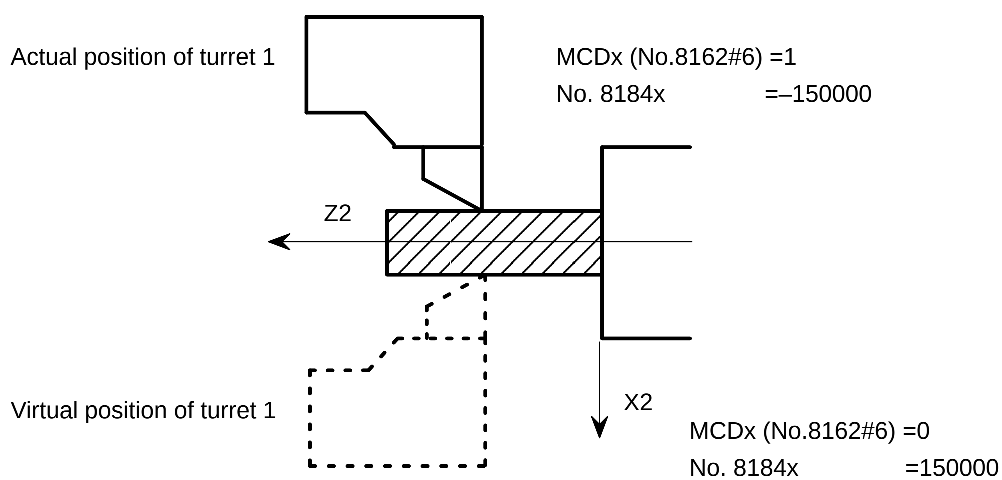
1 It is not always necessary to cause a coordinate system to be set up automatically when composite control begins or ends. If automatic coordinate system setting is not specified, an appropriate coordinate system is set by program.

2 When the X–axis is under composite control, the X2–axis can be moved in path 1 using move commands for the X–axis.

3 The above parameter setting specifies that turret 1 is located on the negative side of the X–coordinate in the workpiece coordinate system of path 2. So, for example, to move turret 1 toward the center of the workpiece, specify U+10, and to move it away from the center, specify U–10 (note the sign is a minus). If this is inconvenient, set the following parameters as follows:

Bit 6 of parameter No. 8162 (MCDx) = 0
Parameter No. 8184x = 150000

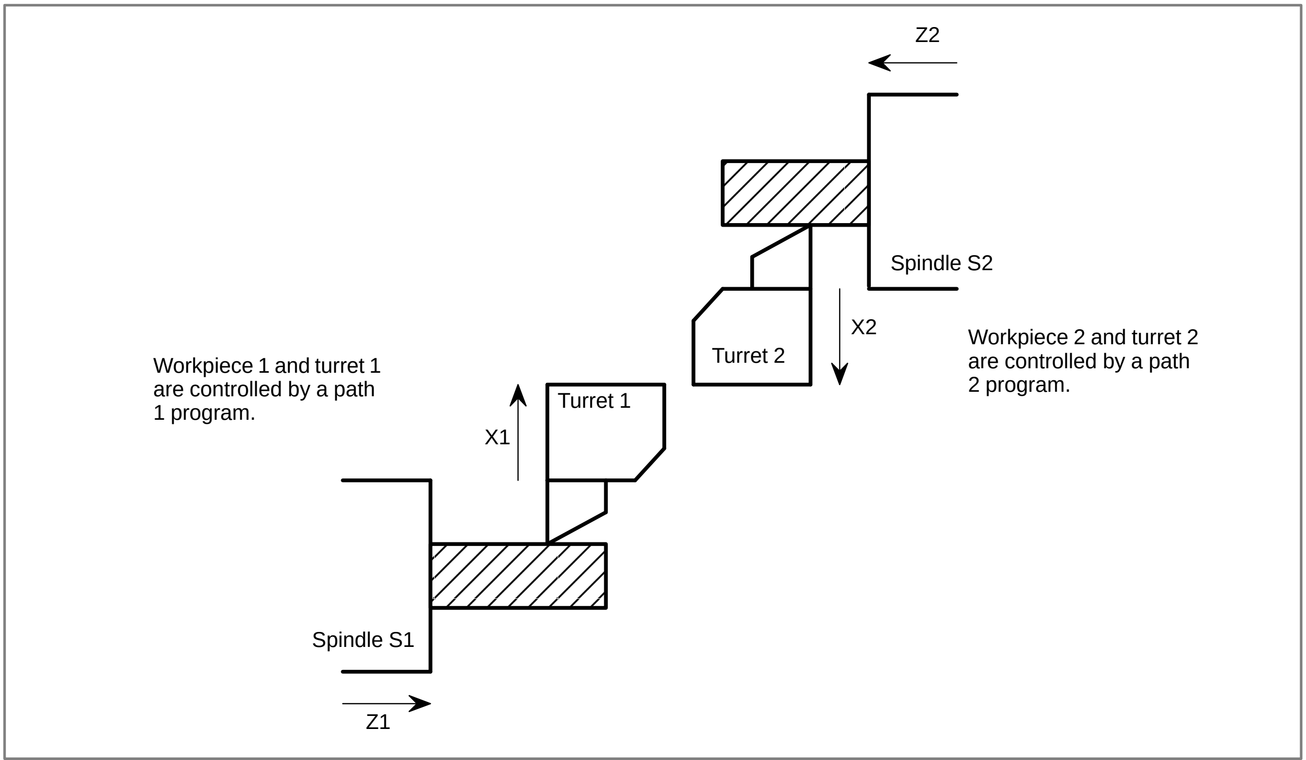
This parameter setting specifies that turret 1 be located virtually on the positive side of the X–coordinate.



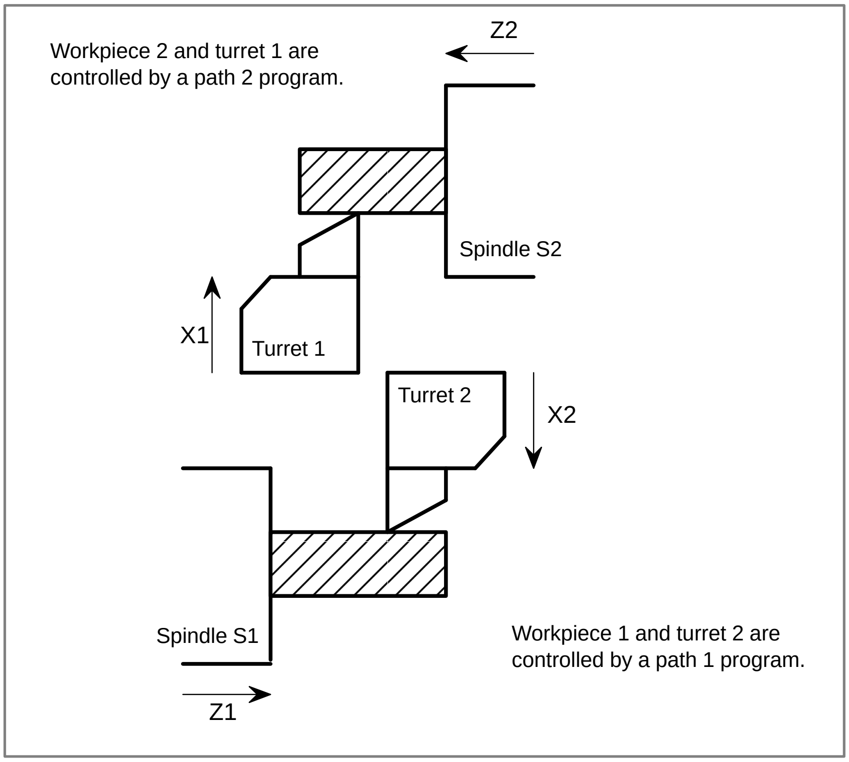
1.9.9.3 Independent control and interpolation between the X1– and Z2–axes and between the X2– and Z1–axes

(1) Machine configuration

(a) Independent control

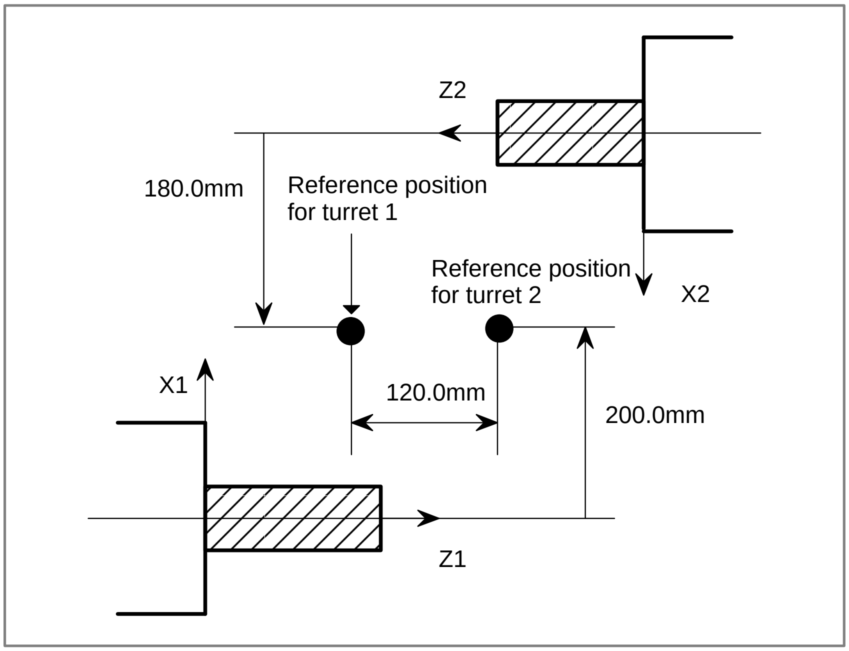


(b) Interpolation between the X1– and Z2–axes and between the X2– and Z1–axes



(2) Parameter setting

- To specify composite control in which commands for the X1–axis are interchanged with those for the X2–axis, set parameter No. 8183x of path 2 to “1”.
- Set MCDx (bit 6 of parameter No. 8162) of paths 1 and 2 to “1”, because the direction of the X1–axis is opposite to that of the X2–axis.
- To cause the position of a turret in one path to be specified automatically in the workpiece coordinate system of the other path when composite control begins, set MPMx (bit 4 of parameter No. 8162) to “1”.
- To cause the position of a turret in each path to be specified automatically in the workpiece coordinate system of that path when composite control ends, set MPSx (bit 5 of parameter No. 8162) to “1”.
- Assuming that the relationships between the workpiece coordinates and reference position of each path are as shown below, set “200000” in parameter No. 8184x of path 1 and “180000” in parameter No. 8184x of path 2 for automatic coordinate system setting.



(3) Signal operation

- Set signal G0128#0 MIX1 to “1” when composite control begins for the X2– and X1–axes.
- Reset signal G0128#0 MIX1 to “0” when composite control ends.
- Also reset G0128#0 MIX1 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G0128#0 MIX1 reset to “0”.

(4) Sample program

Path 1	Path 2	
N1000 ········	N2000 ········	Machining under control independent of the other path
N1010 M350 ;	N2010 M350 ;	Waits for composite control to begin.
	N2020 M55 ;	Begins composite control for the X1– and X2–axes.
N1030 M351 ;	N2030 M351 ;	Composite control has begun.
N1040 ;	N2040 ;	Dummy block (performing no move command)

N1050 T0313	N2050 T0212 ;	Selects a tool for composite control and sets the offset.
N1060 G50 W120. ;	N2060 G50 W120. ;	Shifts the Z–axis workpiece coordinate system.
N1070 S1000 M4 ;	N2070 S1500 M4 ;	Performs machining under composite control.
N1080 G0 X20. Z15. ;	N2080 G0 X15. Z30.;	
N1090 G1 F0. 5 W- 8. ;	N2090 G1 F0. 1 W- 5. ;	
N1100 ······	N2100 ········	
N1110 M360 ;	N2110 M360 ;	Waits for composite control to be terminated.
	N2120 M56 ;	Terminates composite control.
N1130 M361 ;	N2130 M361 ;	Composite control has ended.
N1140 ;	N2140 ;	Dummy block (performing no move command)
N1150 G50 W- 120. ;	N2150 G50 W- 120. ;	Shifts the Z–axis workpiece coordinate system.
N1160 ·······	N2160 ········	Machining under control independent of the other path

In this example, assume that M55 begins composite control (sets signal G0128#0 MIX1 to “1”) and that M56 terminates composite control (resets signal G0128#0 MIX1 to “0”).

NOTE

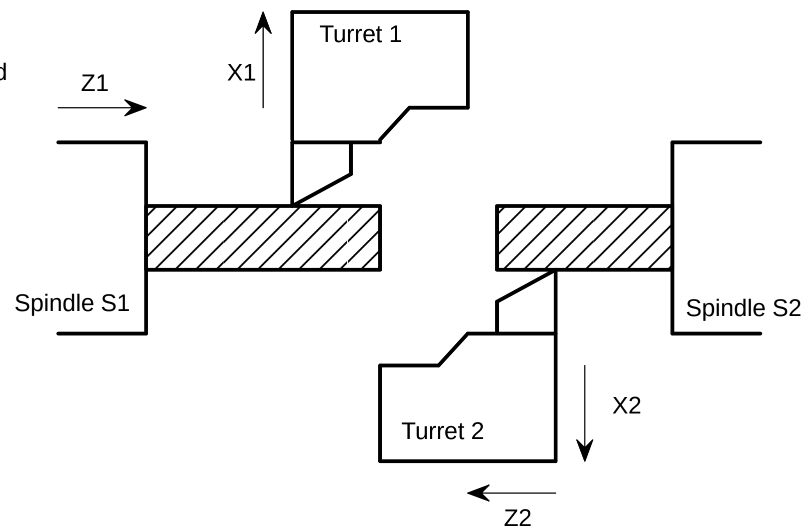
It is not always necessary to cause a coordinate system to be set up automatically when composite control begins or ends. If automatic coordinate system setting is not specified, an appropriate coordinate system is set by program.

1.9.9.4
Independent control and superimposed control for the Z1– and Z2–axes

(1) Machine configuration

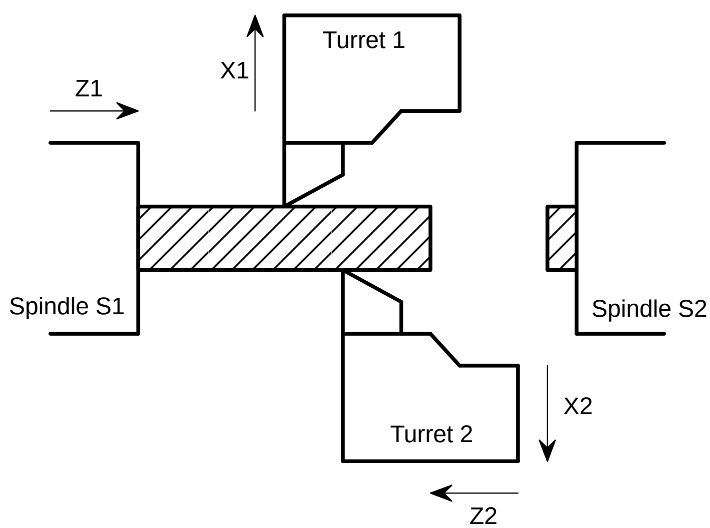
(a) Independent control

Path 1 (X1– and Z1–axes) performs machining sepa-rately from path 2 (X2– and Z2–axes).



(b) Superimposed control for the Z1– and Z2–axes

A workpiece mounted on spindle S1 is machined using paths 1 (X1– and Z1–axis) and 2 (X2– and Z2–axes).



(2) Parameter setting

- To superimpose the move commands for the Z1–axis on the Z2–axis, set parameter No. 8186z of path 2 to “2”.
- To apply mirror–imaged superimposed control (because the positive directions of the Z1– and Z2–axes do not match), set parameter No. 8162 (OMRz) of path 2 to “1”.

- Set the feedrate along each Z–axis for superimposed control in parameter Nos. 8190z, 8191z, and 8193 of both paths. Each value to be set must be about half the one for independent control.
- Set the rapid traverse time constant for each Z–axis under superimposed control in parameter No. 8192z of both paths. Each value to be set must be 1 to 2 times the one for independent control.

(3) Signal setting

- Set signal G1190#1 OVLS2 to “1” when superimposed control begins for the Z1– and Z2–axes.
- Reset signal G1190#1 OVLS2 to “0” when superimposed control ends.
- Also reset G1190#1 OVLS2 to “0” when an emergency stop, NC reset, or alarm occurs.
- Keep signals other than G1190#1 OVLS2 reset to “0”.

(4) Sample program

Path 1	Path 2	
N1000 ········	N2000 ········	Machining under control independent of the other path
N1010 M300 ;	N2010 M300 ;	Waits for superimposed control to begin.
N1020 M55 ;		Begins superimposed control in which commands for the Z1–axis are superimposed on those for the Z2–axis.
N1030 M301 ;	N2030 M301 ;	Superimposed control has begun.
	N2040 T0414 ;	Selects a tool for superimposed control and sets the offset.
N1050 S1000 M3 ;		Performs machining with turrets 1 and 2.
N1060 G0 X20. Z15. ;	N2060 G0 X18. Z120. ;	
N1070 G1 F0. 5 W- 8. ;	N2070 G1 F0. 1 W5. ;	
N1080 ······	N2080 ······	
N1090 M302 ;	N2090 M302 ;	Waits for superimposed control to end.
N1100 M56 ;		Terminates superimposed control.
N1110 M303 ;	N2110 M303 ;	Superimposed control has ended.

N1120 ·······	N2120 ········	Machining under control independent of the other path

> **WARNING**
> When using constant surface speed control, be careful about which path has the spindle command that is effective for spindle S1.

> **NOTE**
> The speed of spindle S1 (feedback pulses from the position coder) is specified for both paths 1 and 2.

1.9.9.5 Miscellaneous

- Synchronous control and composite control were described so far. In reality, however, it is possible to perform more than one set of synchronous control and/or composite control selectively or simultaneously. For this purpose, specify all necessary parameters and select which synchronous control or composite control to be performed using the appropriate signals. However, be careful not to perform more than one set of synchronous control or composite control for one axis at one time.
- Usually, it is possible to specify only one pair of axes for synchronous control and one pair for composite control. If it is necessary to specify more than one pair, specify so in a parameter with a program, using the programmable parameter input function (G10). This must be done when the related axes are not under synchronous or composite control.

(Example) Changing the parameter so that the Z2–axis is synchronized with the Y1–axis

(To set parameter No. 8180z of path 2 to “4”, run the following program in path 2.)

N0200 ········	
N0210 G10 L50 ;	Begins parameter setting.
N0220 N8180 P2 R4 ;	Sets parameter No. 8180z to “4”.
N0230 G11 ;	Terminates parameter setting.
N0240 ········	

The blocks with G10 to G11 must be run when the Z2– or Y1–axis is not under synchronous or composite control.

1.9.10 Troubleshooting

1.Synchronous, composite, or superimposed control cannot be started, but no alarm is issued.

(1) The synchronous or composite control option has not been specified.

⇒ The synchronous and composite control must be specified.

(2) The G0128, G0138, G0190, G1128, G1138, or G1190 signal has not risen.

⇒ Synchronous, composite, or superimposed control begins on the positive–going edge of the G0128, G0138, G0190, G1128, G1138, or G1190 signal. If synchronous, composite, or superimposed control ends because of a reset or alarm, merely releasing the reset or alarm cannot restart synchronous, composite, or superimposed control. It is also necessary to raise the signal.

(3) The axis number of an axis to be subjected to synchronous, composite, or superimposed control has not been specified in a parameter.

⇒ To use synchronous control, specify the axis number of the target master axis in parameter No. 8180. To use composite control, specify the axis number of the target axis in parameter No. 8183 of path 2. To use superimposed control, specify the axis number of the target master axis in parameter No. 8186.

(4) Synchronous, composite, or superimposed control cannot be started when the NC unit is under one of the following conditions.

- Emergency stop
- Reset
- Servo alarm
- P/S000 alarm
- Alarm related to synchronous, composite, or superimposed control

In addition, synchronous, composite, or superimposed control cannot be started when the NC unit is under one of the following conditions.

- Servo–off
- Overtravel

2. The P/S225 alarm occurs when a signal for synchronous, composite, or superimposed control arises.

   (1) An attempt was made to perform synchronous, composite, or superimposed control for an axis that was already under synchronous, composite, or superimposed control.
   ⇒ It is impossible to place an axis under more than one combination of synchronous, composite, and/or superimposed control simultaneously. However, a synchronous master axis can be the master of more than one synchronous slave axis, and other slave axes can be added under the same synchronous control.

   (2) The axis number specified in a parameter is greater than the number of controllable axes.
   ⇒ The axis number of a synchronous master axis, an axis under composite control, or the master axis under superimposed control in one path must not be greater than the number of controllable axes in the other path (or in the same path if synchronous control is performed within one path).

   (3) An axis to be placed in synchronization is already moving.
   ⇒ When synchronous, composite, or superimposed control begins, the target axis must be at a stop. An axis being at a stop means that the speed that is effective after acceleration/deceleration is zero.

3. The P/S225 alarm occurs when synchronous, composite, or superimposed control ends.

   (1) An axis to be released from synchronization is moving.
   ⇒ When synchronous, composite, or superimposed control ends, the target axis must be at a stop. An axis being at a stop means that the speed that is effective after acceleration/deceleration is zero. When terminating synchronization, make sure that the axis moving signal F0102/F1102 is “0”.

NOTE
Before the P/S225 alarm that occurs when synchronous, composite, or superimposed control is terminated can be reset, it is necessary to place the machine in an emergency stop state.

4. The P/S226 alarm occurs during synchronous or composite control.

   (1) A move command was issued to a synchronous slave axis.
   ⇒ A move command (either automatic or manual) cannot be used for a synchronous slave axis.

(2) A move command was issued to an axis under composite control for which parameter MUMx (bit 7 of parameter No. 8162) is “1”.

⇒ No move command (either automatic or manual) can be issued to an axis under composite control for which parameter MUMx (bit 7 of parameter No. 8162) is “1”.

5. Servo alarm No. 407 occurs during synchronous control.

(1) There is an excessive difference in the positional deviation between the synchronous master and slave axes.

⇒ Alarm SV407 can occur only when a check is being made for synchronous error. This alarm occurs typically when there is a large difference in acceleration/deceleration constants or servo parameters between the synchronous master and slave axes or when the actual machine movement is incorrect for any reason (such as incorrect synchronization).

6. The machine position deviates during synchronous control.

(1) The acceleration/deceleration constants or servo parameters (such as loop gain) do not match between the master and slave axes.

⇒ During synchronous control, acceleration/deceleration and servo control are performed for master and slave axes separately. (Instead, move commands are placed in synchronization.) Acceleration/deceleration time constants or servo characteristics may vary between the master and slave axes. In such cases, the actual machine movement does not match between axes.

7. The amount of movement is incorrect during synchronous, composite, or superimposed control.

(1) The diameter/radius specification or inch/metric input setting does match between the master and slave axes.

⇒ Synchronous, composite, superimposed control does not make conversion on diameter/radius or inch/metric input specifications between the master and slave axes. The least command input must match between the master and slave axes.

8. An axis does not move to a specified position after synchronous or composite control switching.

   (1) A move command was issued within two blocks after synchronous or composite control.

   ⇒ The coordinate system in the CNC must be re–set at synchronous or composite control switching. No move command can be issued to an axis subjected to synchronous or composite control during automatic operation within two (or three for tool–tip radius compensation) blocks (including the current one) after synchronous or composite control switching. However, this restriction does not apply when the current block is an M code that does not buffer the next block or when the target axis is a synchronous master axis.

9. Synchronous, composite, or superimposed control was terminated when the G0128, G0138, G0190, G1128, G1138, or G1190 did not drop.

   ⇒ Synchronous, composite, or superimposed control is terminated automatically, if one of the following conditions occurs in either path.

   - Emergency stop
   - Reset
   - Servo alarm
   - P/S000 alarm
   - Alarm related to synchronous, composite, or superimposed control

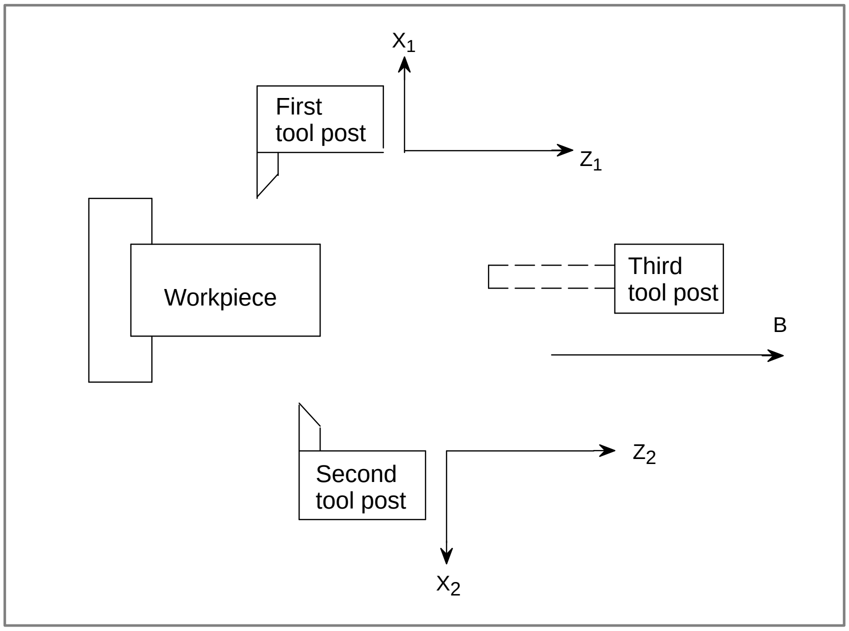
   In addition, synchronous, composite, or superimposed control is terminated for all axes, if one of the following conditions occurs for any axis under synchronous, composite, or superimposed control.

   - Servo–off
   - Overtravel

1.10 B–AXIS CONTROL (T SERIES)

General

This function sets an axis (B–axis) independent of the basic controlled axes X_1, Z_1, X_2, and Z_2 and allows drilling, boring, or other machining along the B–axis, in parallel with the operations for the basic controlled axes. The X_2 and Z_2 axes can be used in two–path control mode



Format

- **Registering operation programs**

G101–G100 : Starts registering the first program.
G102–G100 : Starts registering the second program.
G103–G100 : Starts registering the third program.
G100 : Ends registering of the programs.

Three operations (programs) on the B–axis can be registered. (In two–path control mode, three programs can be registered for each tool post.)
The B–axis operation program must be specified in the blocks between G101, G102, or G103 and G100, allowing it to be discriminated from the normal NC program.
The registered operation is started upon executing the corresponding M code, described below.

```
O1234 ;
  :          Normal NC program
G101 ;  ------------------------  Starts registering of a B–axis
                                  operation program.
  :          B–axis operation program
G100 ;  ------------------------  Ends registering of the B–axis
                                  operation program.
  :          Normal NC program
M30 ;
```

Note) In the block of G101, G102, G103, or G100, specify no other codes.

- **Command used to start the operation**

To start an operation, the miscellaneous functions (M**) specified in parameters 8251 to 8253 are used.

Parameter 8251:
M code used to start operation of the first program
Parameter 8252:
M code used to start operation of the second program
Parameter 8253:
M code used to start operation of the third program

```
O1234  ;
  :
M** ; -        Starts executing the registered B–axis operation.  In
               subsequent blocks, the normal NC program and the
  :            B–axis operation program are executed in parallel.
               (** is specified in parameters 8251 to 8253.)
M30  ;
```

Example

```
01234 ;
G50 X100. Z200. ;
G101 ;                  ---- [1] Starts registering of an
                                 operation program.
G00 B10. ;              -+
M03 ;                    |
G04 P2500 ;              +-- [2] Blocks of the B–axis
G81 B20. R15. F500 ;     |       operation program
G28 ;                   -+
G100 ;                  ---- [3] Ends registering of the
                                 operation program.
G00 X80. Z50. ;
G01 X45. F1000 ;
  :
G00 X10. ;
M** ;                   ---- [4] Command used to start the
                                 programmed operation
G01 Z30. F300 ;
  :
M30 ;
```

[1] to [3] : Specify the B–axis operation program in blocks between G101, G102, or G103 and G100. The program is registered in program memory.

[4] : Starts executing the B–axis operation registered with [1] to [3] above. In subsequent blocks, the normal NC operation and the B–axis operation are executed in parallel. An M code of the miscellaneous function is used to start the B–axis operation. The M code, used to start the operation, is specified in parameters 8251 to 8253.

- **Single–motion operation**

G110 [operation command];

A single–motion operation for the B–axis can be specified and executed as shown above. Such an operation need not be registered as a special (first to third) program. Nor does it need to be by a special command, as described above.

Explanations

- **Specifying two–path control mode**

One of the following three two–path control modes can be selected:

1 B–axis control is executed for either tool post 1 or 2.

2 B–axis control is executed separately for tool posts 1 and 2.

3 Identical B–axis control is executed for tool posts 1 and 2.

The mode is selected according to the value specified for parameter 8250 for each tool post.

- **Codes that can be used in a B–axis operation program**

The following 13 G codes, and the M, S, and T codes of the miscellaneous functions, can be used in a B–axis operation program:

Code	Description
G00	Positioning (rapid traverse)
G01	Linear interpolation (cutting feed)
G04	Dwell
G28	Reference position return, automatic coordinate system setting
G80	Canned cycle, cancel
G81	Drilling cycle, spot drilling
G82	Drilling cycle, counterboring
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G98	Feed per minute
G99	Feed per rotation
M**	Auxiliary function
S**	Auxiliary function
T**	Auxiliary function, tool offset

G28 (reference position return)
Unlike the normal G28 cycle, the G28 cycle for a B–axis operation does not include intermediate point processing. For example, the following cannot be specified:

G28 B99.9;

G80 to G86 (canned drilling cycle)
Of the canned drilling cycles supported by the CNC for machining centers, those cycles equivalent to G80 to G86 can be executed.
Data can be specified in the same way as for the CNC for machining centers, except for the following points:

1. The drilling position is not specified with X and Y.
2. The distance from point R to the bottom of the hole is specified with B.

3. All operations are executed in the initial level return mode.
4. The repetition count (K) cannot be specified.
5. In canned cycle mode, point R must be specified. (If point R is omitted, P/S alarm No. 5036 is output.)
6. The drilling start point (d) for the G83 (peck drilling) cycle is specified with parameter 8258.

G98, G99 (feed per minute, feed per rotation)

The MDF bit (bit 2 of parameter 8241) specifies an initial continuous–state G code for G110, or the G code to start registration of the operation program (G101, G102, G103).
When the MDF bit is set to 0, the initial continuous–state code is G98.
When the MDF bit is set to 1, the initial continuous–state code is G99.

Example)
When MDF is set to 0
G110 B100. F1000. ; 1000 mm/min
G110 G99 B100. F1 ; 1 mm/rev

> **NOTE**
> In two–path control mode, the system uses the actual spindle speed, calculated from the feedback signal output by the position coder connected to the tool post to which the controlled axis belongs.

M, S, and T codes (auxiliary functions)

According to a numeric value subsequent to address M, S, or T, the binary code and strobe signal are sent to the PMC. The codes and signals for addresses M, S, and T are all output to an identical interface (auxiliary function code signals (EM 11g to EM 48g) and auxiliary function strobe signals (EMFg)) and can be used to control on or off of the PMC machine. For this purpose, the PMC axis control interface is used, which differs from that used for the miscellaneous functions for the normal NC program. The following M codes, used to control the spindle, are automatically output during the G84 (tapping) or G86 (boring) cycle:
M03: Forward spindle rotation
M04: Reverse spindle rotation
M05: Spindle stop

T** to T(** + 9), where ** is the number specified in parameter 8257, are used as the codes of the auxiliary functions to adjust the tool offset.

Example)
T50 to T59 if parameter 8257 is set to 50

> **NOTE**
> 1 Range of commands of M, S, and T codes
> 2 An M, S, or T code must not be specified in a block containing another move command. The M, S, and T codes must not be specified in an identical block.
> 3 Usually, normal NC operation and B–axis operation are independent of each other. Synchronization between operations can be established by coordinating the miscellaneous functions of the normal NC program and B–axis operation program.

```
(Normal NC operation)   (Registered B–axis operation)
    :                              :
M11 ;                          G00 B111 ;
G01 X999 :                     G01 B222 ;
G28 Z777 ;                     G28 ;
M50 ;                          M50 ;
G00 X666 ;                     G81 B444 R111 F222 ;
    :                              :
```

Upon receiving M50 of both the normal NC program and the B–axis program in the PMC ladder, the completion signals (FIN and EFINg) are turned ”1”. G00 X666 of the normal NC program and G81 B444 R111 F222 of the B–axis program are executed simultaneously.

Custom macro

Custom macro variables (local variables, common variables, system variables #****) can be used in an operation program between G101, G102, or G103 and G100.

1. The value of the macro variable is calculated not from the data existing upon execution of the B–axis operation, but from the data existing at registration of the operation program.
2. An instruction that causes a branch to a location beyond the range of G101, G102, or G103 to G100 is processed without being checked.
3. In the two–path control mode, tool posts 1 and 2 use different macro variables.

• Operation program

When a new operation program is registered, the previous operation program is automatically deleted.
If an error is detected in an operation program to be registered, the program is initialized but is not registered.

• Modal

In the same way as a normal NC program, the B–axis operation program can use the following as modal data: modal G codes, F codes, and P, Q, and F codes in the canned cycle. These codes do not affect the modal information of the normal NC program. When a B–axis operation program is started (by G101, G102, or G103), the initial modal data is set for the program. It is not affected by the previous modal information.

Example)

```
    :
G01 X10. F1000 ;              [1]
G101 (G102, G103) ;           [2]
B10. ;                        [3]
G01 B–10. F500 ;              [4]
G100 ;                        [5]
X–10. ;                       [6]
    :
```

Irrespective of the modal information for normal operation (G01 specified in block), block [3] specifies G00 if the MDG bit (bit 1 of parameter 8241) is set to 0, or G01 if the MDG bit is set to 1.
Block [6] causes movement with F1000, specified in block [1].

- **Operation start command**

The MST bit (bit 7 of parameter 8240) specifies the method used to start the B–axis operation as described below:
If the MST bit is set to 1, the B–axis operation is started when the M code to start the operation is executed.
If the MST bit is set to 0, the B–axis operation is started when the M code used to start the operation is executed and the PMC outputs the completion signal (FIN).
Up to five M codes for starting the programs can be stored. The programs corresponding to these M codes are executed in succession. (In two–path control mode, up to five codes can be stored for each tool post.)

Example)
When the first, second, and third programs are started by M40, M41, and M42, respectively

```
O1234. ;
    :
    :
M40 ;   M code for starting the first program
M41 ;   M code for starting the second program
M42 ;   M code for starting the third program
M40 ;   M code for starting the first program
M41 ;   M code for starting the second program
    :
    :
M30 ;
```

As M41 is specified while the program started by M40 is being executed, the second program is automatically started upon termination of the first program.
M42, M40, and M41, specified during execution of the first program, are stored such that the corresponding programs are executed in the same order as that in which the M codes are specified.
If six or more M codes for starting the programs are specified while a program is being executed, P/S alarm 5038 is output.
In two–path control mode, the M code specified for tool post 1 starts the B–axis program registered for tool post 1. The M code specified for tool post 2 starts the B–axis program registered for tool post 2.

- **Specifying absolute or incremental mode**

The amount of travel along the B–axis can be specified in either absolute or incremental mode. In absolute mode, the end point of travel along the B–axis is programmed. In incremental mode, the amount of travel along the B–axis is programmed directly.
The ABS bit (bit 6 of parameter 8240) is used to set absolute or incremental mode. When the ABS bit is set to 1, absolute mode is selected. When the ABS bit is set to 0, incremental mode is selected. The mode is specified with this parameter when the program is registered.

- **Specifying a tool offset**

The T**; command shifts the end point of the specified B–axis travel, in either the positive or negative direction, by the amount specified with the B–axis offset screen. If this function is used to set the difference between the programmed tool position and actual tool position in machining, the program need not be modified to correct the tool position.
The value specified with parameter 8257 is assigned to the auxiliary function to cancel the offset. The subsequent nine numbers are assigned to the tool offset functions. These auxiliary function numbers are displayed on the B–axis offset screen. For details, see Operator's Manual.

- **Single–motion operation**

If a G110 block is specified, a single–motion operation along the B–axis can be specified and executed. In single–motion operation mode, a single block results in a single operation. The single–motion operation is executed immediately provided if it is specified before the B–axis operation is started. If the operation is specified while a registered program is being executed, the operation is executed once that program has terminated.
After the specified single–motion operation has been executed, the next block is executed.

```
  :
G110 G01 B100. F200 ;   Block for single–motion
                        operation along B–axis
G00 X100. Z20. ;
  :
```

- **Program memory**

An operation program is registered in program memory as a series of different blocks of the move, dwell, auxiliary, and other functions. Program memory can hold a desired number of blocks, up to a maximum of 65535 blocks for each program. If the program memory contains no free space when an attempt is made to register a B–axis program, P/S alarm 5033 is output. Six blocks require 80 characters of program memory. A canned cycle (G81 to G86) is also registered as a series of blocks, such as travel and dwell.
The entire program memory is backed up by battery. The programs registered in program memory are thus retained even after the system power is turned off. After turning the system power on, the operation can be started simply by specifying the M code for starting the program.

Example)
:
G101 ;
G00 B10. ; . . . . One block
G04 P1500 ; . . . . One block
G81 B20. R50. F600 ; . . . . Three blocks
G28 ; . . . . One block
M15 ; . . . . One block
G100 ;
: (Total 7 blocks)

- **Reset**

When the NC is reset by pressing the MDI reset key or by the issue of an external reset signal, reset and rewind signal, or emergency stop, B–axis control is also reset. The reset signal (ECLRg) can reset only B–axis control.

- **PMC–controlled axis**

A B–axis operation can be executed only when the B–axis can be controlled by the PMC. For details, refer to the manual supplied by the machine tool builder.

Limitations

- **Single–motion operation**

1. Only a single–motion operation can be specified with G110.

   G110 G00 B100. ; . . . . OK
   G110 G28 ; . . . . OK
   G110 G81 B100. R150.0 F100 ; . . . P/S alarm No.5034

2. A canned cycle (G81 to G86), and other operations containing multiple motions, cannot be specified with G110.
   If an inhibited operation is specified, P/S alarm No.5034 is output.

3. modal information specified with G110 does not affect the subsequent blocks. In the G110 block, the initial modal value specified at the start of the operation becomes valid, irrespective of the modal information specified the previous blocks.

   Example)
   When the MDG bit (bit 1 of parameter 8241) is set to 1 and the MDF bit (bit 2 of parameter 8241) is set to 1
   G98 G00 X100. F1000 ; . . . . (1)
   G110 B200. F2.; . . . . (2)
   X200. ; . . . . (3)
   G01 X200. ; . . . . (4)

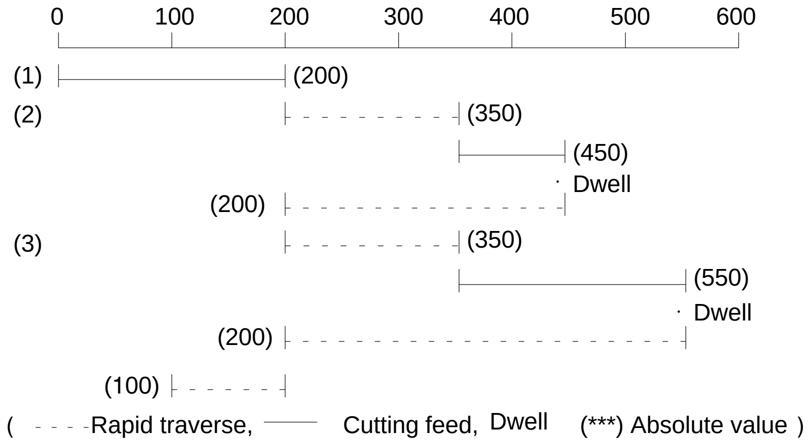
   Block (2) instigates cutting feed (G01) at 2.0 mm/rev (G99).
   Block (3) instigates rapid traverse (G00).
   Block (4) instigates cutting feed (G01) at 1000 mm/min (G98).

4. During tool–nose radius compensation, two or more G110 blocks cannot be specified in succession. If such blocks are specified in succession, P/S alarm No. 5041 is output. To specify two or more G110 blocks in succession for a B–axis operation, register the blocks as a program with G101, G102, or G103 and G100.

Examples

- **Absolute or incremental mode**

Absolute or incremental mode



Incremental mode

```
  G101 (G102, G103) ;
(1) G01 B200. F100 ;
(2) G82 B100. R150. P5000 F200 ;
(3)     B200. R150. P5000 ;
(4) G00 B–100. ;
   G100 ;
      :
   M** ;
      :
   M30 ;
```

Absolute mode

```
  G101 (G102, G103) ;
(1) G01 B200. F100 ;
(2) G82 B450. R350. P5000 F200 ;
(3)     B550. R350. P5000 ;
(4) G00 B100. ;
   G100 ;
      :
   M** ;
      :
   M30 ;
```

- **Tool posts 1 and 2**

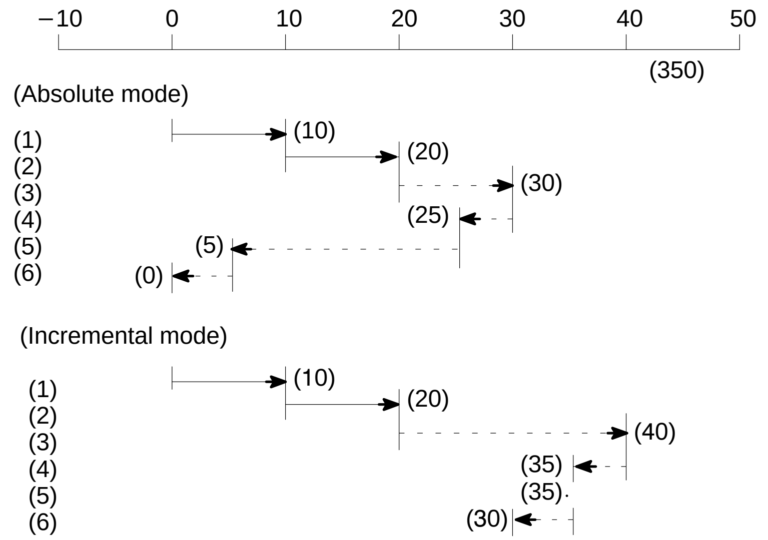
If a single axis is used as the common B–axis of the two tool posts in two–path control, tool posts 1 and 2 share the B coordinate.
For example, after program 1 for tool post 1 and program 2 for tool post 2 are executed in that order, the total travel along the B–axis appears to be +100.

```
 <Program 1>
   G101 ;
      :
   G00 B200. ; (Absolute mode)
   G100 ;
      :
   M30 ;

 <Program 2>
   G101 ;
   G00 B300. ; (Absolute mode)
      :
   G100 ;
      :
   M30 ;
```

- **Tool offset**

Example)
When parameter 8257 is set to 50
Auxiliary function used to cancel the offset: T50
Auxiliary functions used to adjust a tool offset: T51 to T59



Program
```
   G101 (G102, G103) ;
(1)  G01 B10. F100 ;
(2)  T51 ;
(3)  G00 B20. ;
(4)  T52 ;
(5)  B0. ;
(6)  T50 ;
   G100 ;
     :
   M**;
     :
```

Where the offset of T51 is 10.0 and the offset of T52 is 5.0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8240	MST	ABS	SOV	TEM	REF			

[Data type] Bit

REF Reference position return operation by G28:

0 : Always uses deceleration dogs in the same way as a manual reference position return operation.
1 : Uses deceleration dogs when a reference position has not yet been set, but is performed by rapid traverse when a reference position has already been set (in the same way as an ordinary G28 command).

TEM When an offset movement is made in a block containing a T code:

0 : Auxiliary function code signal and auxiliary function strobe signal are output before a movement along an axis.
1 : Auxiliary function code signal and auxiliary function strobe signal are output after a movement along an axis.

SOV A G110 block:

0 : Overlaps the next block.
1 : Does not overlap the next block.

ABS The B–axis command is:

0 : An incremental command.
1 : An absolute command.

MST When an M code for starting a movement along the B–axis is specified:

0 : Operation is started after a ready notice using the FIN signal is received.
1 : Operation is started without waiting for a ready notice.

	#7	#6	#5	#4	#3	#2	#1	#0
8241						MDF	MDG	FXC

[Data type] Bit

FXC In canned cycle G84:

0 : The spindle is rotated clockwise or counterclockwise after M05 is output.
1 : The spindle is rotated clockwise or counterclockwise without first outputting M05.

MDG The initial continuous–state value for starting B–axis operation command registration is:

0 : G00 mode (rapid traverse).
1 : G01 mode (cutting feed).

MDF The initial continuous–state value for starting B–axis operation command registration is:

0 : G98 (feed per minute).
1 : G99 (feed per rotation).

	#7	#6	#5	#4	#3	#2	#1	#0
8242								COF

[Data type] Bit

COF For tool post 1 and tool post 2 (under two–path control):

0 : A separate B–axis offset value is set.
1 : A common B–axis offset value is set.

8250	Axis number used for B–axis control

[Data type] Byte

[Valid data range] 1 to number of controlled axes (in one–system control)
11 to ((number of controlled axes for tool post 1) + 10), or
21 to ((number of controlled axes for tool post 2) + 20) (in two–path control)

This parameter sets which axis is to be used for B–axis control.

In one–system control, set the controlled axis number of a selected B–axis.

In two–path control, set the axis number, used for B–axis control on tool post 1, added to 10 when a tool post 1 axis is used.

Set an axis number, used for B–axis control on tool post 2, added to 20 when a tool post 2 axis is used.

Example of setting:

(1) For one–system control
When the fourth axis is controlled as the B–axis, set 4 in this parameter. Furthermore, specify a DI/DO number to be used for the fourth axis in parameter No. 8010.

(2) For two–path control

(a) When B–axis control is applied to tool post 1 only
When the fourth axis of tool post 1 is controlled as the B–axis, set 14 with this parameter of tool post 1. Furthermore, specify the DI/DO number to be used for the fourth axis with parameter No. 8010 for tool post 1.

(b) When B–axis control is applied to tool post 2 only
When the fourth axis on tool post 2 is controlled as the B–axis, set 24 with this parameter of tool post 2. Furthermore, specify a DI/DO number to be used for the fourth axis in parameter No. 8010 for tool post 2.

(c) When B–axis control is applied separately to tool post 1 and tool post 2
Make the settings described in (a) and (b) above.

(d) When B–axis control is simultaneously applied to both tool post 1 and tool post 2
When the fourth axis for tool post 1 is controlled as the common B–axis, set 14 with this parameter for both tool post 1 and tool post 2. Furthermore, specify a DI/DO number to be used for the fourth axis in parameter No. 8010 for tool post 1.

8251	M code (G101) for specifying the start of first program operation
8252	M code (G102) for specifying the start of second program operation
8253	M code (G103) for specifying the start of third program operation

[Data type] 2–word

[Valid data range] 6 to 99999999

These parameters set M codes for starting previously registered B–axis operation programs. M codes (such as M30, M98, and M99), already used for other purposes, cannot be set.

8257	T code number for tool offset cancellation

[Data type] Byte

[Valid data range] 0 to 90

This parameter sets a T code number for tool offset cancellation. When a T code from (setting + 1) to (setting + 9) is specified, tool offset is specified.

8258	Clearance, used in canned cycle G83, for the B–axis

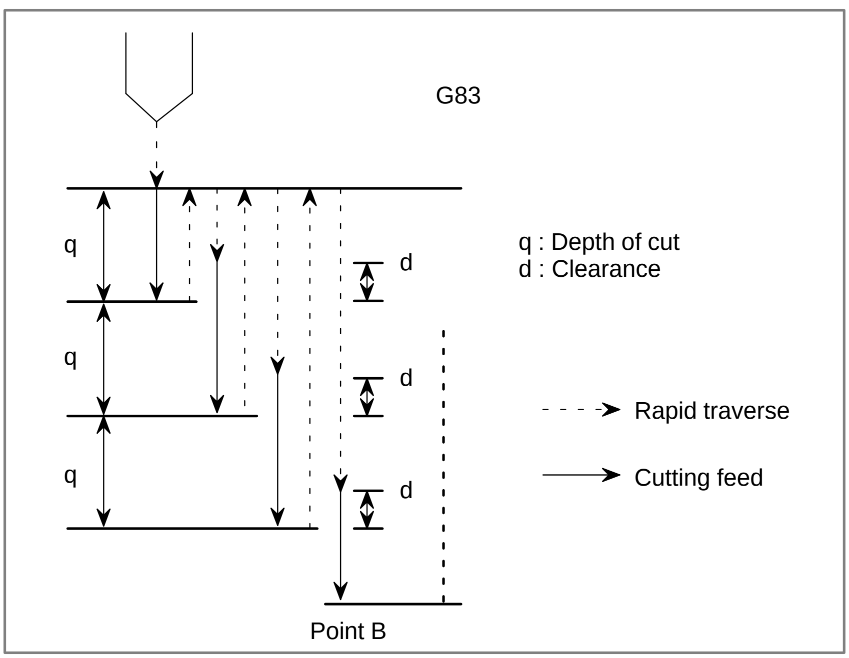
[Data type] 2–word

[Valid data range] 0 to 99999999

[Unit of data]

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

This parameter sets the clearance used for peck drilling cycle G83.



Alarm and message

Number	Message	Contents
5030	ILLEGAL COMMAND (G100)	The end command (G110) was specified before the registration start command (G101, G102, or G103) was specified for the B–axis.
5031	ILLEGAL COMMAND (G100, G102, G103)	While a registration start command (G101, G102, or G103) was being executed, another registration start command was specified for the B–axis.
5032	NEW PRG REGISTERED IN B–AXS MOVE	While the machine was moving about the B–axis, at attempt was made to register another move command.
5033	NO PROG SPACE IN MEMORY B–AXIS	Commands for movement about the B–axis were not registered because of insufficient program memory.
5034	PLURAL COMMAND IN G110	Multiple movements were specified with the G110 code for the B–axis.
5035	NO FEEDRATE COMMANDED B–AXIS	A feedrate was not specified for cutting feed about the B–axis.
5036	ADDRESS R NOT DEFINED IN G81–G86	Point R was not specified for the canned cycle for the B–axis.
5037	ADDRESS Q NOT DEFINED IN G83	Depth of cut Q was not specified for the G83 code (peck drilling cycle). Alternatively, 0 was specified in Q for teh B–axis.
5038	TOO MANY START M–CODE COMMAND	More than six M codes for starting movement about the B–axis were specified.
5039	START UNREGISTERED B–AXIS PROG	An attempt was made to execute a program for the B–axis which had not been registered.
5040	CAN NOT COMMANDED B–AXIS MOVE	The machine could not move about the B–axis because parameter No.8250 was incorrectly specified, or because the PMC axis system could not be used.
5041	CAN NOT COMMANDED G110 BLOCK	Blocks containing the G110 codes were successively specified in tool–tip radius compensation for the B–axis.

Caution

CAUTION

B–axis control uses the PMC axis control interface. This means that, before a B–axis command can be executed, the axis used as the B–axis must be placed under PMC axis control. To specify operations such as single block and feed hold, use PMC axis control signals.
Referring to Section 15.1, set the PMC axis control parameters and signals.

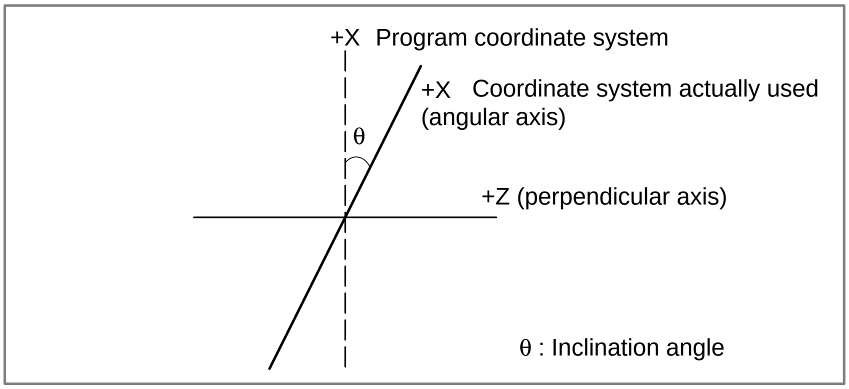
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.2.7	Displaying the B–axis Operation State
		III.11.4.15	Setting and Displaying B–axis Tool Compensation
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.2.6	Displaying the B–axis Operation State
		III.11.4.15	Setting and Displaying B–axis Tool Compensation
	CONNECTION MANUAL (This manual)	15.1	PMC Axis Control/PMC Axis Speed Control Function

1.11 ANGULAR AXIS CONTROL/ ARBITRARY ANGULAR AXIS CONTROL

General

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



Explanations

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

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

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

$$Za = Zp – Xp \tan\theta$$

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

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

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

- **Method of use**

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

- **Absolute and relative position display**

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

- **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

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

[Classification] Input signal

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

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

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

Signal address

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

Parameter

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

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

[Data type] Bit

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

AZR 0 : The machine tool is moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.
1 : The machine tool is not moved along the Cartesian axis during manual reference position return along the slanted axis under angular axis control.

8210	Inclination angle for angular axis control

[Data type] 2 words

[Unit of data] 0.001 degree

[Valid data range] 20000 to 60000

8211	Axis number of a slanted axis subject to slanted axis control
8212	Axis number of a Cartesian axis subject to slanted axis control

[Data type] Word

[Unit of data] Number

[Valid data range] 1 to number of controlled axes

These parameters set the axis numbers of a slanted axis and Cartesian axis subject to slanted axis control.

Warning

WARNING

1 After angular axis control parameter setting, be sure to perform manual reference point return operation.
2 If bit 2 (AZK) of parameter No. 8200 has been set to 0, such that manual reference position return along the angular axis also causes movement along the perpendicular axis, once manual reference position return has been performed along the angular axis, also perform manual reference position return along the perpendicular axis.
3 Once the tool has been moved along the angular axis with perpendicular/angular axis control disable signal NOZAGC set to 1, manual reference position return must be performed.
4 Before attempting to manually move the tool along both the angular and perpendicular axes simultaneously, set perpendicular/angular axis control disable signal NOZAGC to 1.

Note

NOTE

1 If an inclination angle close to 0° or ±90° is set, an error can occur. A range from ±20° to ±60° should be used.
2 Before a perpendicular axis reference point return check (G27) can be made, angular axis reference point return operation must be completed.
3 For arbitrary angular axis control, if the same axis number has been specified in both parameters No. 8211 and 8212, or if a value outside the valid data range has been specified for either parameter, the angular and perpendicular axes will be as follows:
Angular axis: First axis
Perpendicular axis: Second axis

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.5	Augular axis control/axis control
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.19.6	Augular axis control/axis control
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.19.5	Augular axis control

1.12 CHOPPING FUNCTION (M SERIES)

General

When contour grinding is performed, the chopping function can be used to grind the side face of a workpiece. By means of this function, while the grinding axis (the axis with the grinding wheel) is being moved vertically, a contour program can be executed to instigate movement along other axes.

In addition, a servo delay compensation function is supported for chopping operations. When the grinding axis is moved vertically at high speed, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The servo delay compensation function compensates for any displacement by increasing the feedrate. Thus, grinding can be performed almost up to the specified position.

There are two types of chopping functions: that specified by programming, and that activated by signal input. For details of the chopping function activated by signal input, refer to the manual provided by the machine tool builder.

Format

G81.1 Z__ Q__ R__ F__ ;

Z : Upper dead point
(For an axis other than the Z–axis, specify the axis address.)
Q : Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value, relative to the upper dead point.)
R : Distance from the upper dead point to point R
(Specify the distance as an incremental value, relative to the upper dead point.)
F : Feedrate during chopping

G80; Cancels chopping

Explanations

• Chopping activated by signal input

Before chopping can be started, the chopping axis, reference position, upper dead point, lower dead point, and chopping feedrate must be set using the parameter screen (or the chopping screen).

Chopping is started once chopping start signal CHPST has been set to 1. This signal is ignored, however, during chopping axis movement.

When chopping hold signal *CHLD is set to 0 during chopping, the tool immediately moves to point R. Again setting the chopping hold signal to 1 restarts chopping.

Chopping can also be stopped by setting chopping start signal CHPST to 0, but only when chopping was started by using that signal.

Methode of starting chopping	Methode of stopping chopping	State
Signal CHPST = 1	Signal CHPST = 0	Stopped
	G80	Stopped
G81.1	Signal CHPST = 0	Not stopped
	G80	Stopped

NOTE

1 Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop chopping.
2 In chopping mode, a chopping axis move command or canned cycle command cannot be specified.
3 If a G81.1 command is specified during chopping started by the signal, chopping is not stopped. If point R, the upper dead point, lower dead point, or chopping feedrate has been modified by using the G81.1 command, chopping is continued, but using the modified data.
4 The use of chopping start signal CHPST to start chopping is not enabled immediately after power–on; it is not enabled until the completion of manual reference position return.

- **Chopping feedrate (feedrate of movement to point R)**

From the start of chopping to point R, the tool moves at the rapid traverse rate (specified by parameter No. 1420).

The override function can be used for either the normal rapid traverserate or chopping feedrate, one of which can be selected by setting CPRPD (bit 0 of parameter No. 8360).

When the chopping feedrate is overridden, settings between 110% and 150% are clamped to 100%.

- **Chopping feedrate (feedrate of movement from point R)**

Between point R, reached after the start of chopping, and the point where the chopping is canceled, the tool moves at the chopping feedrate (specified by parameter No. 8374).

The chopping feedrate is clamped to the maximum chopping feedrate (set with parameter No. 8375) if the specified feedrate is greater than the maximum chopping feedrate.

The feedrate can be overridden by 0% to 150% by applying the chopping feedrate override signal.

- **Setting chopping data**

Set the following chopping data:

- Chopping axis: Parameter No. 8370
- Reference point (point R): Parameter No. 8371
- Upper dead point: Parameter No. 8372
- Lower dead point: parameter No. 8373
- Chopping feedrate: Parameter No. 8374
- Maximum chopping feedrate: Parameter No. 8375

All data items other than the chopping axis and maximum chopping feedrate can be set on the chopping screen.

For details of how to set chopping data on the chopping screen, refer to the operator's manual (III 11.4.13 Displaying and Setting Chopping Data.)

- **Chopping after the upper dead point or lower dead point has been changed**

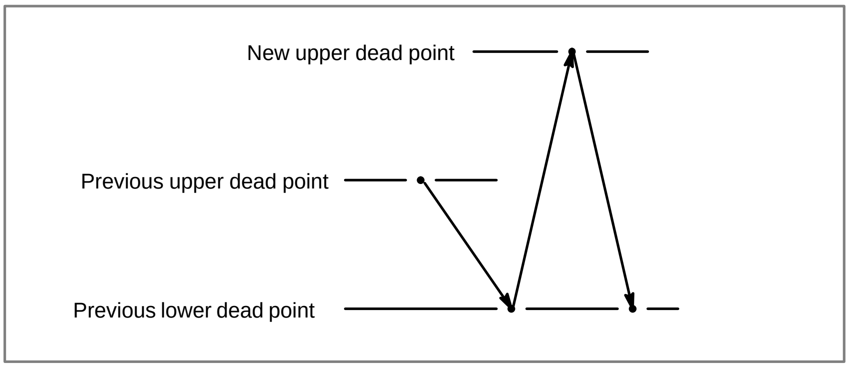
When the upper dead point or lower dead point is changed while chopping is being performed, the tool moves to the position specified by the old data. Then, chopping is continued using the new data.

While chopping is being performed, data can be changed only on the chopping screen. Changing the data on the parameter screen has no effect on the current chopping operation.

When movement according to the new data starts, the servo delay compensation function stops the servo delay compensation for the old data, and starts the servo delay compensation for the new data.

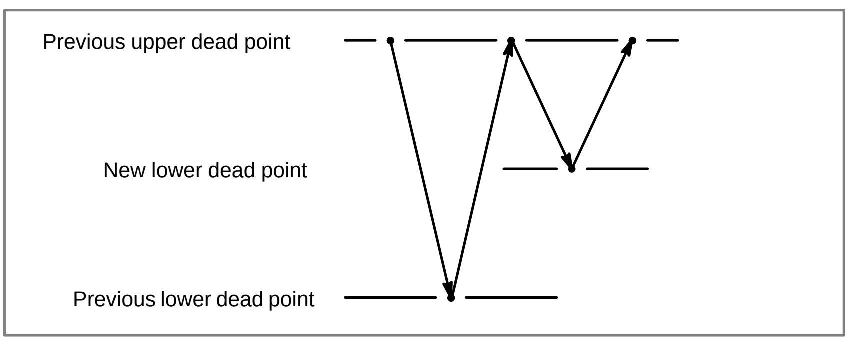
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



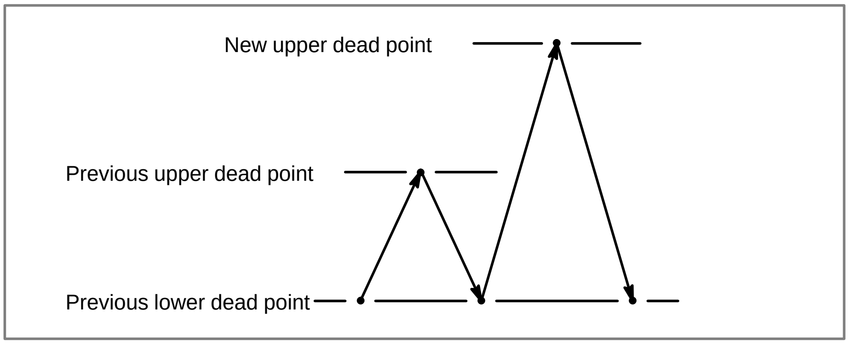
The tool first moves to the lower dead point, then to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



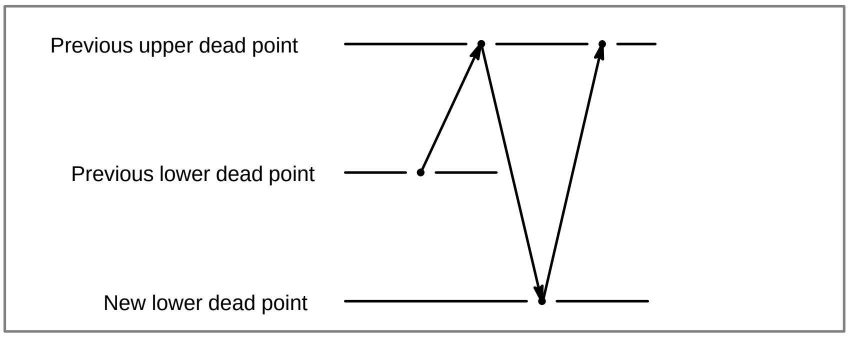
The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

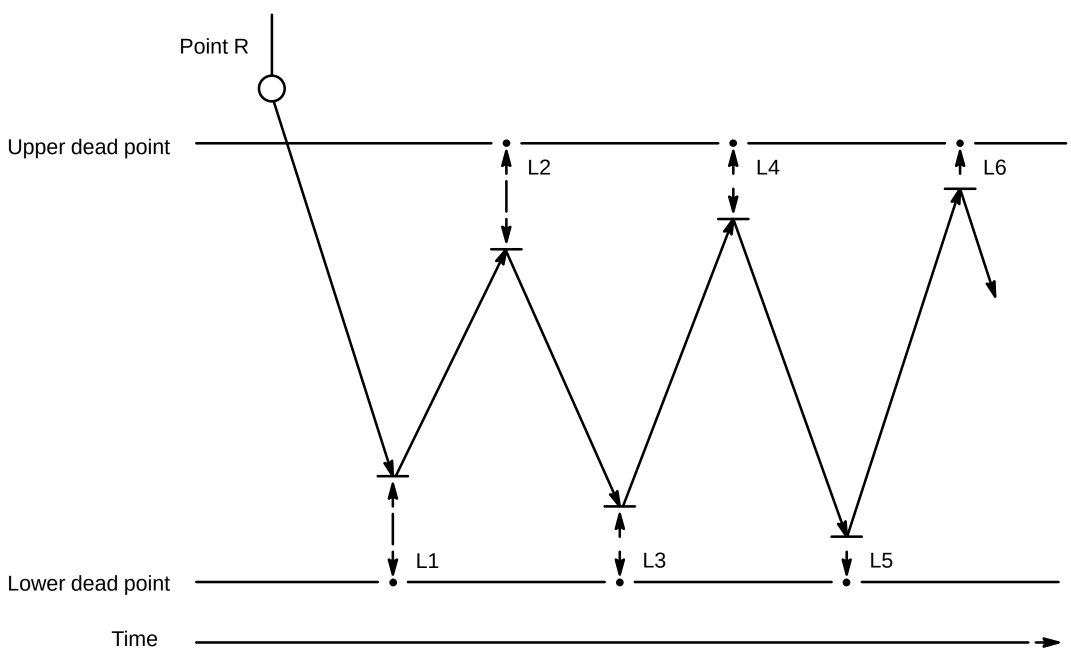
- **Servo delay compensation function**

When high–speed chopping is performed with the grinding axis, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The control unit measures the difference between the specified position and the actual tool position, and automatically compensates for the displacement of the tool. To compensate for this displacement, an amount of travel equal to the distance between the upper and lower dead points, plus an appropriate compensation amount, is specified. When a chopping command is specified, the feedrate is determined so that the chopping count per unit time equals the specified count. When the difference between the displacement of the tool from the upper dead point and the displacement of the tool from the lower dead point becomes smaller than the setting of parameter No. 8377, after the start of chopping, the control unit performs compensation.

When compensation is applied, the chopping axis moves beyond the specified upper dead point and lower dead point, and the chopping feedrate increases gradually.

When the difference between the actual machine position and the specified position becomes smaller than the effective area setting (parameter No. 1826), the control unit no longer applies compensation, allowing the tool to continue moving at its current feedrate.

A coefficient for the compensation amount for the displacement generated by the servo delay incurred by chopping and the delay incurred during acceleration/deceleration can be specified in parameter No. 8376.



Displacement between the tool and the upper dead point: L2, L4, L6
Displacement between the tool and the lower dead point: L1, L3, L5

Compensation starts when:

| L3 – L2 | < (parameter No. 8377)

When the following condition is satisfied, compensation is no longer applied, and the tool continues to move at its current feedrate:

| L6 | < effective area setting (parameter No. 1826)

- **Acceleration**

Exponential acceleration/deceleration is used for chopping axis.

- **Mode switching during chopping**

If the mode is changed during chopping, chopping does not stop. In manual mode, the chopping axis cannot be moved manually. It can, however, be moved manually by means of the handle interrupt.

- **Reset during chopping**

When a reset is performed during chopping, the tool immediately moves to point R, after which chopping mode is canceled.

If an emergency stop or servo alarm occurs during chopping, mode is canceled, and the tool stops immediately.

- **Stopping chopping**

The following table lists the operations and commands that can be used to stop chopping, the positions at which chopping stops, and the operation performed after chopping stops:

Operation/command	Stop position	Operation after chopping stops
G80	Point R	Canceled
CHPST: "0"	The tool moves to the lower dead point, then to point R.	Canceled
*CHLD: "0"	Point R	Restart after *CHLD goes "1"
Reset	Point R	Canceled
Emergency stop	The tool stops immediately.	Canceled
Servo alarm	The tool stops immediately.	Canceled
P/S alarm	The tool moves to the lower dead point, then to point R.	Canceled
OT alarm	The tool moves from the upper or lower point to point R.	Canceled

- **Background editing**

When an alarm of background editing or battery alarm is issued, the tool does not stop at point R.

- **Single block signal**

Even when single block signal SBK is input during chopping, chopping continues.

- **Part program storage length**

When this function is used, a part program storage length of approximately 0.6 m is required.

Limitations

- **Workpiece coordinate system**

While chopping is being performed, do not change the workpiece coordinate system for the chopping axis.

- **PMC axis**

When the chopping axis is selected as the PMC axis, chopping is not started.

- **Mirror image**

While chopping is being performed, never attempt to apply the mirror image function about the chopping axis.

- **Move command during chopping**

If a move command is specified for the chopping axis while chopping is being performed, a P/S 5050 alarm is issued.

- **Look–ahead control**

This function does not support the look–ahead control function.

- **Program restart**

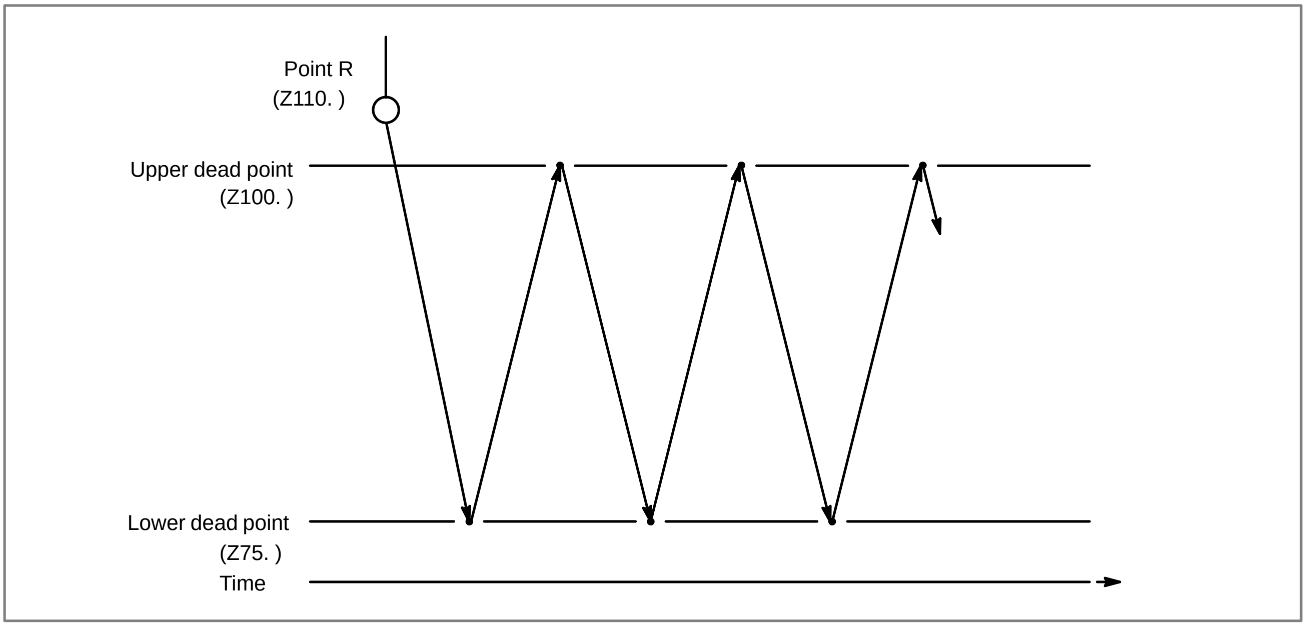
When a program contains G codes for starting chopping (G81.1) and stopping chopping (G80), an attempt to restart that program results in a P/S 5050 alarm being output.

When a program that does not include the chopping axis is restarted during chopping, the coordinates and amount of travel set for the chopping axis are not affected after the restart of the program.

Examples

G90 G81.1 Z100. Q–25. R10. F3000 ;

- Perform rapid traverse to position the tool to Z110. (point R).
- Then, perform reciprocating movement along the Z–axis between Z100. (upper dead point) and Z75. (lower dead point) at 3000 mm/min. Chopping override is enabled.



To cancel chopping, specify the following command:

G80 ;

- The tool stops at point R.

Signal

Chopping hold signal *CHLD <G051#7>

[Classification] Input signal

[Function] Suspends chopping.

[Operation] Once this signal has been set to 0, the tool is moved from the current position to point R, thus suspending chopping. Again setting this signal to 1while chopping is suspended causes chopping to be restarted.

Chopping start signal CHPST <G051#6>

[Classification] Input signal

[Function] Starts and stops chopping.

[Operation] Setting this signal to 1 starts chopping.
Again setting this signal to 0 during chopping causes chopping to be stopped.

> **NOTE**
> 1 If an attempt to start chopping using chopping start signal CHPST is ignored, set the signal to 0 then back to 1.
> 2 This signal is not enabled until the completion of manual reference position return.

Chopping feedrate override signals *CHP0 to *CHP8 <G051#0 to G051#3>

[Classification] Input signal

[Function] Overrides the chopping feedrate.

[Operation] The actual feedrate during chopping becomes the specified feedrate multipled by the override value specified with this signal. The following table lists the correspondence between the signal states and the override value:

*CHP8	*CHP4	*CHP2	*CHP0	Override value
0	0	0	0	150%
0	0	0	1	140%
0	0	1	0	130%
0	0	1	1	120%
0	1	0	0	110%
0	1	0	1	100%
0	1	1	0	90%
0	1	1	1	80%
1	0	0	0	70%
1	0	0	1	60%
1	0	1	0	50%
1	0	1	1	40%
1	1	0	0	30%
1	1	0	1	20%
1	1	1	0	10%
1	1	1	1	0%

Chopping–in–progress signal CHPMD <F039#2>

[Classification] Output signal

[Function] Posts notification of chopping in progress.

[Operation] This signal is set to 1 in the following case:

· Upon chopping start signal CHPST being set to 1 to start chopping

This signal is set to 0 in the following cases:

· Upon chopping start signal CHPST being set to 0 to stop chopping

· Upon chopping being terminated by a reset.

Chopping cycle signal CHPCYL <F039#3>

[Classification] Output signal

[Function] Posts notification of a chopping cycle being performed between the upper and lower dead pionts.

[Operation] This signal is set to 1 in the following case:

· Upon a chopping cycle being started between the upper and lower dead pionts

This signal is set to 0 in the following cases:

· Once chopping has been stopped

· When the tool is stopped at the upper or lower dead point

· Upon chopping hold signal *CHLD being set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G051	*CHLD	CHPST			*CHP8	*CHP4	*CHP2	*CHP0
G039					CHPCYL	CHPMD		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8360	CHPX							CPRPD

[Data type] Bit

CPRPD For the chopping function, a rapid traverse override for a section from the current position to the R point is determined as follows:

0 : A chopping override is enabled.
1 : An ordinary rapid traverse override is enabled.

CHPX On the chopping screen, the chopping speed can:

0 : Be set.
1 : Not be set.

8370	Chopping axis

[Data type] Byte

[Valid data range] 1 to the number of controlled axes

This parameter specifies which servo axis the chopping axis corresponds to.

8371	Chopping reference point (R point)
8372	Chopping upper dead point
8373	Chopping lower dead point

[Data type] 2–word

[Valid data range]

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

[Valid data range] –99999999 to 99999999

The data set in these parameters are absolute coordinates.

8374	Chopping speed

[Data type] 2–word

[Unit of data]

Increment system	Unit
Linear axis (metric input)	1.00 mm/min
Linear axis (inch input)	0.01 inch/min

Valid data range : For IS–A and –B, 240000 mm/min or 9600 inches/min
For IS–C, 100000 mm/min or 4800 inches/min

8375	Maximum chopping feedrate

[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Metric machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000
Rotation axis	1 deg/min	30 to 240000	30 to 100000

The chopping speed is clamped at a value specified in this parameter. When the parameter is 0, no chopping operation occurs.

8376	Chopping compensation scaling factor

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 100

This parameter specifies a scaling factor used to multiply the compensation value for a servo delay or acceleration/deceleration delay in an chopping operation. When this parameter is 0, servo delay compensation will not be applied.

8377	Compensation start tolerance

[Data type] Word

[Unit of data]

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

[Valid data range] 0 to 32767

Compensation is applied when the difference between an amount of shortage at the upper dead point and that at the lower dead point is less than the value specified in this parameter. In other words, this parameter is used to enable compensation after the chopping operation settles. When the parameter is 0, compensation will not be applied.

Alarm and message

Number	Message	Contents
5050	ILL–COMMAND IN CHOPPING MODE	When the chopping function is used, a move command was specified for a chopping axis in chopping mode (during reciprocation between a upper dead point and a lower dead point).

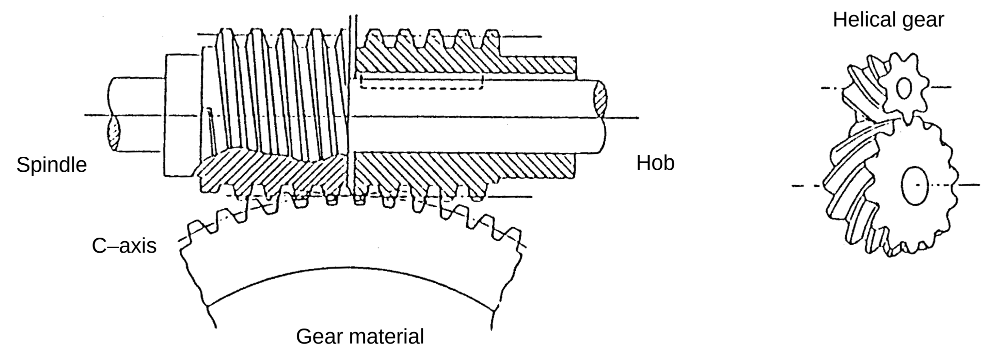
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.6	Chopping function (G80, G81.1)
		III.11.4.13	Displaying and setting chopping data

1.13 HOBBING MACHINE FUNCTION (M SERIES)

General

Gears can be cut by turning the workpiece (C–axis) in sync with the rotation of the spindle (hob axis) connected to a hob.
Also, a helical gear can be cut by turning the workpiece (C–axis) in sync with the motion of the Z–axis (axial feed axis).



Format

G81 T _ L _ Q _ P _ ;

T : Number of teeth (specifiable range: 1 to 5000)

L : Number of hob threads (specifiable range: 1 to 20 with a sign)
- The sign of L specifies the direction of rotation of the C–axis.
- If L is positive, the C–axis rotates in the positive direction (+).
- If L is negative, the C–axis rotates in the negative direction (–)

Q : Module or diametral pitch
For metric input, specify a module.
(Units = 0.00001 mm; specifiable range = 0.01 to 25.0 mm)
For inch input, specify a diametral pitch.
(Units = 0.00001 inch^{-1}; specifiable range = 0.01 to 250.0 inch^{-1})

P : Gear helix angle
(Units = 0.0001 deg; specifiable range = –90.0 to +90.0 deg)

P and Q must be specified when a helical gear is to be cut.

G81 ; Cancels synchronization between the hob axis and C–axis.

Explanations

- **Setting the C–axis**

The C–axis (workpiece) is usually the fourth axis. However, any axis can be set as the C–axis by setting the corresponding parameter appropriately (parameter No. 7710).

- **Maintaining the synchronization status**

The synchronization status is maintained provided:

- The interlock signal for the C–axis is turned on.
- The feed hold state exists.

- **Releasing the synchronization status**

Synchronization between the hob axis and C–axis can also be canceled when:

- The power is turned off.
- An emergency stop or servo alarm occurs.
- A reset (external reset signal ERS, reset & rewind signal RRW, or reset key on the MDI panel) is issued.

By setting bit 0 (HBR) of parameter No. 7700, the release of the synchronization status by a reset can be suppressed.

- **Helical gear compensation**

When a helical gear is to be cut, compensation for the C–axis, according to the amount of travel along the Z–axis (third axis) (axial feed) and gear helix angle, is required.

Helical gear compensation is performed by adding compensation pulses, calculated using the following formula, to the C–axis which is synchronized with the hob axis:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (For metric input)}$$

or

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (For inch input)}$$

where

Compensation angle: Signed absolute value (deg)

Z : Amount of travel along the Z–axis after the specification of G81 (mm or inches)
Total amount of travel along the Z–axis in both automatic and manual modes

P : Signed gear helix angle (deg)

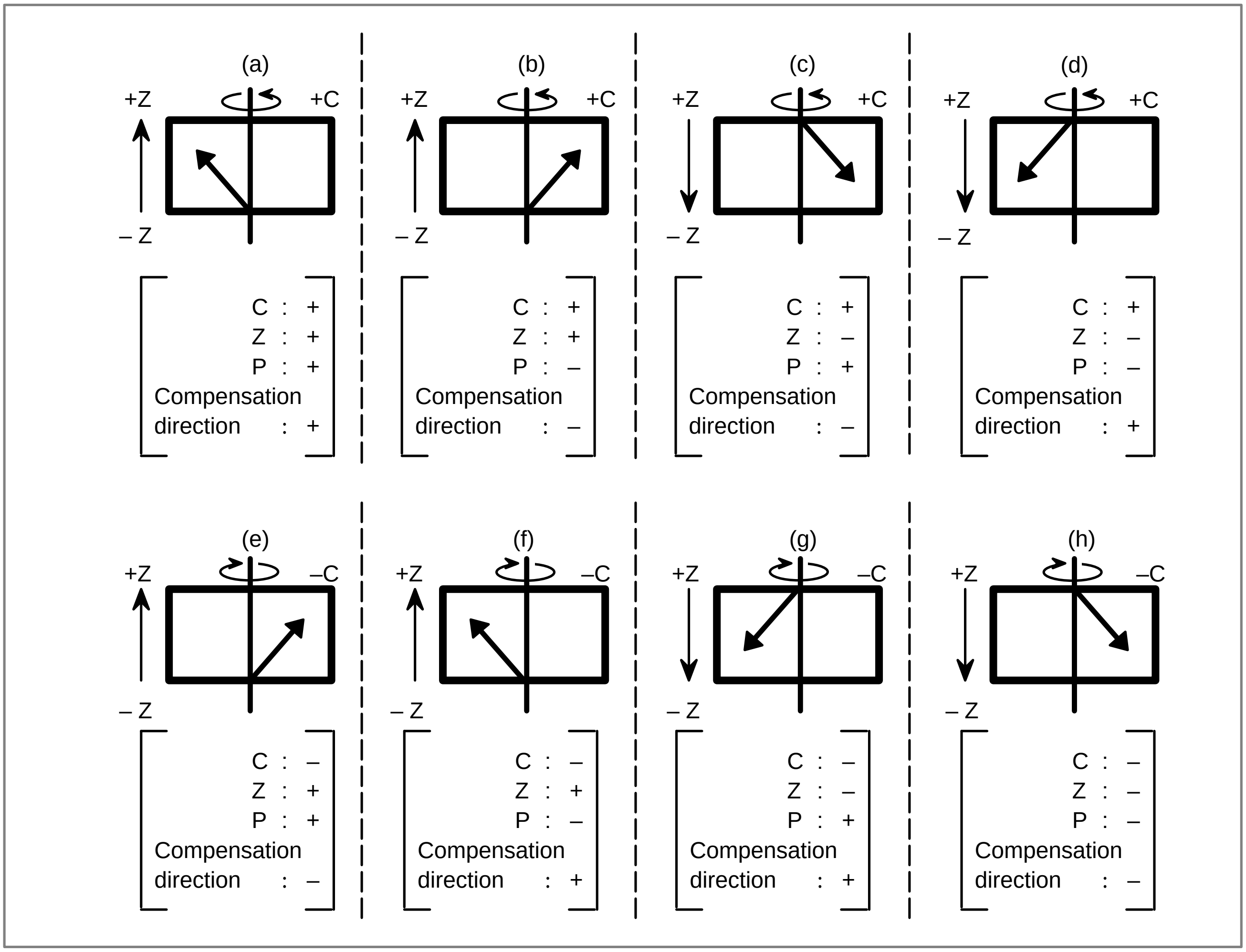
T : Number of teeth

Q : Module (mm) or diametral pitch (inch^{-1})

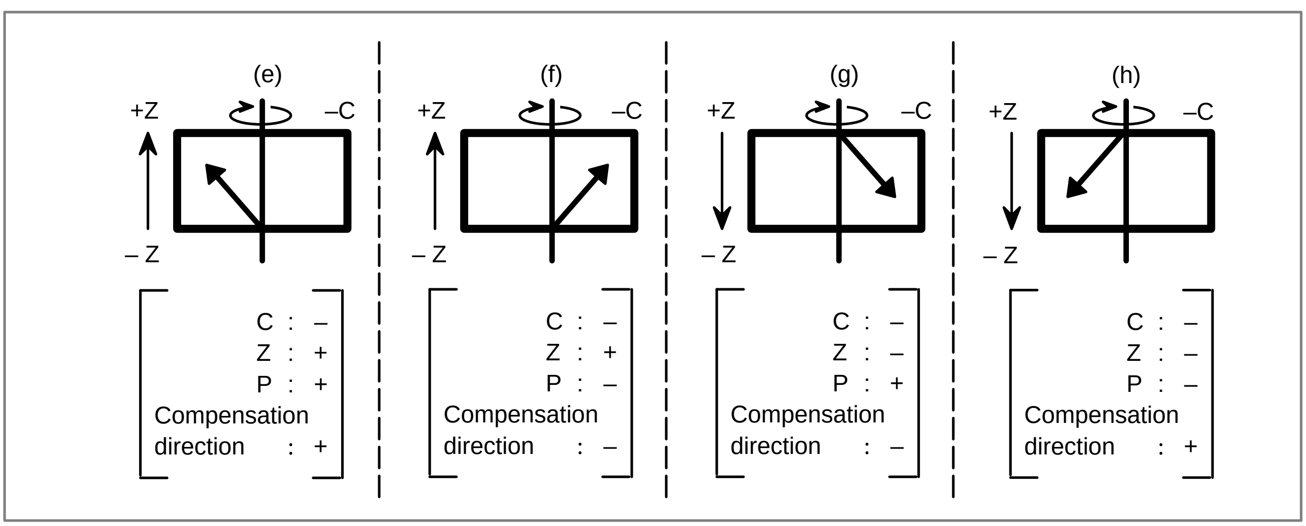
The values of P, T, and Q must be programmed.

- **Direction of the helical gear compensation**

1 When bit 2 (HDR) of parameter No. 7700 = 1



2 When bit 2 (HDR) of parameter No. 7700 = 0 (Items (a) to (d) are the same as for 1.)



- **Setting the helical gear axial feed axis**

The Z–axis (axial feed axis) is usually the third axis. However, any axis can be set as the Z–axis by setting the corresponding parameter appropriately (parameter No. 7709).

- **C–axis servo delay compensation (G82, G83, G84)**

The servo delay is proportional to the speed of the hob axis. Therefore, in a cycle where rough machining and finish machining are performed at different hob axis speeds, compensation for the servo delay is required.

The servo delay is calculated as follows:

$$E = \{ (\frac{Fc}{60}) \times (\frac{1}{Ks} + C \ast M + L) Sup\} \times N$$

where

E : C–axis servo delay compensation (deg)
Fc : C–axis speed when G83 is specified (deg/min)
Ks : Servo loop gain (LPGIN of parameter No. 1825) (s^{-1})
C : Delay incurred in the CNC (s)
M : Delay compensation magnification 1 in the CNC (SVCMP1 of parameter No. 7715)
L : Delay incurred by smoothing, as specified by parameter No. 7701 (s)
Sup : Remaining pulse error caused by acceleration/deceleration (deg)
N : C–axis servo delay compensation magnification 2 (SVCMP2 of parameter No. 7714)

When the hob axis speed is changed, C–axis servo delay compensation is performed using either of the following two methods:

- Compensation is specified both before and after the speed is changed. Each time G83 is specified, compensation for the delay at that time is applied.
- Before the speed is changed, the servo delay is recorded. After the speed is changed, compensation for the difference between the recorded delay and that observed when the command is specified is performed.

The latter method, in which the compensation before speed change is recorded, can be used by setting bit 5 (DLY) of parameter No. 7701 to 1. This method, in comparison with that where the amount of compensation is not recorded, offers the advantage of increasing the processing speed.

- Method in which compensation for the delay when a command is specified is performed (G82, G83)

G82 : Cancels C–axis servo delay compensation.

G83 : Executes C–axis servo delay compensation.

(Example)

```
G81  T___ L___ ;     . . Starts synchronization.
M03  S100  ;         . . Rotates the hob axis.
G04  P2000  ;        . . Causes the tool to dwell to assure
                         constant hob axis rotation.
G01  G83  F___  ;    . . Performs C–axis delay compensation.
G01  X___ F___  ;
        .
        .
G82  ;               . . Cancels C–axis servo delay.
S200  ;              . . Changes the speed.
G04  P2000  ;        . . Causes the tool to dwell to assure
                         constant hob axis rotation.
G01  G83  F___  ;    . . Performs C–axis delay compensation.
```

- Method in which the delay before change is recorded (G82, G83, G84)

  G82 : Cancels C–axis servo delay compensation.

  G83 : Applies compensation for the difference between the C–axis servo delay, observed when G83 is specified, and the delay recorded by G84.

  G84 : Records the C–axis servo delay observed when G84 is specified. (The recorded value remains as is until G81 is specified or another G84 is specified.)

  (Example)

G81 T___ L___ ;	. . Starts synchronization.
M03 S100 ;	. . Rotates the hob axis.
G04 P2000 ;	. . Causes the tool to dwell to assure constant hob axis rotation.
G84 ;	. . Records the C–axis servo delay at the current speed.
G01 X___ F___ ;	
.	
.	
.	
S200 ;	. . Changes the speed.
G04 P2000 ;	. . Records the C–axis servo delay at the current speed.
G01 G83 F___ ;	. . Applies C–axis delay compensation.

- Notes
  - Specify the G83 block in G01 mode. Also, specify a feedrate using the F code.
  - Once G83 has been specified, another G83 command cannot be specified until compensation is canceled by specifying G82, or C–axis synchronization is canceled.
  - Specify G83 once a constant hob axis rotation speed has been achieved.
  - In C–axis servo delay compensation (G83), compensation is not applied to the integer part of the gear pitch. The compensation direction is opposite to that of the C–axis rotation.

• C–axis synchronous shift

- C–axis handle interrupt

  During synchronization between the hob axis and C–axis, manual handle interrupt can be performed for the C–axis. The C–axis is shifted by the amount of the handle interrupt.
  For details of handle interrupts, refer to the manual supplied by the machine tool builder.

- Synchronous shift by programming

  During synchronization between the hob axis and C–axis, the C–axis can be interrupted using G01. In this case, be careful not to exceed the maximum cutting speed.
  Example: Hob shifting during synchronization
  G01 Y___ C___ F___ ;

- **Manual setting of one–rotation signal**

When the rotation of the position coder is stopped, setting the one–rotation signal set signal MSPC to 1 causes the position of the one–rotation signal to shift in the CNC as if the one–rotation signal had been output with the position coder at the current position.
When the shift request is accepted, the one–rotation position setting completed signal MSPCF is output. According to this signal, the operator knows when the one–rotation signal position is shifted. The shift request is accepted if the C–axis is in sync–off state (G80). The shift request is canceled when the synchronization cancel command (such as G80 or a reset) is issued.

- **Retract function**

In both automatic and manual operation modes, setting the retract signal RTRCT to 1 (the rising edge is detected) enables retraction over the distance specified by parameter No. 7741, along the axis set by bit 0 (RTRx) of parameter No. 7730.
Upon the completion of retraction, the retract completed signal RTRCTF is output.

- The feedrate used for retracting is that specified using parameter No. 7740. Feedrate override is not supported for retracting.
- Feed hold is not supported for retraction.
- If the retract signal becomes 1 during automatic operation, retraction occurs, and automatic operation is discontinued.
- The retract completed signal becomes 0 when any retract axis is shifted.

Limitations

- **Setting a rotation axis**

Set a rotation axis as the C–axis (workpiece axis). (Bit 0 (ROTx) of parameter No. 1006 = 1)

- **Gear ratio of the spindle and position coder**

The gear ratio of the spindle to the position coder must be 1. (Bits 0 and 1 of parameter No. 3706 = 0)

Signal

Retract signal RTRCT<G066#4>

[Classification] Input signal

[Function] Causes retraction along an axis specified using bit 0 (RTR) of parameter No. 7730.

[Operation] When this signal becomes 1, the control unit behaves as follows:

- The control unit detects the positive–going edge of this signal, and can cause retraction along an axis specified using bit 0 (RTR) of parameter No. 7730 on that positive–going edge. The amount and speed of retraction are those specified in advance using parameter Nos. 7741 and 7740. Upon the completion of retraction, the retract completed signal RTRCTF becomes 1. The retract signal is usable for both automatic operation (such as MEM and MDI) and manual operation (such as HNDL and JOG) modes. If the retract signal is set to 1 during automatic operation, retraction is started, and automatic operation is discontinued.

Retract completed signal RTRCTF<F065#4>

[Classification] Output signal

[Function] Indicates when retraction is complete.

[Operation] The signal becomes 1 when:

- Retraction ends.

The signal becomes 0 when:

- Any movement along the axis occurs once retraction along that axis has been completed.

One–rotation position manual set signal MSPC<G066#5>

[Classification] Input signal

[Function] Shifts the one–rotation signal position of the position coder.

[Operation] When this signal becomes 1, the control unit behaves as follows:

- The position of the position coder when the signal becomes 1 is recorded as the one–rotation signal position in the CNC. In addition, the one–rotation signal setting completed signal MSPCF becomes 1 to indicate that the one–rotation signal position has been recorded. Once the one–rotation position manual set signal is issued, synchronization between the hob axis and C–axis is established according to the specified one–rotation signal position specified by G81. This signal is ignored when:
  - The hob axis and C–axis are already in sync (G81 mode).
  - The one–rotation signal setting completed signal MSPCF is already 1.

When bit 1 (CMS) of parameter No. 7700 is 0, issuing a synchronization cancel command (such as G80 or a reset) cancels a specified one–rotation signal position. Once the one–rotation signal position is canceled, synchronization between the hob axis and C–axis is established according to a one–rotation signal from the position coder as specified by the G81.

One–rotation position setting completed signal MSPCF<F065#5>

[Classification] Output signal

[Function] Indicates that the one–rotation signal position has been set up according to one–rotation position manual set signal MSPC.

[Operation] The signal becomes 1 when:

- The one–rotation signal position is set up according to one–rotation position manual set signal MSPC.

The signal becomes 0 when:

- The one–rotation signal position is canceled, that is, a synchronization cancel command (such as G80 or a reset) is issued when bit 1 (CMS) of parameter No. 7700 is 0.

Sync–with–C–axis signal HOBSYN<F065#7>

[Classification] Output signal

[Function] Indicates that the hob axis is in sync with the C–axis.

[Operation] The signal is 1 when:

- The hob axis is in sync with the C–axis (during G81 mode).

The signal is 0 when:

- The hob axis is not in sync with the C–axis (during G80 mode).

Cancel–sync–with–C–axis signal HOBCAN <G066#2>

[Classification] Input signal

[Function] Cancels synchronization between the hob axis and C–axis.

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

- The control unit cancels synchronization between the hob axis and C–axis (G81 mode). The specified one–rotation position (MSPCF) varies with bit 1 (CMS) of parameter No. 7700.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066			MSPC	RTRCT		HOBCAN		

	#7	#6	#5	#4	#3	#2	#1	#0
F065	HOBSYN		MSPCF	RTRCTF				

Parameter

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

[Data type] Bit axis

ROTx Specifies whether each axis is a linear or rotation axis.

0 : Linear axis
1 : Rotation axis

The C–axis (workpiece axis) must always be a rotation axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PSG2	PSG1

[Data type] Bit

PSG2, PSG1 PSG2 and PSG1 Gear ratio of the spindle to the position coder. Both bits must be 0 (magnification = 1).

	#7	#6	#5	#4	#3	#2	#1	#0
7700		DPS	RTO		MLT	HDR	CMS	HBR

[Data type] Bit

HBR Specifies whether to cancel synchronization between the C–axis and hob axis (G81) when a reset occurs.

0 : Canceled.
1 : Not canceled.

CMS Specifies whether to cancel the manually set one–rotation signal position when a synchronization cancel command (such as G80 or a reset) is issued.

0 : Canceled.
1 : Not canceled.

HDR HDR Specifies the direction of helical gear compensation
(This bit is usually set to 1.)

MLT Specifies the data unit for the C–axis servo delay compensation amount magnification (parameter No. 7714).

0 : 0.001
1 : 0.0001

RTO Specifies whether to enable the gear ratio of the spindle to the position coder specified in parameter No. 3706.

0 : Disabled. (This bit must always be 0.)
1 : Enabled.

DPS Specifies what is to be displayed as the actual spindle speed.

0 : Rotational speed of the hob axis
1 : Rotational speed of the spindle

	#7	#6	#5	#4	#3	#2	#1	#0
7701	HBD		DLY	JHD		SM3	SM2	SM1

[Data type] Bit

SM1, SM2, SM3 Specify how many times the hobbing machine is to sample feedback pulses from the position coder.

SM3	SM2	SM1	Number of times sampling is to be performed
0	0	0	4
0	0	1	1
0	1	0	2
0	1	1	16
1	0	0	32
1	1	0	4
1	1	1	4

JHD Specifies whether to enable C–axis jog and handle feed during synchronization between the C–axis and hob axis (G81 mode).

0 : Disabled.
1 : Enabled.

DLY Specifies whether to enable C–axis servo delay compensation based on G84.

0 : Disabled.
1 : Enabled.

HBD Specifies whether to enable the specification of diametral pitch for inch input.

0 : Disabled.
1 : Enabled.

7709	Axis number of a helical gear axial feed axis

[Data type] Byte

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

This parameter specifies the placing of the helical gear axial feed axis. If a value that falls outside the valid data range is specified, the helical gear axial feed axis is specified as the third axis.

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

7710	Axis number of a synchronous axis

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

[Data type] Byte

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

This parameter specifies the placing of an axis (workpiece) in sync with the hob axis (tool). If a value that falls outside the valid data range is specified, the synchronous axis is specified as the fourth axis.

7711	Gear ratio of the hob axis to the position coder

[Data type] Byte

[Valid data range] 1 to 20

[Unit of data] 1

This parameter specifies the gear ratio of the hob axis to the position coder.

7712	Acceleration/deceleration time constant applied to the C–axis when it is in sync with the hob axis

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

This parameter specifies an acceleration/deceleration (exponential acceleration/deceleration) time constant applied to the C–axis when it is in sync with the hob axis.

NOTE

In G01, G83, and helical gear compensation, acceleration/deceleration is performed according to the acceleration/deceleration time constant for cutting feed and FL feedrate (parameter Nos. 1622 and 1623).

7713	Acceleration/deceleration FL feedrate applied to the C–axis when it is in sync with the hob axis

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	Valid data range	
	IS–B	IS–C
1 deg/min	6 ～ 15000	6 ～ 12000

This parameter specifies the FL feedrate for acceleration/deceleration (exponential acceleration/deceleration) applied to the C–axis when it is in sync with the hob axis.

7714	Magnification 2 for a G83–based C–axis servo delay compensation

[Data type] Word

[Unit of data] 0.0001/0.001

[Valid data range] 500 to 2000

This parameter specifies magnification 2 for a G83–based C–axis servo delay compensation.

7715	Magnification 1 for a G83–based C–axis servo delay compensation

[Data type] Word

[Unit of data] 0.0001/0.001

[Valid data range] 500 to 2000

This parameter specifies magnification 1 for a G83–based C–axis servo delay compensation.

7730	#7	#6	#5	#4	#3	#2	#1	#0
								RTRx

[Data type] Bit axis

RTRx Specifies whether to apply the retract function for each axis.

0 : Not applied.
1 : Applied.

7740	Retract speed for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Metric machine	1 mm/min	30 ~ 240000	30 ~ 100000
Inch machine	0.1 inch/min	30 ~ 96000	30 ~ 48000

This parameter specifies a retract speed for each axis.

7741	Retract amount for each axis

[Data type] Two–word axis

[Valid data range] ±99999999

[Unit of data]

Increment system	Unit of data	
	IS–B	IS–C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

This parameter specifies a retract amount for each axis.

Alarm and message

Number	Message	Description
181	FORMAT ERROR IN G81 BLOCK	The G81 block is not of the correct format. 1 T (number of teeth) has not been specified. 2 Data specified for T, L, Q, or P falls outside the valid data range.
182	G81 NOT COMMANDED	G83 (C–axis servo delay compensation) was specified when G81–based synchronization had not been specified.
183	DUPLICATE G83 (COMMANDS)	G83–based C–axis servo delay compensation was requested again before the previous request had been canceled using G82.
184	ILLEGAL COMMAND IN G81	An invalid command was issued during G81–based synchronization. 1. C–axis commands based on G00, G27, G28, G29, and G30 2. G20– or G21–based commands for switching between inch and metric inputs
185	RETURN TO REFERENCE POINT	G81 was issued before no reference position return was made after the power was switched on or an emergency stop occurred.
186	PARAMETER SETTING ERROR	A value for a parameter related to G81 is invalid. 1. The C–axis has not been specified as a rotation axis (bit 0 (ROT) of parameter No. 1006). 2. A value specified as the gear ratio of the hob axis to the position coder is invalid (parameter No. 7711). Note) These errors may be detected when G81 is issued.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.20.7	Hobbing machine

1.14 SIMPLE ELECTRIC GEAR BOX (G80, G81) (M SERIES)

General

To machine (grind/cut) a gear, the rotation of the workpiece axis connected to a servo motor is synchronized with the rotation of the tool axis (grinding wheel/hob) connected to the spindle motor. To synchronize the tool axis with the workpiece axis, an electric gear box (EGB) function is used for direct control using a digital servo system. With the EGB function, the workpiece axis can trace tool axis speed variations without causing an error, thus machining gears with great precision.

- **Example control axis configuration**

The 4th axis is always assigned as the workpiece axis.

Another servo axis is dedicated to the tool axis, which is connected to the spindle motor and for which the rotational position must be read directly by the digital servo system (this axis is called the EGB axis). It is necessary to assign these axes as odd and even servo axes in succession (parameter No. 1023). See Section 1.4.3 for an example showing how to specify the parameters for configuring the simplified electronic gearbox.

Format

G81 T _ L _ Q _ P _ ;	**Starts synchronization.**
S_ M03 (or M04) ;	**Starts tool axis rotation.**
M05 ;	**Stops tool axis rotation.**
G80 ;	**Cancels synchronization.**

T : Number of teeth (Specifiable range: 1 to 1000)

L : Number of hob threads
(Specifiable range: –21 to +21 with 0 excluded)

Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001 mm, Specifiable range: 0.01 to 25.0 mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001 $inch^{-1}$, Specifiable range: 0.01 to 25.0 $inch^{-1}$)

P : Gear helix angle
(Unit: 0.0001 deg, Specifiable range: –90.0 to 90.0 deg.)

* When specifying Q and P, the user can use a decimal point.

Explanations

• Synchronization control

1 Start of synchronization

When synchronization mode is set with G81, the synchronization switch of the EGB function is closed, and synchronization between the tool axis and workpiece axis starts. At this time, synchronization mode signal SYNMOD is turned on. During synchronization, the rotation of the tool axis and workpiece axis is controlled so that the relationship between T (number of teeth) and L (number of hob threads) can be maintained. Moreover, the synchronous relationship is maintained regardless of whether the operation is automatic or manual during synchronization.

G81 cannot be specified again during synchronization. Moreover, the specification of T, L, Q, and P cannot be modified during synchronization.

2 Start of tool axis rotation

When the rotation of the tool axis starts, the rotation of the workpiece starts so that the synchronous relationship specified in the G81 block can be maintained.

The rotation direction of the workpiece axis depends on the rotation direction of the tool axis. That is, when the rotation direction of the tool axis is positive, the rotation direction of the workpiece axis is also positive; when the rotation direction of the tool axis is negative, the rotation direction of the workpiece axis is also negative. However, by specifying a negative value for L, the rotation direction of the workpiece axis can be made opposite to the rotation direction of the tool axis.

During synchronization, the machine coordinates of the workpiece axis and EGB axis are updated as synchronous motion proceeds. On the other hand, a synchronous move command has no effect on the absolute and relative coordinates.

3 Termination of tool axis rotation

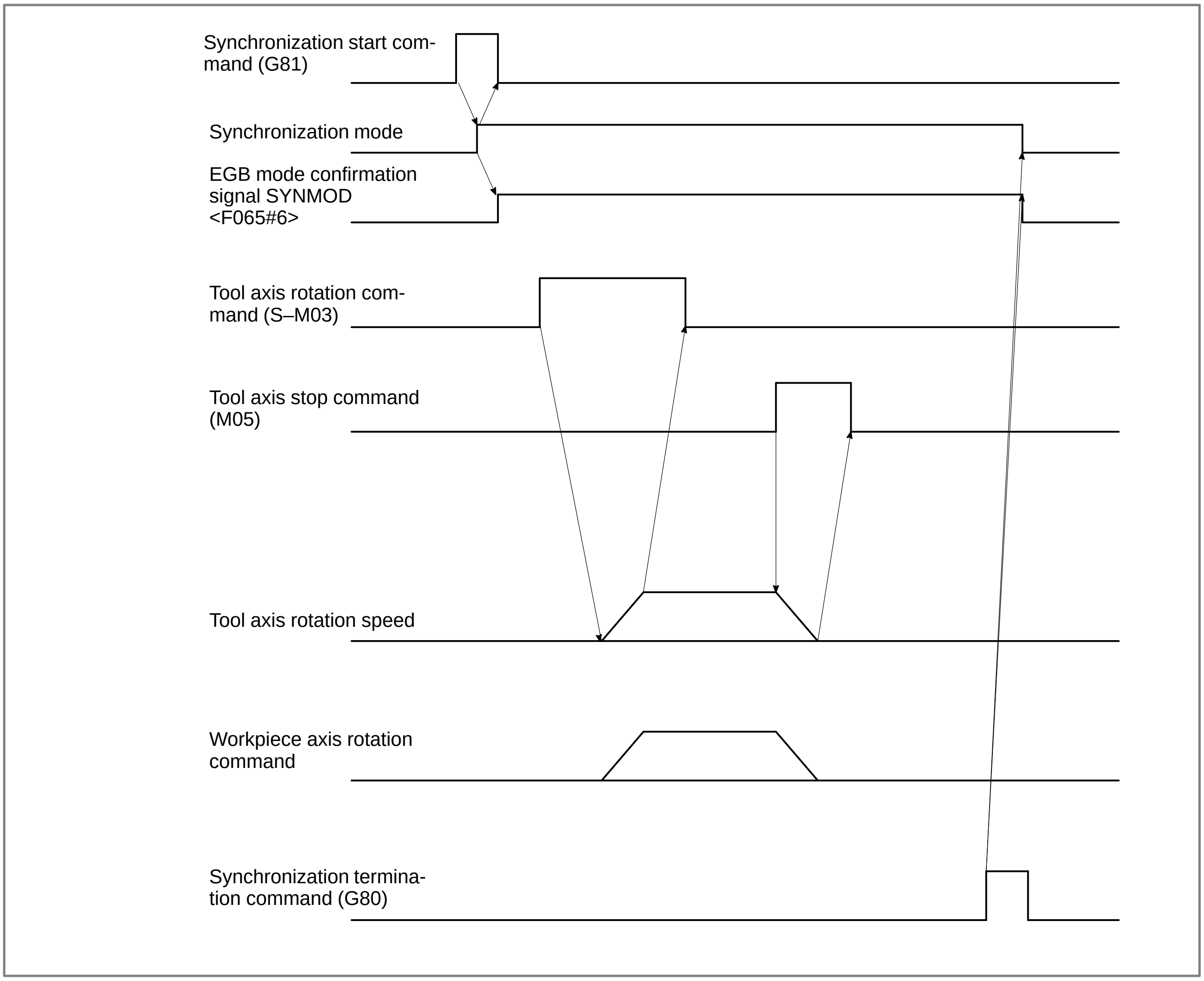
In synchronism with gradual stop of the tool axis, the workpiece axis is decelerated and stopped. By specifying the command below after the spindle stops, synchronization is canceled, and the EGB synchronization switch is opened. At this time, the synchronization mode signal (SYNMOD) is turned off.

4 Cancellation of synchronization

The position of the workpiece axis after travel during synchronization is reflected in the absolute coordinates when synchronization is canceled; from this point, absolute command programming is enabled for the workpiece axis. By setting bit 0 (HBR) of parameter No. 7700 to 0, synchronization can also be canceled upon reset.

* The synchronization mode is canceled by a servo alarm, PS000 alarm, or emergency stop.

- **Example timing for starting/terminating synchronization**



- **Helical gear compensation**

When a helical gear is to be produced, the compensation of workpiece axis rotation is needed according to the travel distance on the Z–axis (axial feed).

Helical gear compensation is performed by adding compensation pulses calculated from the formula below to the workpiece axis:

$$\text{Compensation angle} = \frac{Z \times \sin(P)}{\pi \times T \times Q} \times 360 \text{ (For metric input)}$$

or

$$\text{Compensation angle} = \frac{Z \times Q \times \sin(P)}{\pi \times T} \times 360 \text{ (For inch input)}$$

where

Compensation angle: Signed absolute value (deg)

Z : Amount of travel on the Z–axis after the specification of G81 (mm or inch)

P : Signed gear helix angle (deg)

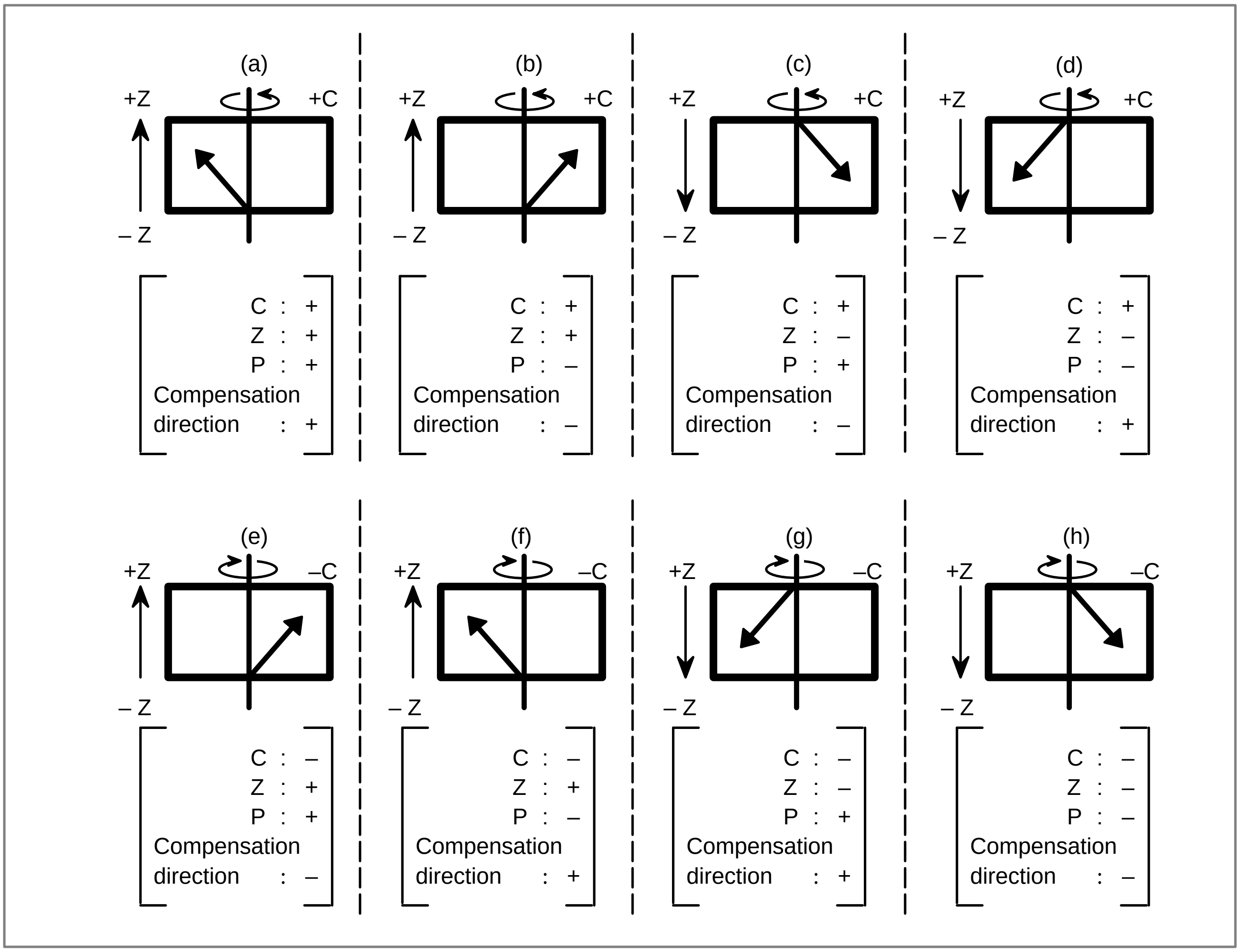
T : Number of teeth

Q: Module (mm) or diametral pitch ($inch^{-1}$)

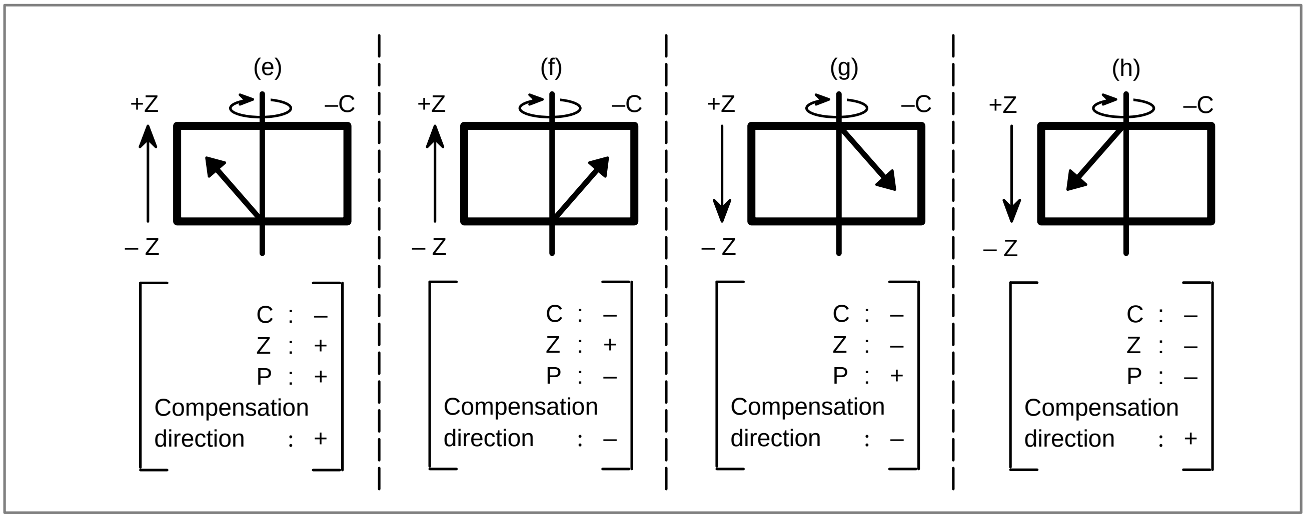
The values of P, T, and Q are to be programmed.

- **Direction of helical gear compensation**

1 When bit 2 (HDR) of parameter No. 7700 = 1



2 When bit 2 (HDR) of parameter No. 7700 = 0 (Items (a) to (d) are the same as for 1.)



- **Coordinates in helical compensation**

In helical compensation, the machine coordinates and absolute coordinates of the workpiece axis (4th axis) are updated by the amount of helical compensation.

- **Retraction**

By turning on the retract signal RTRCT (on a rising edge) in automatic operation mode or manual operation mode, a retract movement can be made over the distance specified in parameter No. 7741 on the axis set in bit 0 (RTRx) of parameter No. 7730. Upon completion of retract operation, the retract completion signal RTRCTF is output.

- **Feedrate at retraction**

For retract operation, the feedrate specified in parameter No. 7740 is used. During retract operation, the feedrate override capability is disabled.

- **Retraction during automatic operation**

When the retract signal is turned on in automatic operation, retract operation is performed, and automatic operation is stopped at the same time.

- **Synchronization coefficient**

A synchronization coefficient is internally represented using a fraction (K2/K1) to eliminate an error. The formula below is used for calculation. (α, β : Number of detector pulses per rotation of the tool axis, and number of detector pulses per rotation of the workpiece axis (parameter Nos. 7772 and 7773), respectively)

$$\text{Synchronization coefficient} = \frac{K2}{K1} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

In the formula above, K2/K1 is obtained by reducing the right side to lowest terms, but K1 and K2 must satisfy the following restriction:

$$-2147483648 \leqq K2 \leqq -2147483647$$
$$1 \leqq K1 \leqq 65535$$

When this restriction is not satisfied, the PS181 alarm is issued when G81 is specified.

- **Manual handle interrupt**

During synchronization, a manual handle interrupt can be used for the workpiece axis and other servo axes.

- **Move command during synchronization**

During synchronization, a move command can be programmed for the workpiece axis and other servo axes. Note, however, that incremental command programming for cutting feed must be used to specify a workpiece axis move command.

Limitations

- **Feed hold during retraction**
For retract movement, the feed hold capability is disabled.

- **Retraction when alarm is issued**
This function does not include a retract function used when an alarm is issued.

- **Rapid traverse during synchronization**
In synchronization mode, a cutting feedrate can be specified for the workpiece axis (4th axis). Rapid traverse cannot be specified using G00.

- **Maximum speed**
The maximum speeds of the tool axis and workpiece axis depend on the detectors used.

- **G code command during synchronization**
During synchronization, G00, G28, G27, G29, G30, G53, G20, and G21 cannot be specified.

- **Drilling canned cycle**
When this function is used, the drilling canned cycle cannot be used.

Examples

O1000 ;	
N0010 M19 ;	Performs tool axis orientation.
N0020 G28 G91 C0 ;	Performs reference position return operation of the workpiece axis.
N0030 G81 T20 L1 ;	Starts synchronization between the tool axis and workpiece axis. (The workpiece axis rotates 18° when the tool axis makes one rotation.)
N0040 S300 M03 ;	Rotates the tool axis at 300 rpm.
N0050 G01 X____ F____ ;	Makes a movement on the X–axis (for cutting).
N0060 G01 Z____ F____ ;	Makes a movement on the Z–axis (for machining).
– – – – – – – – – – – – – – – – ;	
– – – – – – – – – – – – – – – – ;	
N0100 G01 X____ F____ ;	Makes a movement on the X–axis (for retraction).
N0110 M05 ;	Stops the tool axis.
N0120 G80 ;	Cancels synchronization between the tool axis and workpiece axis.
N0130 M30 ;	

Signal

Retract signal RTRCT <G066#4>

[Classification] Input signal

[Function] Performs retraction for the axis specified with a parameter.

[Operation] When this signal is set to 1, the control unit performs the following:

· Performs retraction on the axis specified with bit 0 (RTRx) of parameter No. 7730. The retract speed and amount of retraction are specified with parameter Nos. 7740 and 7741.
The retract signal is effective both in automatic operation mode and manual operation mode. Setting the retract signal to 1 during automatic operation suspendsautomatic operation and causes retraction to be performed.

Retract completion signal RTRCTF <F065#4>

[Classification] Output signal

[Function] Posts notification of the completion of retraction.

[Operation] This signal is set to 1 in the following case:

· Upon the completion of retraction
This signal is set to 0 in the following case:

· Upon the completion of retraction, when a move command follows immediately after

> **NOTE**
> The retract signal is not accepted while the retract completion signal is set to 1.

EGB mode signal SYNMOD <F065#6>

[Classification] Output signal

[Function] Posts notification that synchronization using the EGB is in progress.

[Operation] This signal is set to 1 in the following case:

· While synchronization using the EGB is in progress
This signal is set to 0 in the following case:

· Once synchronization using the EGB has terminated

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066				RTRCT				

	#7	#6	#5	#4	#3	#2	#1	#0
F065		SYNMOD		RTRCTF				

Parameter

When setting the parameters for the simple electric gear box, note the following:

(1) Set SYNMOD (bit 0 of parameter No. 2011) to 1 for the workpiece (4th) axis and EGB axis.

(2) If FFALWY (bit 1 of parameter No. 2011) is set to 1, the values set in parameters No. 2068 and 2069 are used as the feed–forward factor and velocity loop feed–forward factor, respectively. For details, refer to ”Feed–forward setting” in ”Geometric error suppression function” in the FANUC AC SERVO MOTOR α Series Parameter Manual (B–65150E).

(3) Set the servo parameters for the EGB axis (No. 2000 and subsequent parameters) such that they do not conflict with the settings made for the 4th (workpiece) axis.

(4) Set the command multiplication (CMR) for the EGB axis (No. 1820) in the same way as for the 4th axis.

(5) The following EGB axis parameters need not be set:

- Reference counter capacity (No. 1821)
- In–position width (No. 1826)
- Excessive error while moving/stopped (No. 1828 and 1829)
- Stored stroke limits (No. 1320 to 1327)

(6) The flexible feed gear for the EGB axis (No. 2084 and 2085) is set automatically upon the execution of a synchronization mode command.

(7) Set the servo axis numbers in parameter No. 1023, using as many bits as the number of servo axes.

(Example)

When using the Y–axis as the EGB axis in a 4–axis configuration

No. 1023 1st axis:	1
No. 1023 2nd axis:	4
No. 1023 3rd axis:	2
No. 1023 4th axis:	3

When using the 5th axis as the EGB axis in a 6–axis configuration

No. 1023 1st axis:	1
No. 1023 2nd axis:	2
No. 1023 3rd axis:	5
No. 1023 4th axis:	3
No. 1023 5th axis:	4
No. 1023 6th axis:	6

(8) Parameter setting related to feed–forward control

1) To set the velocity loop to 1 msec, modify the motor type for the workpiece axis and EGB axis, thus reperforming automatic setting.

   Parameter No. 2020 = Motor number for 1 msec velocity loop
   Parameter No. 2000 bit 1 = 0
   Set the above, then turn the power off then back on again.

2) Re–set the parameters related to the EGB.

   Parameter No. 2011 bit 0 = 1 (for both the workpiece and EGB axes)
   Parameter No. 2011 bit 1 = 1 (for both the workpiece and EGB axes)(Note)

> **NOTE**
> Set this parameter when applying feed–forward control to rapid traverse also.

(3) Other parameters

Parameter No. 2003 bit 3 = 1 (P–I control)
Parameter No. 2005 bit 1 = 1 (feed–forward control enabled)
Parameter No. 2068 = 10000 (feed–forward factor)

(4) Suppressing load variation

Increase the value of parameter No. 2021 (within the range in which the motor does not oscillate).
Set this parameter to the value obtained from the following:
256 x (machine load inertia)/(motor rotor inertia)

For details of parameter setting, refer to ”Feed–forward setting” in ”Geometric error suppression function” in the FANUC AC SERVO MOTOR α Series Parameter Manual (B–65150E).

	#7	#6	#5	#4	#3	#2	#1	#0
2011							FFALWY	SYNMOD

SYNMOD EGB synchronous control is:
1 : Performed.
0 : Not performed.

FFALWY The feed–forward function is:
1 : Always enabled.
0 : Enabled only for cutting feed.

	#7	#6	#5	#4	#3	#2	#1	#0
7700						HDR		HBR

[Data type] Bit

HBR 0 : Performing a reset cancels synchronous of the C–axis (G81).
1 : Performing a reset does not cancel synchronous of the C–axis (G81).

HDR Setting of the direction for compensating a helical gear (1 is usually specified.)

7709	Number of the axial feed axis for a helical gear

[Data type] Byte

[Valid range] 1 to the maximum number of controlled axes

This parameter sets the number of the axial feed axis for a helical gear. If the value out of the valid range is specified, 3 (the 3rd axis) is specified.

> **NOTE**
> After setting this parameter, the power must be turned off then on again.

	#7	#6	#5	#4	#3	#2	#1	#0
7730								RTRx

[Data type] Bit axis

RTRx Specifies whether the retraction function is effective for each axis.
0 : Retraction is disabled.
1 : Retraction is enabled.

7740	Feedrate during retraction for each axis

[Data type] 2–word axis

[Unit of data and valid range]

Increment system	Unit of data	Valid range	
		IS–B	IS–C
Millimeter machine	1 mm/min	30 to 240000	30 to 100000
Inch machine	0.1 inch/min	30 to 96000	30 to 48000

This parameter sets the feedrate during retraction for each axis.

7741	Retracted distance for each axis

[Data type] 2–word axis

[Valid range] ±99999999

Unit of data	Valid range	
	IS–B	IS–C
Millimeter input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

7771	Number of EGB axis

NOTE
After setting this parameter, turn off the power. Then, turn the power back on to enable the setting.

[Data type] Byte

[Valid data range] 1 to the number of controlled axes

This parameter specifies the number of the EGB axis.

NOTE
1 You cannot specify four because the fourth axis is used as the workpiece axis.
2 For a machine using the inch increment system, linear axes cannot be used as the EGB axis.

7772	Number of position detector pulses per rotation about tool axis

[Data type] 2–word

[Data unit] Detection unit

[Valid data range] 1 to 99999999

This parameter specifies the number of pulses per rotation about the tool axis (on the spindle side), for the position detector.

> **NOTE**
> Specify the number of feedback pulses per rotation about the tool axis for the position detector, considering the gear ratio with respect to the position coder.

7773	Number of position detector pulses per rotation about workpiece axis

[Data type] 2–word

[Data unit] Detection unit

[Valid data range] 1 to 99999999

This parameter specifies the number of pulses per rotation about the workpiece axis (on the fourth axis side), for the position detector.

[Example] The number of feedback pulses for the position detector is 360000 for a rotation axis for which the detection unit is 0.001 deg.

Alarm and message

Number	Message	Contents
181	FORMAT ERROR IN G81 BLOCK (gear hobbing machine, EGB)	G81 block format error (hobbing machine) 1) T (number of teeth) has not been instructed. 2) Data outside the command range was instructed by either T, L, Q or P. 3) Calculation of the synchronous coefficient has overflowed. Modify the program.
184	ILLEGAL COMMAND IN G81 (gear hobbing machine, EGB)	A command not to be instructed during synchronous by G81 was instructed. (hobbing machine) 1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed. 2) Inch/Metric switching by G20, G21 was instructed.
186	PARAMETER SETTING ERROR (gear hobbing machine, EGB)	Parameter error regarding G81 (hobbing machine) 1) The C axis has not been set to be a rotary axis. 2) A hob axis and position coder gear ratio setting error

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.8	Simple electric gear box (G80, G81)

2 PREPARATIONS FOR OPERATION

2.1 EMERGENCY STOP

General

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

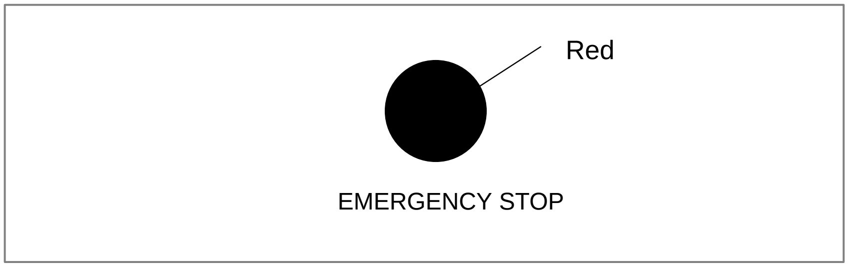


Fig. 2.1 (a) EMERGENCY STOP

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Signal

Emergency stop *ESP<X008#4,G008#4>

[Classification] Input signal

[Function] 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 stored stroke check function, and a limit switch for normal overtravel detection is not needed. To prevent the machine from moving beyond the software limit through servo feedback error, always install a stroke end limit switch (shown in Fig. 2.1 (b) as follows).

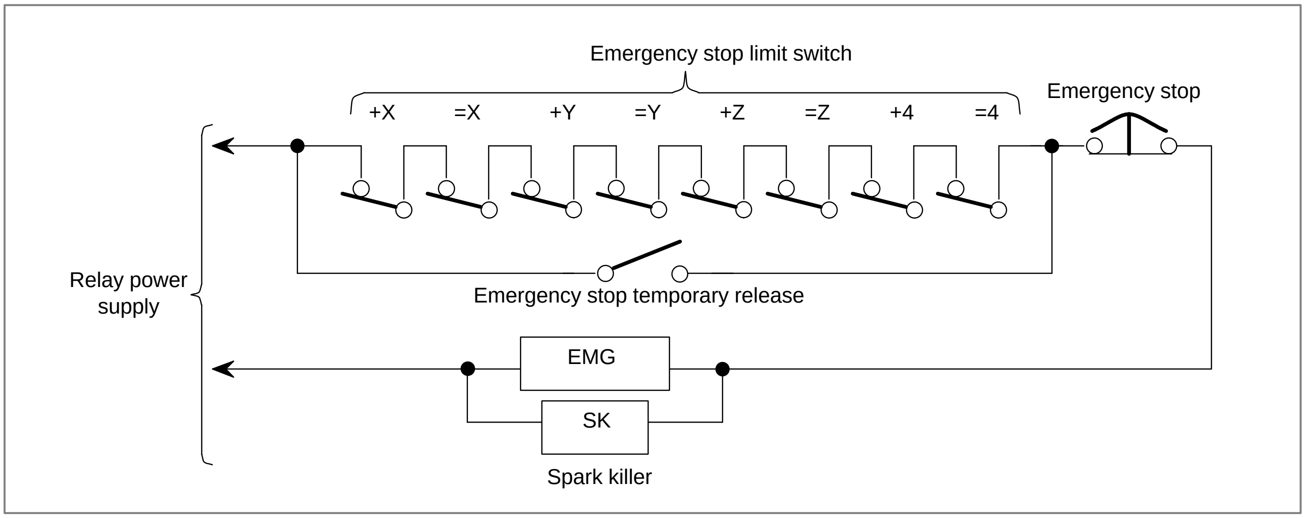


Fig. 2.1 (b) Connection of emergency stop limit switch

The distance from the position where the dynamic brake is applied to that where the tool stops moving is given in the “AC Servo Motor Descriptions.”

WARNING

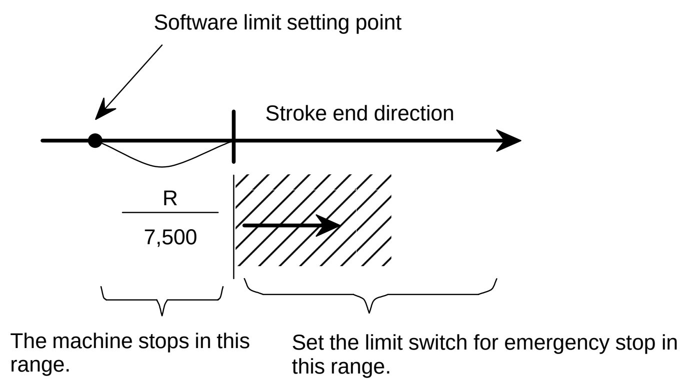
Software limit setting point and operating point of limit switch for emergency stop

The stop point by the software limit goes beyond the setting point by as much as the following distance.

$$\frac{R}{7{,}500}\ \text{(mm)}$$

R: Rapid traverse rate (mm/min)

The actual stopping point may exceed the position set by a parameter (Nos.1320 and 1321) by as much as R/7500 (mm). Set the limit switch for emergency stop including the allowance for the above value.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X008				*ESP				

	#7	#6	#5	#4	#3	#2	#1	#0
G008				*ESP				

Reference item

FANUC AC SERVO MOTOR α 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<F001#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.

	#7	#6	#5	#4	#3	#2	#1	#0
F001	MA							

Servo Ready Signal SA <F000#6>

[Classification] Output signal

[Function] Signal SA turns to “1” when the servo system is ready to operate. For an axis that is to be braked, release the brake when this signal is sent and apply the brake when this signal is not sent.
Time chart of this signal is as follows:

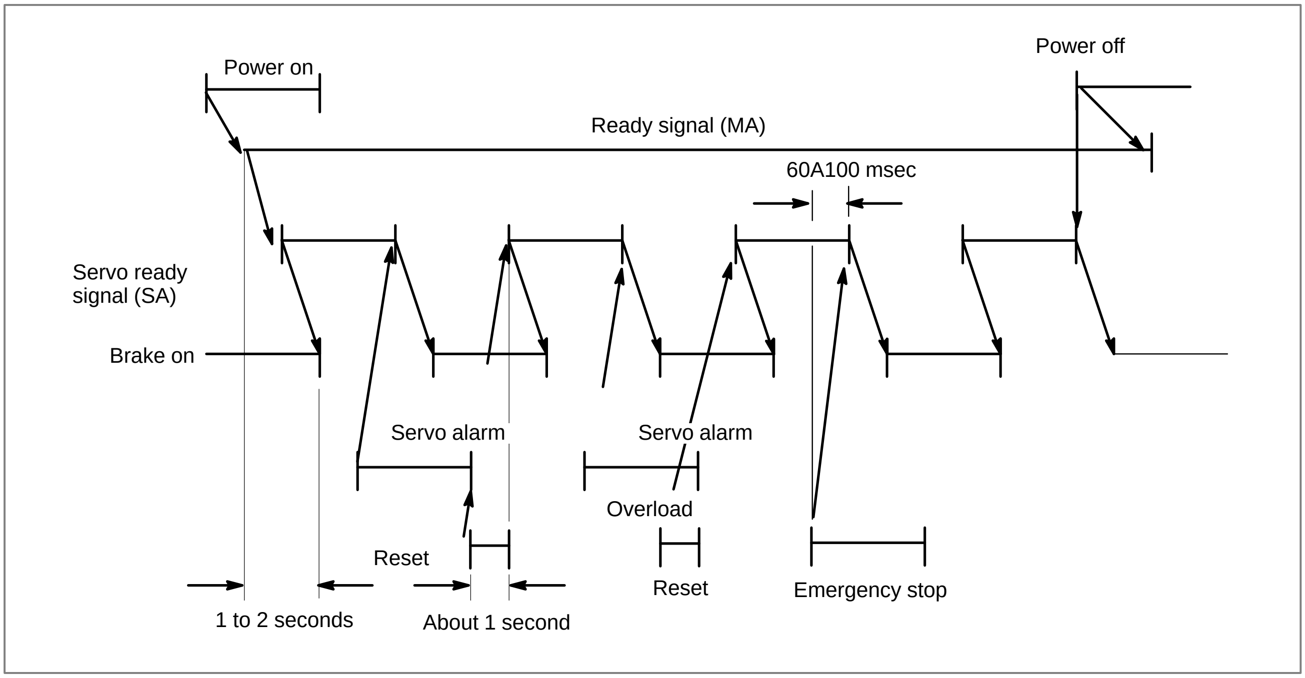


Fig. 2.2 Time chart for servo ready signal

	#7	#6	#5	#4	#3	#2	#1	#0
F000		SA						

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 *+L1 to *+L8<G114> *+L1 to *–L8<G116>

[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 number corresponds to the control axis.

+ L 1

- 1 No. 1 axis is at stroke limit.
- 2 No. 2 axis is at stroke limit.
- 3 No. 3 axis is at stroke limit.
- : :
- 8 No. 8 axis is at stroke limit.

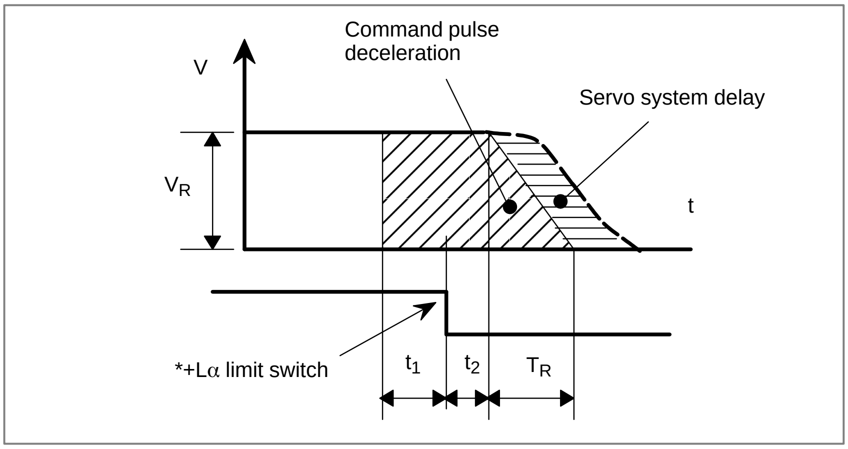
- \+ Has moved to and reached a + direction.
- – Has moved to and reached a – direction.

[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))

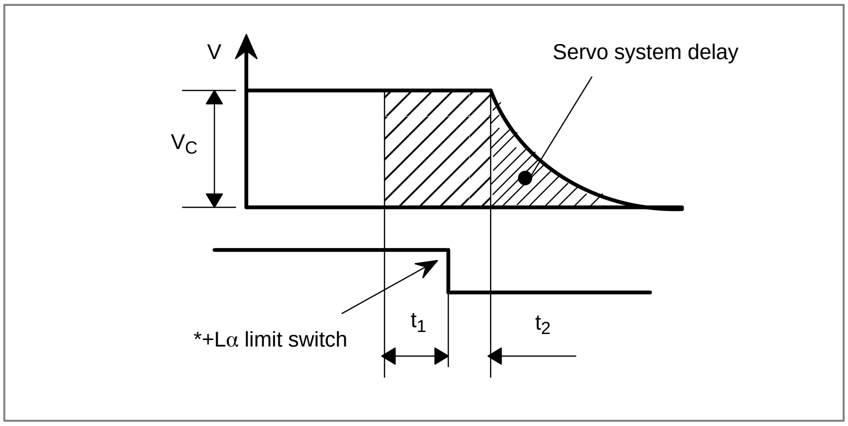
t_2: Receiver delay time 30ms

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

T_S: Servo system time constant (ms)

NOTE
Servo system time constant TS is 33 msec when the servo unit is adjusted to the standard setting.

(ii)Cutting feed



$$L_2=V_C(t_1+t_2+\frac{T_R}{2}+T_S)\cdot\frac{1}{60000}\ \text{[mm or inch]}$$

L_2:Deceleration distance

Vc: Maximum feedrate (mm/min or inch/min)

t_1, t_2, Ts: Same as (i).

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G114	*+L8	*+L7	*+L6	*+L5	*+L4	*+L3	*+L2	*+L1

	#7	#6	#5	#4	#3	#2	#1	#0
G116	*–L8	*–L7	*–L6	*–L5	*–L4	*–L3	*–L2	*–L1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

[Data type] Bit

OTH The overtravel signal is:

0 : Checked
1 : Not checked

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

Alarm and message

Number	Message	Description
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.6.2	Overtravel
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.6.2	Overtravel
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.6.2	Overtravel
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.6.2	Overtravel

2.3.2 Stored Stroke Check 1

General

When the tool tries to exceed 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. 1320, 1321 or Nos. 1326, 1327) 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 EXLM <G007#6>

[Classification] Input signal

[Function] Selects stroke check 1–I (parameter Nos. 1320 and 1321) or stroke check 1–II (parameter Nos. 1326 and 1327).

[Operation] When this signal is set to 1, the control unit operates as follows:
– Checks stroke check 1 on the basis of parameter Nos. 1326 and 1327, instead of parameter Nos, 1320 and 1321.

Stroke check external setting signals +LM1 to +LM8 <G110> and –LM1 to –LM8 <G112> (M series)

[Classification] Input signal

[Function] Change the values of the parameters governing the stroke check (1320 and 1321).

[Operation] When these signals are set to 1, the control unit operates as follows:
– Change the stored checks, set with parameter Nos. 1320 and 1321, to the machine coordinates when the signals are input.

Stroke check release signal RLSOT <G007#7> (M series)

[Classification] Input signal

[Function] Selects whether the stored stroke check 1 are checked.

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

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007	RLSOT	EXLM						
G110	+LM8	+LM7	+LM6	+LM5	+LM4	+LM3	+LM2	+LM1
G112	–LM8	–LM7	–LM6	–LM5	–LM4	–LM3	–LM2	–LM1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR				LMS		
	BFA	LZR				LMS		

[Data type] Bit

LMS The EXLM signal for switching stored stroke check 1

0: Disabled
1: Enabled

LZR Checking of stored stroke check 1 during the time from power–on to the manual position reference return

0: The stroke check 1 is checked.
1: The stroke check 1 is not checked

NOTE
When the absolute–position detector is being used, and the reference position is already set at power–on, the stored stroke check is checked immediately after the power is turned on, regardless of the setting of this bit.

BFA When a command that exceeds a stored stroke check 1, 3 is issued

0: An alarm is generated after the stroke check 1, 3 is exceeded.
1: An alarm is generated before the stroke check 1, 3 is exceeded.

1320	Coordinate value I of stored stroke check 1 in the positive direction on each axis
1321	Coordinate value I of stored stroke check 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] – 99999999 to 99999999

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:
parameter 1320 < parameter 1321

For movement along the axis for which infinite stroke is set, only incremental commands are available. If an absolute command is issued for this axis, the absolute register may overflow, and normal movement will not result.

1326	Coordinate value II of stored stroke check 1 in the positive direction on each axis
1327	Coordinate value II of stored stroke check 1 in the negative direction each axis

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] – 99999999 to 99999999

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 EXLM is ON, stroke checks are checked with parameters 1326 and 1327, not with parameters 1320 and 1321. The area outside that set by parameters 1326 and 1327 is inhibited.

NOTE
The EXLM signal is enabled only when LMS, #2 of parameter 1300, is set to 1.

Alarm and message

Number	Message	Description
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check I.
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – 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 LZR (bit 6 of No. 1300) selects whether each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on.

2 For the 2–path control, set a forbidden area for each path.

3 Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.6.3	Stroke check
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.6.3	Stroke check
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.6.3	Stroke check
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.6.3	Stroke check

2.3.3 Stored Stroke Check 2, 3

General

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2,and stored stroke check 3.

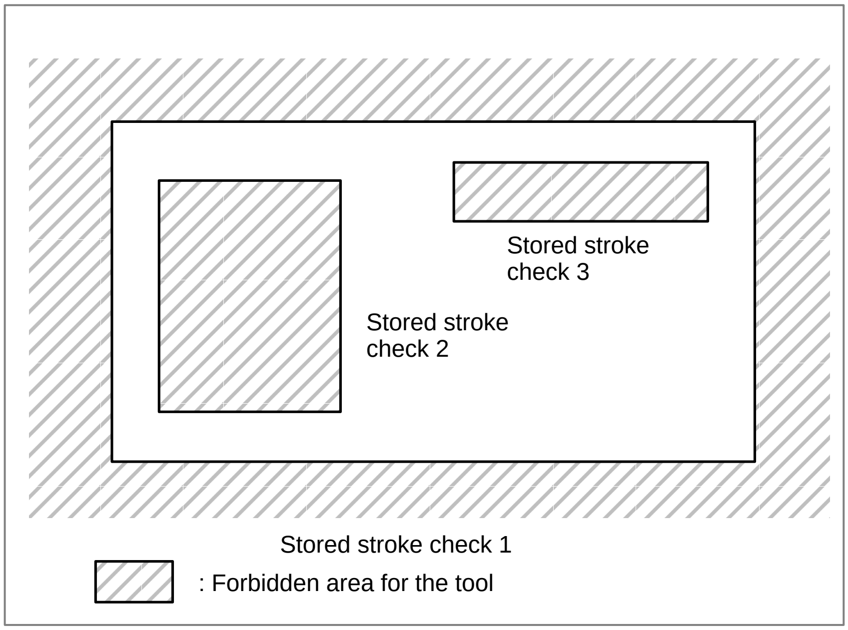


Fig. 2.3.3 (a) Stroke check (T series)

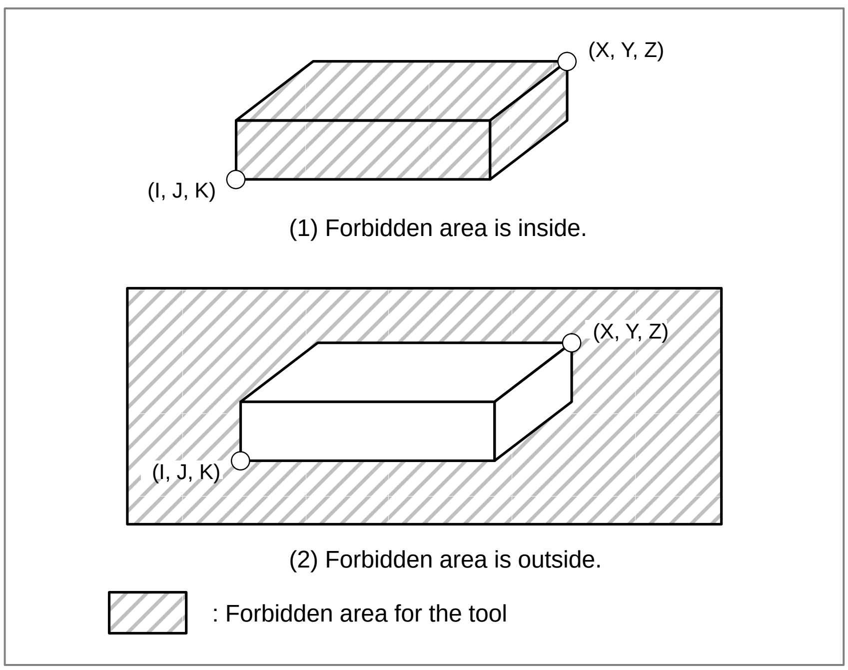


Fig. 2.3.3 (b) Stroke check (M series)

When the tool 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 2

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the check can be set as the forbidden area. Parameter OUT (No. 1300#0) 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:

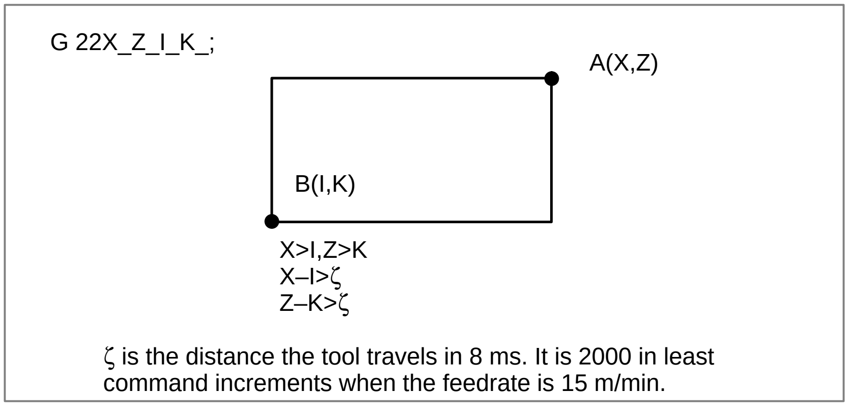


Fig. 2.3.3 (c) Creating or changing the forbidden area using a program (T series)

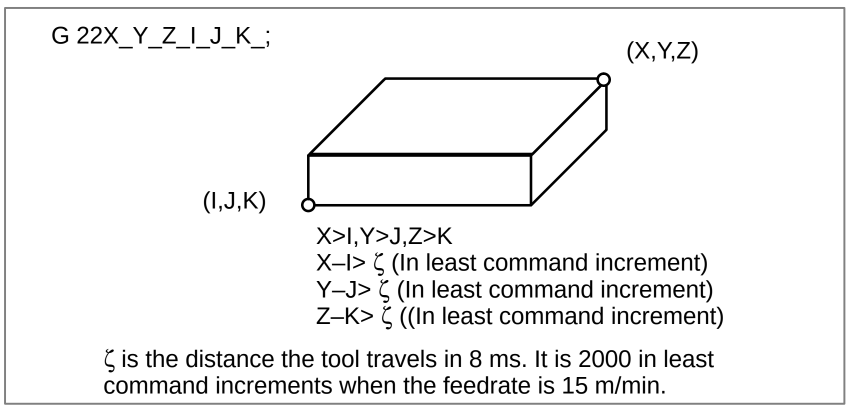


Fig. 2.3.3 (d) Creating or changing the forbidden area using a program (M series)

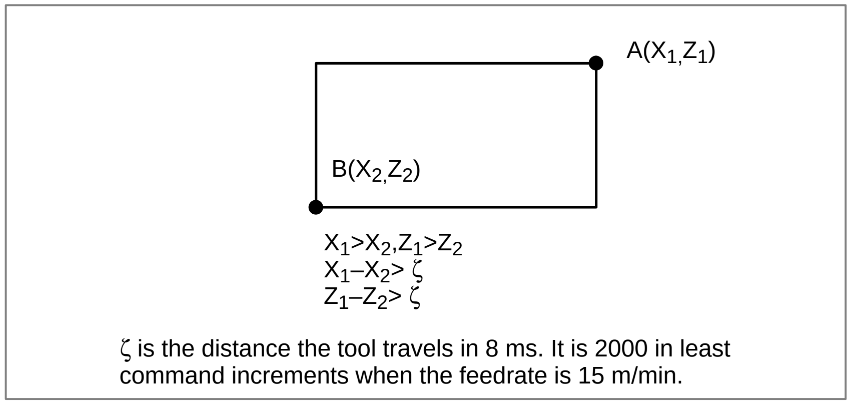


Fig. 2.3.3 (e) Creating or changing the forbidden area using a parameters (T series)

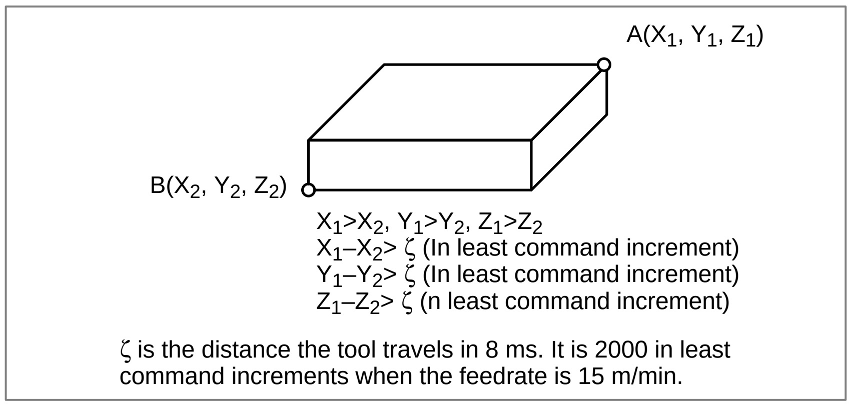


Fig. 2.3.3 (f) Creating or changing the forbidden area using a parameters (M series)

When you set the forbidden area X_1, Y_1, Z_1, X_2, Y_2 and Z_2 through parameters (Nos. 1322, 1323), the data should be specified by the distance from the reference position in the least command increment (output increment).
If set the forbidden area X, Y, Z, I, J, K (X, Z, I, K, on T series) 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 3**

Set the boundary with parameters Nos. 1324 and 1325. The area inside the boundary becomes the forbidden area.

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.
If point C (The 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.

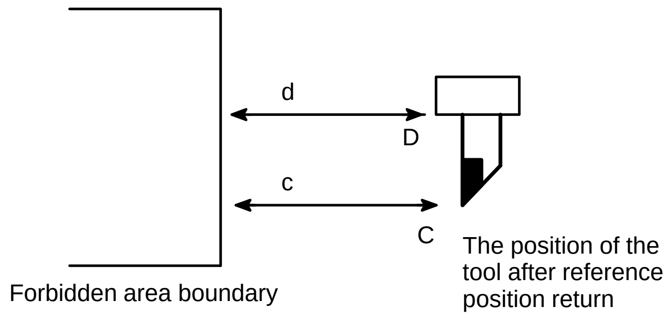


Fig. 2.3.3 (g) Setting the forbidden area (T series)

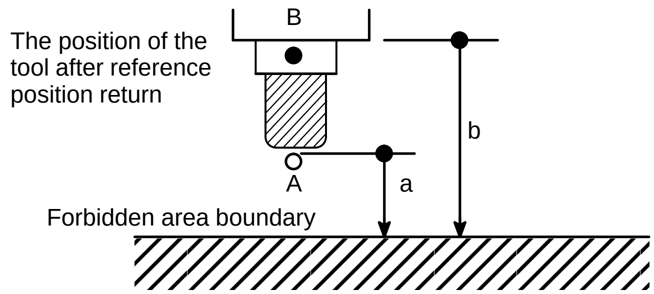


Fig. 2.3.3 (h) Setting the forbidden area (M series)

- **Forbidden area over–lapping**

Area can be set in piles.

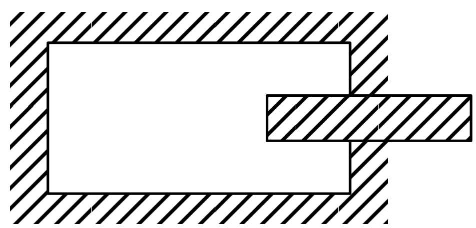
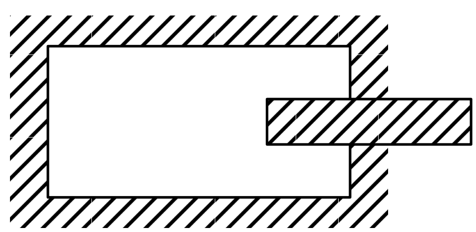


Fig. 2.3.3 (i) Setting the forbidden area overlapping (T series)

Unnecessary checks should be set beyond the machine stroke.



Setting the forbidden area over lapping

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

- **Effective time for a forbidden area**

Parameter LZR (bit 6 of No. 1300) selects whether each check becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed or immediately after the power is turned on.
After the power is turned on, if the reference position is in the forbidden area of each check, an alarm is generated immediately (Only in G22 mode for stored stroke check 2).

- **Releasing the alarms**

When the tool enters and forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came. Please move the tool out of the forbidden area, then please release the alarm by reset. If succeeded, the tool become movable to both paths.

- **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 2–path control**

For the 2–path control, set a forbidden area for each tool post.

Signal

Storoke check 3 release signal RLSOT3 <G007#4>

[Classification] Input signal

[Function] Selects whether stored stroke check 3 is checked.

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

· The control unit does not check stored stroke check 3.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007				RLSOT3				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR	RL3					OUT

[Data type] Bit

OUT The area inside or outside of the stored stroke check 2 is set as an inhibition area.

0: Inside
1: Outside

RL3 Stroke check 3 release signal RLSOT3

0: The signal is disabled.
1: The signal is enabled.

LZR Checking of stored stroke check 1 during the time from power–on to the manual position reference return

0: The stroke check 1 is checked.
1: The stroke check 1 is not checked

> **NOTE**
> When the absolute–position detector is being used, and the reference position is already set at power–on, the stored stroke check is checked immediately after the power is turned on, regardless of the setting of this bit.

BFA When a command that exceeds a stored stroke check 1, 3 is issued

0: An alarm is generated after the stroke check 1, 3 is exceeded.
1: An alarm is generated before the stroke check 1, 3 is exceeded.

	#7	#6	#5	#4	#3	#2	#1	#0
1310							OT3x	OT2x

[Data type] Bit axis

OT2x Whether stored stroke check 2 is checked for each axis is set.

0: Stored stroke check 2 is not checked.
1: Stored stroke check 2 is checked.

OT3x Whether stored stroke check 3 is checked for each axis is set.

0: Stored stroke check 3 is not checked.
1: Stored stroke check 3 is checked.

1322	Coordinate value of stored stroke check 2 in the positive direction on each axis
1323	Coordinate value of stored stroke check 2 in the negative direction on each axis

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] – 99999999 to 99999999

Set the coordinate values of stored stroke checks 2 in the positive and negative directions for each axis in the machine coordinate system. OUT, #0 of parameter 1300, sets either the area outside or the area inside specified by two checks as the inhibition area.

> **WARNING**
> For axes with diameter specification, a diameter value must be set.

1324	Coordinate value of stored stored check 3 in the positive direction on each axis
1325	Coordinate value of stored stroke check 3 in the negatice direction on each axis

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] – 99999999 to 99999999

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
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check 2. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke check 2. (Parameter No.1323)
504	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke check 3. (Parameter No.1324)
505	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke check 3. (Parameter No.1325)

Warning

WARNING

1 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 2 or check 3, all areas are movable areas.

2 In setting a forbidden area, if the two points are set in the wrong order, the area is set as follows:
(1) When the forbidden area is check 1, the stroke check becomes infinite.
(2) When the forbidden area is check 2 or 3, a quadrangle with the two points assumed as vertexes is set as the check.

Note

NOTE
Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area. (check 1, 3 only)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.6.3	Stroke check
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.6.3	Stroke check
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.6.3	Stroke check
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.6.3	Stroke check

2.3.4 Chuck/Tailstock Barrier (T series)

General

The chuck/tailstock barrier function prevents damage to the machine by checking whether the tool tip interferes either the chuck or tailstock.
Specify an area into which the tool may not enter (entry–inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tailstock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.
The tool can be cleared from the area only by retracting it in the direction opposite to that in which the tool entered the area.

Signal

Tailstock barrier select signal *TSB <G060#7>

[Classification] Input signal

[Function] Enables or disables the tailstock barrier.

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

– Disables the tailstock barrier, even when the G22 command (stored stroke check on) is specified in the program.

G code	*TSB	Tailstock barrier	Chuck barrier
G22	0	Enabled	Enabled
	1	Disabled	Enabled
G23	0	Disabled	Disabled
	1	Disabled	Disabled

When the G23 command (stored stroke check off) is specified, the tailstock barrier is disabled regardless of the *TSB signal. When the G22 command (stored stroke check on) is specified, the tailstock can be disabled by setting the signal to 1.

This signal is used to select whether the tailstock area is forbidden, when machining a workpiece by attaching and detaching the tailstock to and from the workpiece according to M commands.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G060	*TSB							

Parameter

● Profile of a chuck

1330	Profile of a chuck

[Data type] Byte

[Valid data range] 0 or 1

0 : Chuck which holds a workpiece on the inner surface
1 : Chuck which holds a workpiece on the outer surface

1331	Dimensions of the claw of a chuck (L)
1332	Dimensions of the claw of a chuck (W)
1333	Dimensions of the part of a claw at which a workpiece is held (L1)
1334	Dimensions of the part of a claw at which a workpiece is held (W1)

[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

1335	X coordinate of a chuck (CX)
1336	ZX coordinate of a chuck (CZ)

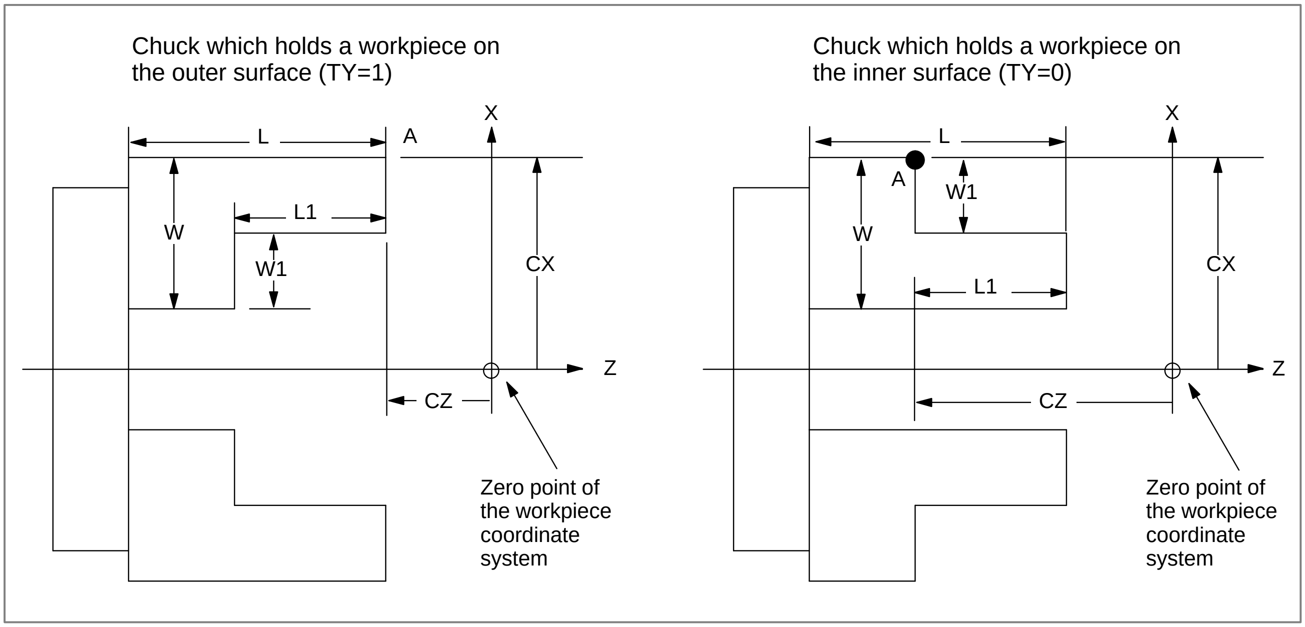
[Data type] Two–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

[Valid data range] – 99999999 to 99999999

Specify the profile of a chuck.



Symbol	Description
TY	Profile of a chuck (0: Chuck which holds a workpiece on the inner surface, 1: Chuck which holds a workpiece on the outer surface)
CX	X coordinate of a chuck
CZ	Z coordinate of a chuck
L	Dimensions of the claw of a chuck
W	Dimensions of the claw of a chuck (radius input)
L1	Dimensions of the part of a claw at which a workpiece is held
W1	Dimensions of the part of a claw at which a workpiece is held (radius input)

TY: Specifies the profile of a chuck. When TY is set to 0, the chuck holding a workpiece on the inner surface is specified. When TY is set to 1, the chuck holding a workpiece on the outer surface is specified. The profile of the chuck is assumed to be symmetrical with respect to the z–axis.

CX, and CZ: Specify the position (point A) of a chuck with the coordinates of the workpiece coordinate system. In this case, do not use the coordinates of the machine coordinate system.

> **WARNING**
> Specifying the coordinates with a diameter or radius depends on whether the corresponding axis conforms to diameter or radius specification . When the axis conforms to diameter specification, the coordinates is specified with a diameter.

L, L1, W, and W1: Define the profile of a chuck.

> **WARNING**
> Always specify W and W1 with radiuses. L and L1 is specified with radiuses when the Z–axis conforms to radius specification.

1341	Length of a tailstock (L)
1342	Diameter of a tail stock (D)
1343	Length of a tailstock (L1)
1344	Diameter of a tail stock (D1)
1345	Length of a tailstock (L2)
1346	Diameter of a tail stock (D2)
1347	Diameter of the hole of a tailstock (D3)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

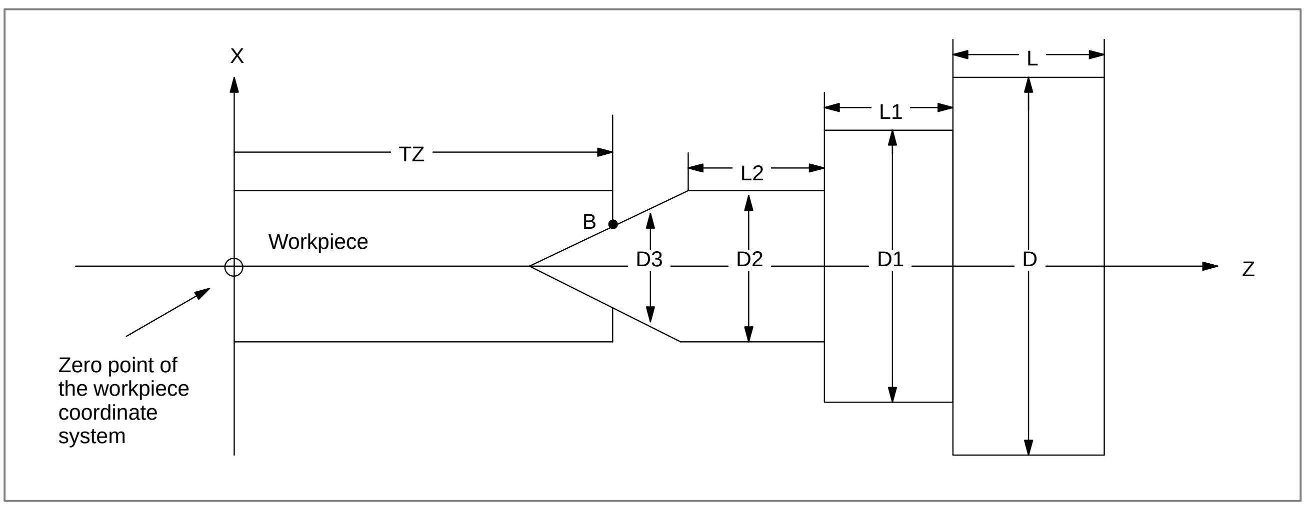
1348	Z coordinate of a tailstock (TZ)

[Data type] Two–word

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

Specify the profile of a tailstock.



Symbol	Description
TZ	Z–axis coordinate of a tailstock
L	Length of a tailstock
D	Diameter of a tailstock (diameter input)
L1	Length of a tailstock (1)
D1	Diameter of a tailstock (1) (diameter input)
L2	Length of a tailstock (2)
D2	Diameter of a tailstock (2) (diameter input)
D3	Diameter of the hole of a tailstock (diameter input)

TZ: Specifies the position (point B) of a tailstock with the Z–axis coordinate of the workpiece coordinate system. In this case, do not use the coordinate of the machine coordinate system. The profile of a tailstock is assumed to be symmetrical with respect to the Z–axis.

WARNING
Specifying the position of a tailstock with a radius or diameter depends on whether the Z–axis conforms to radius or diameter specification.

L, L1, L2, D, D1, D2, and D3:
Define the profile of a tailstock.

WARNING
D, D1, D2, and D3 is always specified with diameters. L, L1, and L2 is specified with radiuses if the Z–axis conforms to radius specification.

Alarm and message

Number	Message	Description
502	OVER TRAVEL : +X	The tool has entered the forbidden area when moving in the positive direction along the X–axis.
	OVER TRAVEL : +Z	The tool has entered the forbidden area when moving in the positive direction along the Z–axis.
503	OVER TRAVEL : –X	The tool has entered the forbidden area when moving in the negative direction along the X–axis.
	OVER TRAVEL : –Z	The tool has entered the forbidden area when moving in the negative direction along the Z–axis.

Warning

WARNING

1 If the setting for the forbidding area is invalid as follows, the area may not be forbidden:
   1) In the setting of the chuck shape, the jaw length (parameter No. 1331) is less than the grasp length (parameter No. 1333) or the jaw width (parameter No. 1332) is less than the grasp width (parameter No. 1334).
   2) In the setting of the tailstock shape, the tailstock diameter (parameter No. 1346) is less than the hole diameter (parameter No. 1347).
   3) The position of a chuck and the position of a tailstock is overlapped.

2 When the options for stored stroke check 2 and chuck/tailstock barrier are used at the same time, the chuck/tailstock barrier is valid but stored stroke check 2 is ignored.

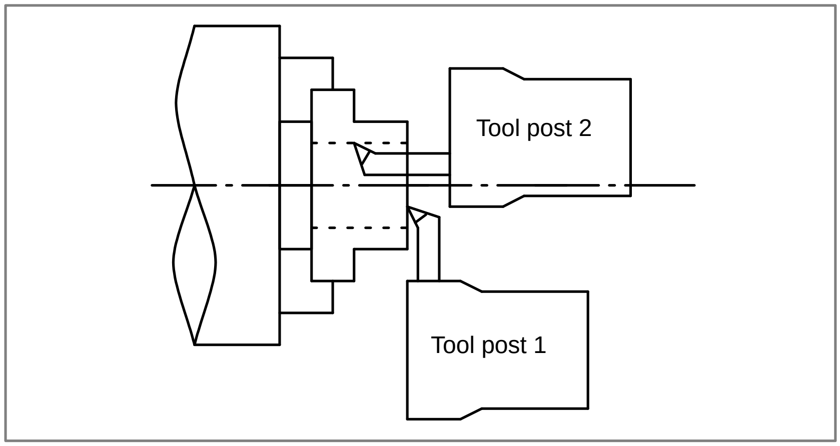
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.6.4	Chuck and Tailstock Barriers
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.6.4	Chuck and Tailstock Barriers

2.3.5 Tool Post Interference Check (T series, Two–path Control)

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 to decelerate and stop the two tool posts before they interfere with each other due to an incorrect command.



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

Signal

Tool post interference check signal TICHK <F064#6>

[Classification] Output signal

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

[Output condition] This signal goes "1" when:

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

This signal goes "0" when:

(i) The requisites for the tool post interference check function are not completely satisfied.

> **NOTE**
> The detailed requisites for the tool post interference check function, please refer to the operator's manual for Lathe.

Tool post interference alarm signal TIALM <F064#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 (i.e., the TICHK signal is “0”).

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 TIALM signal “1” to indicate that an interference alarm has occurred.

2 If the interference alarm is occured, 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 the point which 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 direction that caused the interference.

Signal address

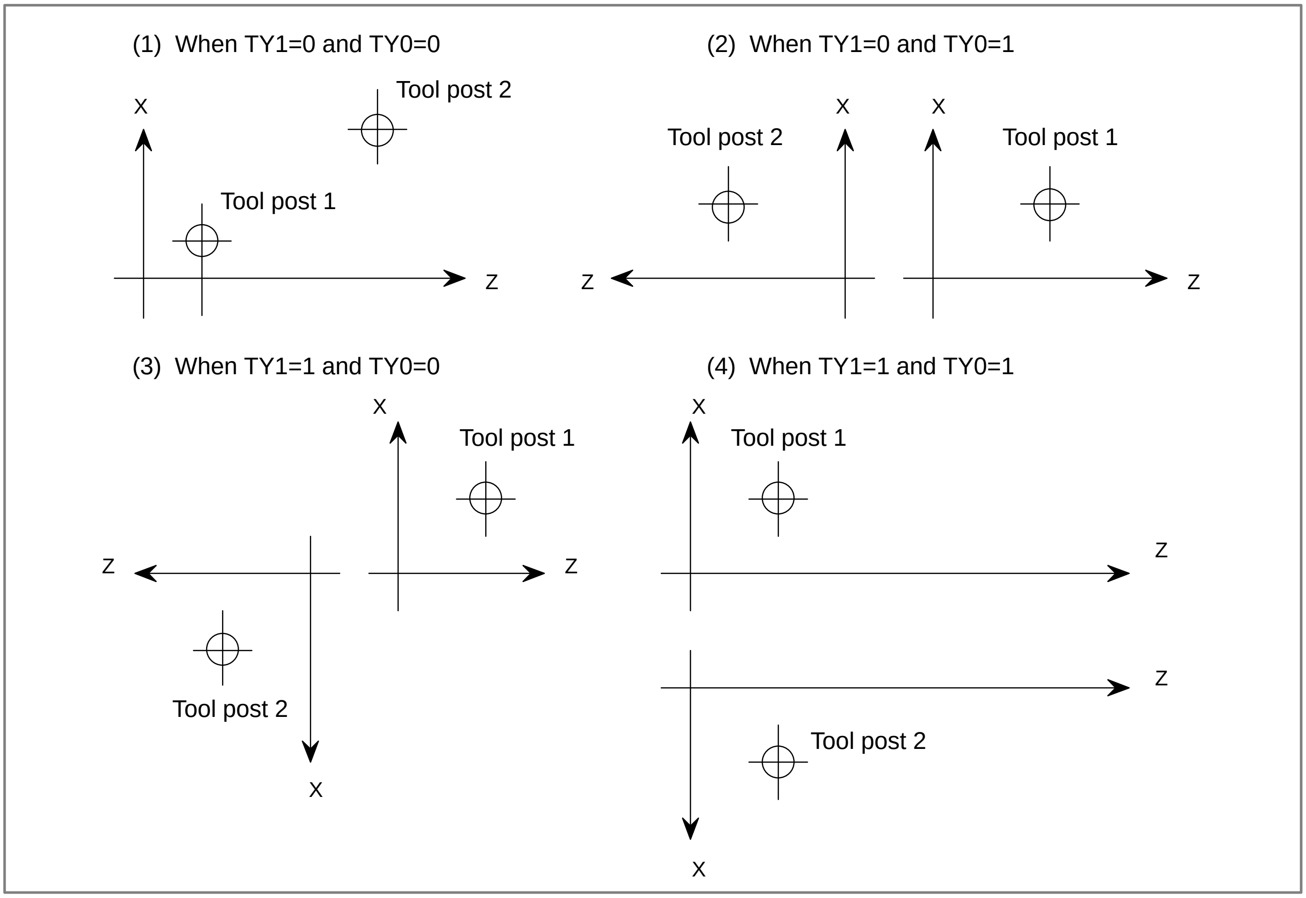
	#7	#6	#5	#4	#3	#2	#1	#0
F064	TIALM	TICHK						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8140			ZCL	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

ZCL 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.)

8151	Distance along the X axis between the reference positions of tool posts 1 and 2
8152	Distance along the Z axis between the reference positions of tool posts 1 and 2

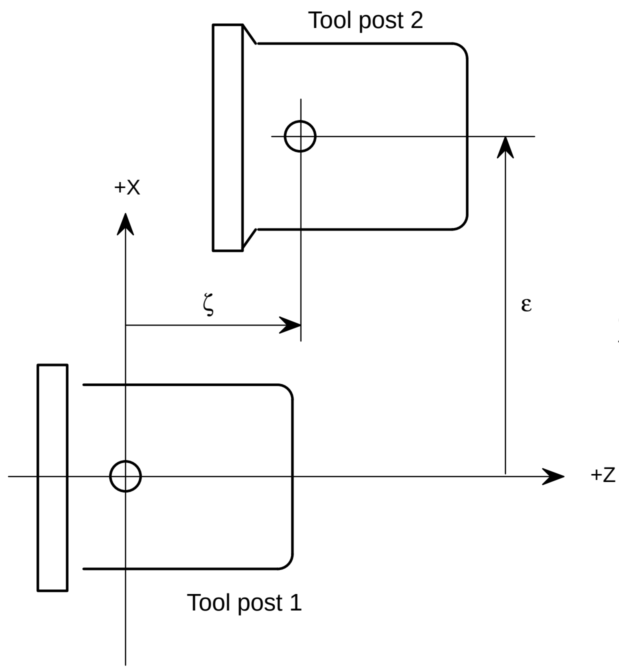
[Data type] Two–word

[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] 0 to 99999999

Indicating the distance between two tool posts.



Specify the X and Z values (ε, ζ) of the reference position of tool post 2 in parameters 8151 and 8152, 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.
508	INTERFERENCE : +X INTERFERENCE : +Z	An interference alarm has generated when X or Z axis is moving in the positive direction.
509	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 time delay. 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 should be set a little larger than the actual shape. 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, check that the interference forbidden area is correctly set by moving the tool posts to interfare with each other in several directions in manual mode (interference check enabled with a parameter).

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.20.3	Tool post interference check

2.3.6 Stroke Limit Check Before Move

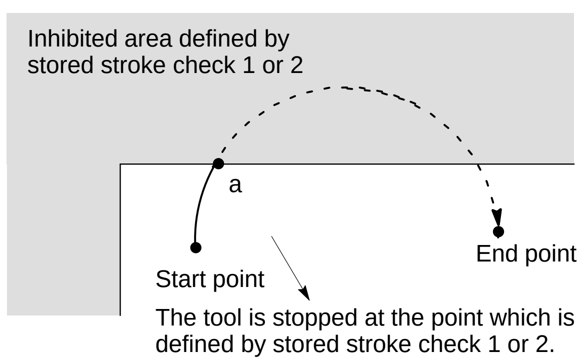
General

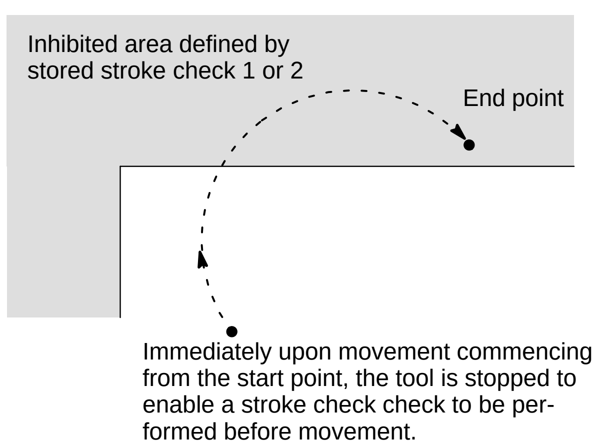
In the automatic operation before the move command by a given block, the position of the end point is determined by the current position of the machine and by a specified amount of travel. And it is determined whether or not the tool enters the inhibit area defined by stored stroke check 1, 2 or 3. If the tool is found to enter the inhibited area defined by a stored stroke check, the tool is stopped immediately after the start for that block, and an alarm is displayed.

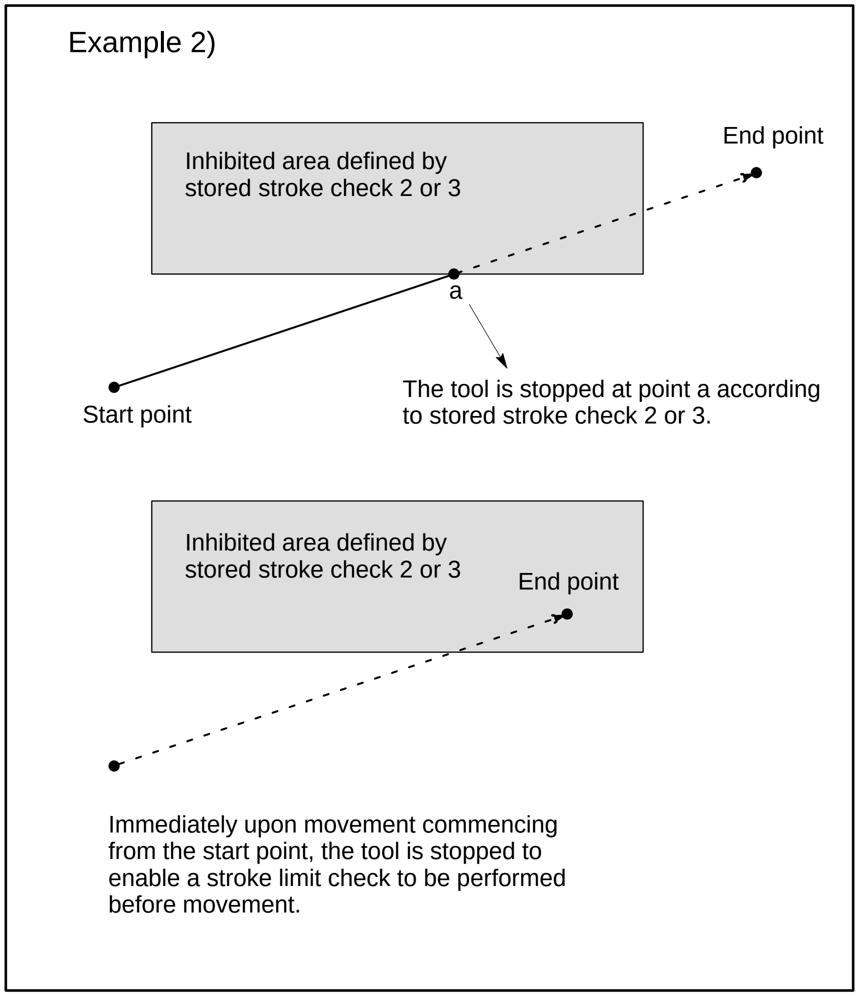
WARNING

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a inhibited area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the inhibited area defined by stored stroke check 1, 2, or 3, an alarm is issued. (See the examples below.)

Example 1)







Explanations

When a stroke limit check before move is performed, whether to check the move performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using NPC (bit 2 of parameter No. 1301).

Limitations

- **Machine lock**

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- **G23**

When stored stroke check 2 is disabled (G23 mode), no check is made to determine whether the tool enters the inhibited area defined by stored stroke check 2.

- **Program restart**

When a program is restarted, an alarm is issued if the restart position is within a inhibited area.

- **Manual intervention following a feed hold stop**

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point following a manual intervention is within a inhibited area.

- **A block consisting of multiple operations**

If a block consisting of multiple operations (such as a canned cycle and exponential interpolation (M series only) is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

- **Cyrindrical interpolation mode**

  In cylindrical interpolation mode, no check is made.

- **Polar coordinate interpolation mode**

  In polar coordinate interpolation mode, no check is made.

- **Angular axis control**

  When the angulalr axis control option is selected, no check is made.

- **Simple synchronous control**

  In simple synchronous control, only the master axis is checked; no slave axes are checked.

- **Three–dimensional coordinate conversion**

  In three–dimensional coordinate conversion mode, no check is made. (M series only)

- **Drawing**

  While drawing in dynamic graphic display mode (only drawing is performed), no check is made. (M series)

- **PMC axis control**

  No check is made for a movement based on PMC axis control.

- **High–speed high–precision contour control (HPCC)**

  No check is made for a movement based on high–speed, high–precision contour control (HPCC). (M series only)

- **Chuck/tailstock barrier**

  The chuck/tailstock barrier area is not checked. (T series)

- **Synchronous control and composite control**

  Axes subject to synchronous control and composite control are not checked. (T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1301	PLC					NPC		

[Data type] Bit

NPC As part of the stroke limit check performed before movement, the movement specified in G31 (skip) and G37 (automatic tool length measurement (for M series) or automatic tool compensation (for T series)) blocks is:

0: Checked
1: Not checked

PLC Stroke limit check before movement is:

0: Not performed
1: Performed

Alarm and message

Number	Message	Contents
510	OVER TRAVEL : +n	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit inhibited area in the positive direction of the n–axis. Correct the program.
511	OVER TRAVEL : –n	The stroke limit check made prior to performing movement reveals that the end point of a block is located within the stroke limit inhibited area in the negative direction of the n–axis. Correct the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.6.4	Stroke Limit Check Prior to Performing Movement
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.6.5	Stroke Limit Check Prior to Performing Movement

2.4 ALARM SIGNAL

General

When an alarm is triggered in the CNC, the alarm is indicated on the screen, and the alarm signal is set to 1.

If the voltage level of the memory backup battery falls to below a specified level while the CNC is turned off, the battery alarm signal is set to 1.

Signal

Alarm signal AL<F001#0>

[Classification] Output signal

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

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.

Battery alarm signal BAL<F001#2>

[Classification] Output signal

[Function] The battery alarm signal indicates that the voltage of the battery for the memory has fallen to below a specified level while the CNC is off. In general, this signal is used to turn on an LED to notify the operator.

[Output condition] The signal is set to 1 when:
– The battery voltage has fallen to below the specified level.
The signal is set to 0 when:
– The battery voltage has risen to the specified level or higher.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F001						BAL		AL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

[Data type] Bit

NPA Action taken when an alarm is generated or when an operator message is entered

0 : The display shifts to the alarm or message screen.
1 : The display does not shift to the alarm or message screen.

2.5 START LOCK/ INTERLOCK

General

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<G007#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, T, or B commands 2nd auxiliary function are executed consecutively until a block containing an axis move command is encountered; the movement 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)).

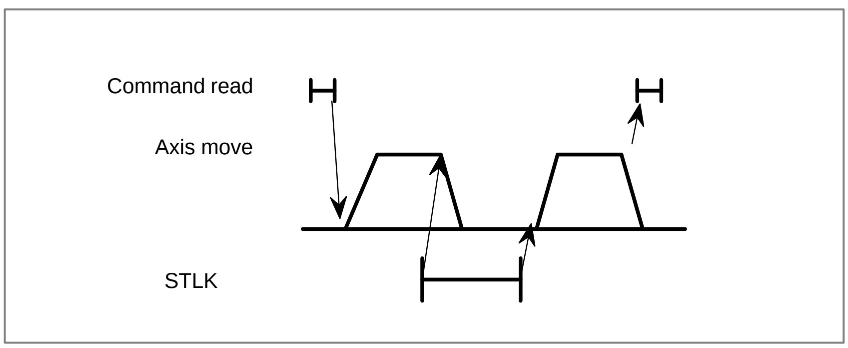


Fig. 2.5 (a) Block containing only axis move command

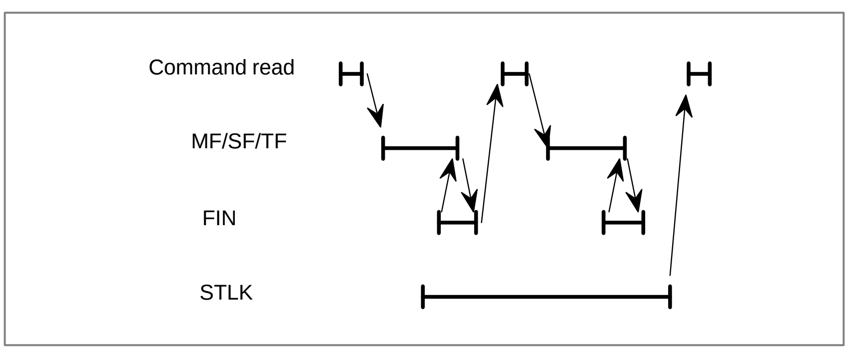


Fig. 2.5 (b) Block containing only auxiliary functions only

All axes Interlock signal *IT<G008#0>

[Classification] Input signal

[Function] This signal is used to inhibit the machine from moving, and is effective regardless of the selected mode.

[Operation] When the *IT signal is “0”, the axis movement is decelerated and stopped. In automatic operation, blocks containing M, S, T or B 2nd auxiliary mode commands are executed consecutively until a block containing an axis move command is encountered; the system then stops and is placed into the automatic operation mode (cycle start lamp signal STL is “1”, feed hold lamp signal SPL is “0”). When the *IT signal turns to “1”, operation resumes (Figs. 2.5(c), (d)).

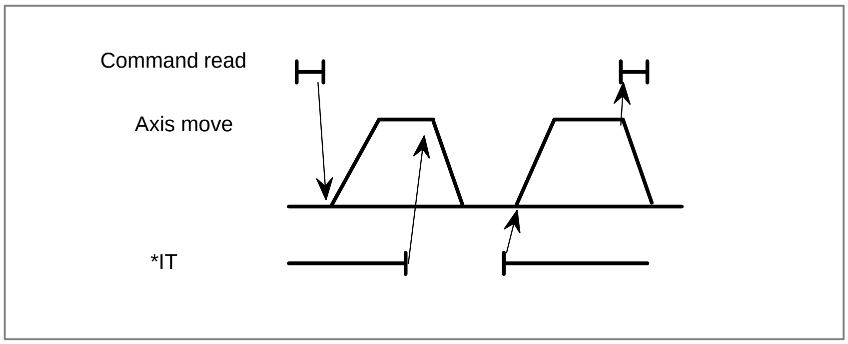


Fig. 2.5 (c) Block containing only axis move command (manual and automatic operation)

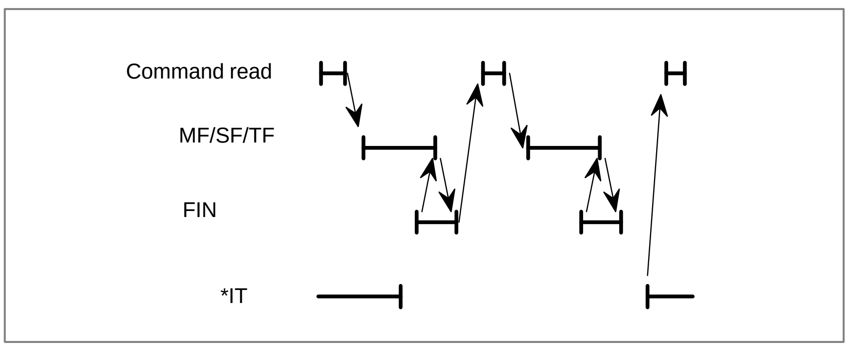


Fig. 2.5 (d) Block containing auxiliary functions only (automatic operation)

> **NOTE**
> The overtravel amount of the motor after turning *IT to “0” is represented by the following formula.
>
> $$Q_{max} = F_m \cdot \frac{1}{60} \times \left(\frac{T_c}{1000} + \frac{T_s}{1000} + \frac{A}{1000} \right)$$
>
> Where
> Q_{max} : Overtravel quantity (mm or inch)
> F_m : Feedrate (mm/min or inch/min)
> T_c : Cutting time constant (ms)
> T_s : Servo time constant (Ts = 33ms normally)
> A : Processing time of CNC
> A = 50ms

Interlock signal for each axis
*IT1 ~ *IT8<G130>

[Classification] Input signal

[Function] These signals disable feed along axes on an axis–by–axis basis.
A separate interlock signal is provided for each controlled axis. The number at the end of each signal name denotes the number of the corresponding controlled axis.

*IT 1

1 ..... Interlock for the first axis
2 ..... Interlock for the second axis
3 ..... Interlock for the third axis
: :
: :

[Operation] a) In manual operation

The movement of an interlocked axis is inhibited, but the other axes are movable. If an axis is interlocked during movement, it stops after being decelerated, and it starts moving again when it is released from interlock.

b) In automatic operation (MEM RMT or MDI mode)

If an axis is interlocked while its movement is being commanded (the move amount is not 0, inclusive of the tool offset), 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
+MIT1, –MIT1, +MIT2, –MIT2, +MIT3, –MIT3, +MIT4, –MIT4 <G132#0 to #3, G134#0 to #3> (M series)
+MIT1, –MIT1, +MIT2, –MIT2 <X004#2~X004#5>(T series)

[Classification] Input signal

[Function] This function allows a directional interlock for each axis.

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

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007							STLK	
G008								*IT
G130	*IT8	*IT7	*IT6	*IT5	*IT4	*IT3	*IT2	*IT1
G132					+MIT4	+MIT3	+MIT2	+MIT1
G134					–MIT4	–MIT3	–MIT2	–MIT1

	#7	#6	#5	#4	#3	#2	#1	#0
X004			–MIT2	+MIT2	–MIT1	+MIT1		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT	ITX		ITL

[Data type] Bit

ITL Interlock signal for all axes
0 : Enabled
1 : Disabled

ITX Interlock signals for each axis
0 : Enabled
1 : Disabled

DIT Interlock for each axis direction
0 : Enabled
1 : Disabled

Note

NOTE
The interlock signal for each axis and direction (T system) is supported regardless of whether the direct input of tool offset value measurement B is provided.

2.6 MODE SELECTION

General

The mode select signal is a code signal consisting of the three bits MD1, MD2, and MD4. The seven modes -- memory edit (EDIT), memory operation (MEM), manual data input (MDI), manual handle/incremental feed (HANDLE/INC), manual continuous feed (JOG), TEACH IN JOG, TEACH IN HANDLE -- can be selected. And in addition, DNC operation mode can be selected by combining the (MEM) mode setting and the DNCI signal. Manual reference position return mode can be selected by combining the manual continuous feed (JOG) mode setting and the ZRN signal.
The currently selected operation mode can be posted by outputting the operation mode check signal.

Signal

Mode selection signal MDI, MD2, MD4 <G043#0~#2> DNCI <G043#5> ZRN <G043#7>

[Classification] Input signal

[Operation] As shown in the following table, the mode select signal is a grey code (a code in which only one bit is different from that of the adjacent mode). To prevent faulty mode switching, use an overcrossing contact rotary switch so that only one bit changes from that of the adjacent mode.
”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 (MEN)	0	0	1	0	0
3	Manual data input (MDI)	0	0	0	0	0
4	Manual handle/incremental feed (HANDLE/INC)	1	0	0	0	0
5	Manual continuous feed (JOG)	1	0	1	0	0
6	TEACH IN HANDLE (THND)	1	1	1	0	0
7	TEACH IN JOG (TJOG)	1	1	0	0	0
8	DNC operation (RMT)	0	0	1	1	0
9	Manual reference position return (REF)	1	0	1	0	1

Operation mode check signal
MMDI, MMEM , MRMT, MEDT, MH, MINC, MJ, MREF, MTCHIN
<F003, F004#6>

[Classification] Output signal

[Function] The currently selected operation mode is output.

[Operation] The following lists the relationship between the mode selection signals and check signals:

Mode		Input signal					Output signal
		MD4	MD2	MD1	DNCI	ZRN	
Automatic operation	Manual data input (MDI) (MDI operation)	0	0	0	0	0	MMDI<F003#3>
	Memory operation (MEM)	0	0	1	0	0	MMEM<F003#5>
	DNC operation(RMT)	0	0	1	1	0	MRMT<F003#4>
Memory edition (EDIT)		0	1	1	0	0	MEDT<F003#6>
Manual operation	Manual handle feed / Incremental feed (HANDLE/INC)	1	0	0	0	0	MH<F003#1> MINC<F003#0>
	Manual continuous feed(JOG)	1	0	1	0	0	MJ<F003#2>
	Manual reference position return (REF)	1	0	1	0	1	MREF<F004#5>
	TEACH IN JOG (TJOG)	1	1	0	0	0	MTCHIN<F003#7>, MJ<F003#2>
	TEACH IN HANDLE (THND)	1	1	1	0	0	MTCHIN<F003#7>, MH<F003#1>

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN		DNCI			MD4	MD2	MD1

	#7	#6	#5	#4	#3	#2	#1	#0
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF					

Note

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 Manual operation in jog feed mode

a) When bit 0 (JHD) of parameter No. 7100 is set to 0
Only jog feed is possible.

b) When bit 0 (JHD) of parameter No. 7100 is set to 1
Both jog feed and manual handle feed are possible, provided the manual handle feed option is installed. Jog feed and manual handle feed cannot, however, be performed simultaneously. Manual handle feed can be performed when the tool is not being moved by means of jog feed.

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

a) When parameter JHD no.7100#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.7100#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.

NOTE

4 Manual operation in TEACH IN JOG mode

a) When bit 1 (THD) of parameter No. 7100 is set to 0
Only jog feed is possible.

b) When bit 1 (THD) of parameter No. 7100 is set to 1
Both jog feed and manual handle feed are possible, provided the manual handle feed option is installed. Jog feed and manual handle feed cannot, however, be performed simultaneously. Manual handle feed can be performed when the tool is not being moved by means of jog feed.

5 During operation in automatic operation mode (MEM, RMT, or MDI), specifying switching to another automatic operation mode (MEM, RMT, or MDI) or memory editing mode (EDIT) first places the CNC in the automatic operation stop state after executing the command of the current block, after which the mode is switched. At this time, signal STL is set to 0. Signal SPL is not, however, set to 1. (Fig. 2.6 (a), (b))

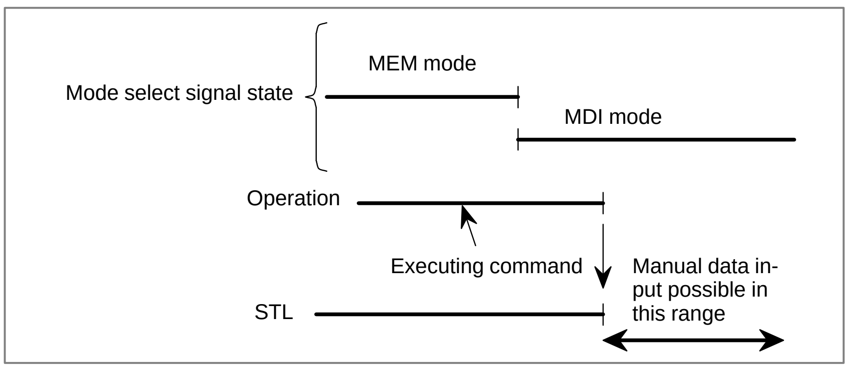


Fig. 2.6 (a)

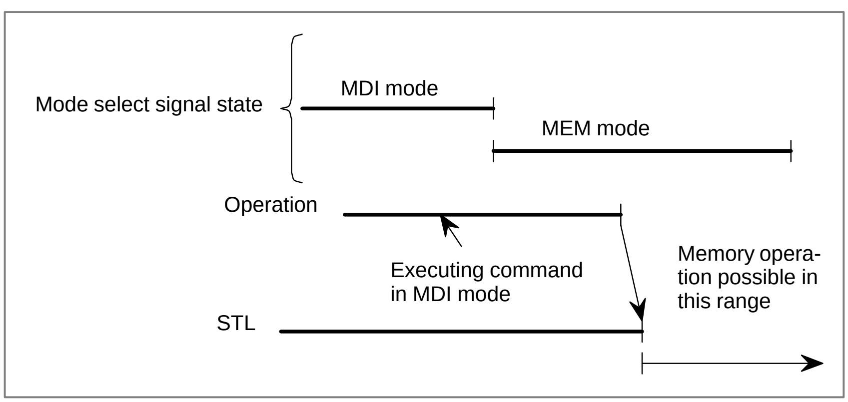


Fig. 2.6 (b)

NOTE

6 When the HANDLE/INC or TEACH IN HANDLE mode is selected while the CNC is operating in the MEM or MDI mode, the automatic or MDI operation stops, the STL signal turns to “0”, the SPL signal simultaneously turns to “1”, and the CNC enters the HANDLE/INC or TEACH IN HANDLE mode. 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)).

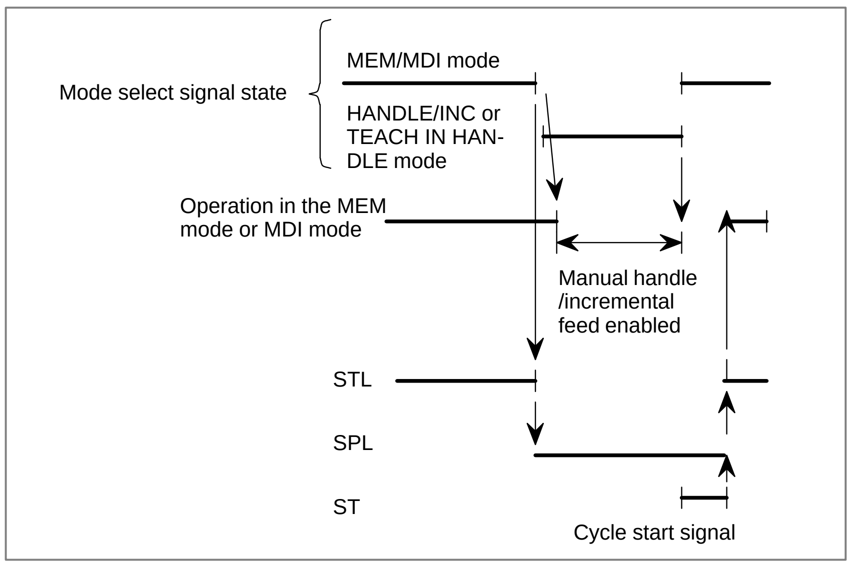


Fig. 2.6 (c)

NOTE

7 When the JOG or TEACH IN JOG mode is selected during RMT, MEM or MDI mode operation, operation stops, the STL signal turns to “0”, the SPL signal simultaneously turns to “1”, and the CNC enters the JOG or TEACH IN JOG mode. 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 (Fig. 2.6 (d)). 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” (Fig. 2.6 (e)). It is possible to perform handle feed in TEACH IN JOG mode by parameter THD no.7100#1. For details, refer to item (2), (4).

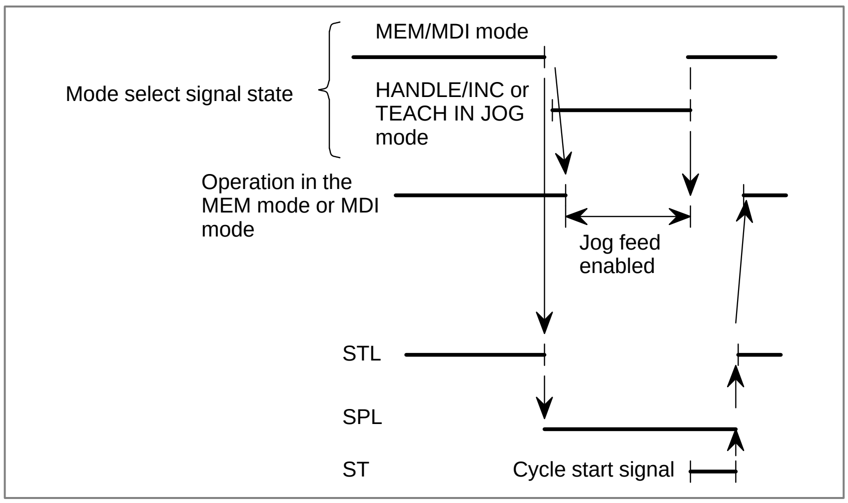


Fig. 2.6 (d)

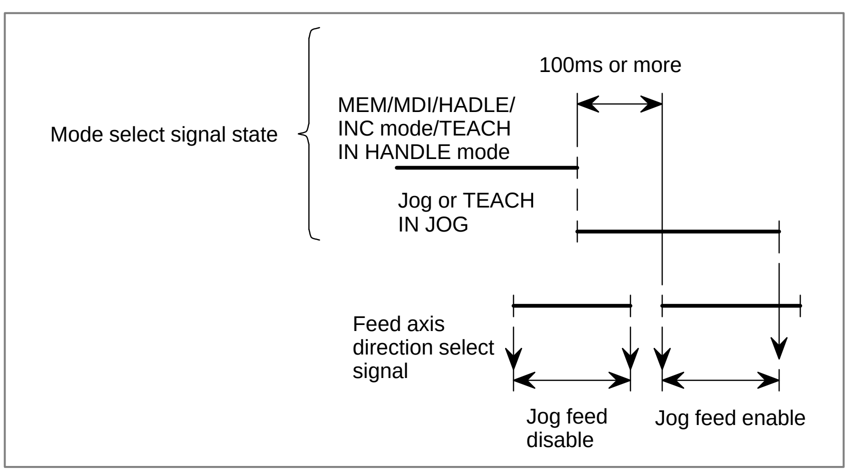


Fig. 2.6 (e)

NOTE

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

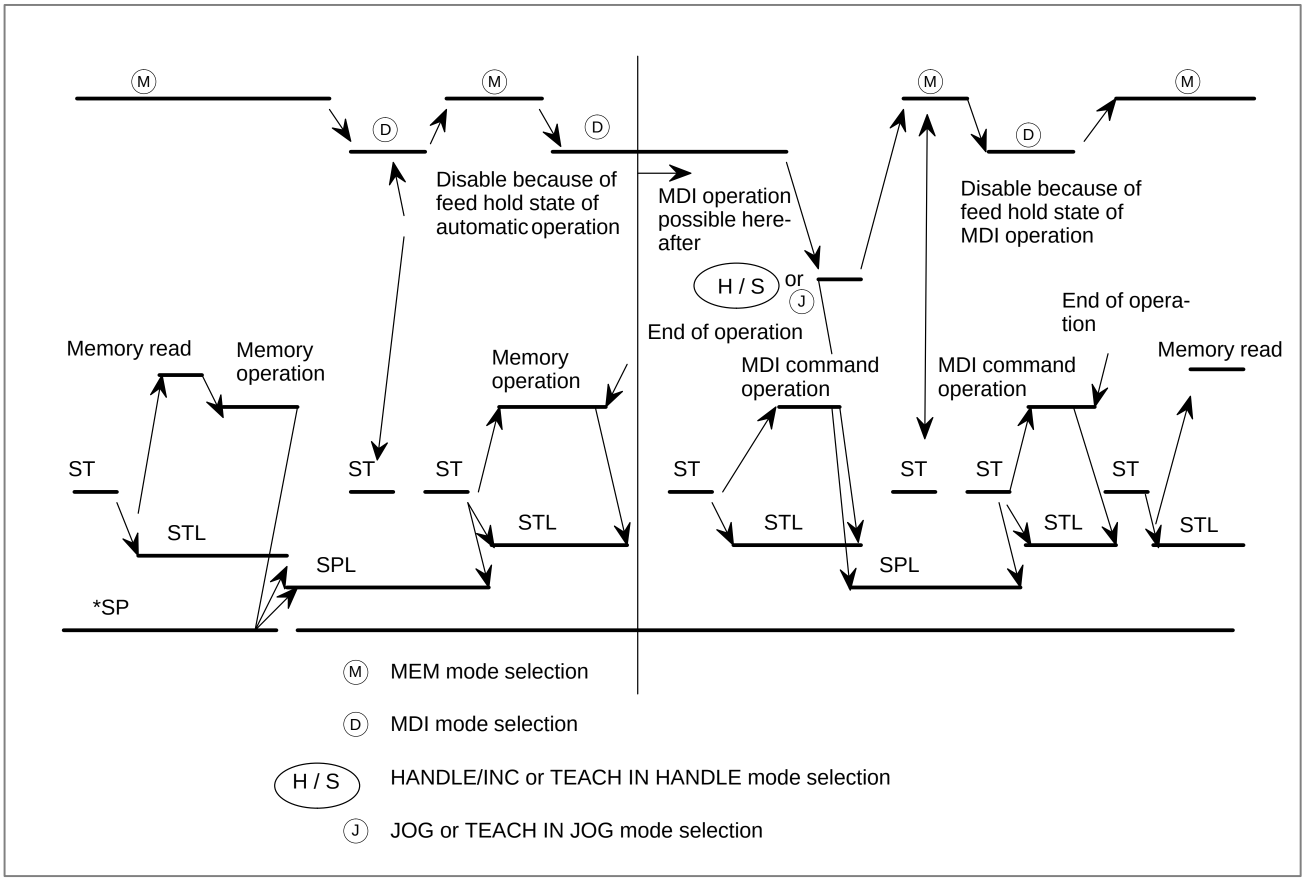


Fig. 2.6 (f) Mode signal time chart

Reference item

CONNECTION MANUAL (This manual)	4.1	Manual Reference Position Return

2.7 PATH SELECTION/ DISPLAY OF OPTIONAL PATH NAMES (TWO–PATH CONTROL)

General

Path selection specifies whether operations performed using the CRT/MDI panel are for path 1 or path 2.
The operations, as used here, include displaying and setting data items (such as tool compensation values), inputting command programs in the MDI mode, and editing machining programs in program memory.
Additionally, names of each path can be changed by parameter.

Signal

Path selection signal (Tool post selection signal) HEAD<G063#0>

[Classification] Input signal

[Function] Selects whether the CRT/MDI panel is used for path 1 or path 2.

[Operation] When this signal turned to “1”, operations performed using the CRT/MDI panel are for path 2.
When this signal turned to “0”, operations performed using the CRT/MDI panel are for path 1.

Signal address

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

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8100							IAL	RST

[Data type] Bit

RST Reset key on the CRT/MDI panel

0 : Effective for both paths
1 : Effective for the path selected by the path select signal

IAL When an alarm is raised in one tool post in the automatic operation mode,

0 : The other path enters the feed hold state and stops.
1 : The other path continues operation without stopping.

Parameters for display of optional path names

3141	Path name (1st character)
3142	Path name (2nd character)
3143	Path name (3rd character)
3144	Path name (4th character)
3145	Path name (5th character)
3146	Path name (6th character)
3147	Path name (7th character)

[Data type] Byte type

Specify a path name with codes (two–path control).
Any character strings consisting of alphanumeric characters and symbols (up to seven characters) can be displayed as path names on the CRT screen, instead of HEAD1 and HEAD2 for T series, and instead of PATH1 and PATH2 for M series.

NOTE
1 This parameter is dedicated to the two–path control.
2 Specify these parameters for each series.
3 For characters and codes, see the correspondence table in 2.1.15 software operator's panel.
4 When codes are 0, HEAD1 and HEAD2 for T series and PATH1 or PATH2 for M series are displayed.

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	MV1 – MV8	1.2.5
Moving direction signals	MVD1 – MVD8	1.2.5
In–position signals	INP1 – INP8	7.2.6.1
Rapid traversing signal	RPDO	2.8 (the section you are reading)
Cutting feed signal	CUT	2.8 (the section you are reading)
Thread cutting signal	THRD	6.4.1
Constant surface speed signal	CSS	9.5
Inch input signal	INCH	11.4

Signal

Rapid traversing signal RPDO <F002#1>

[Classification] Output signal

[Function] This signal indicates that a move command is executed at rapid traverse.

[Output condition] “1” indicates an axis starts moving after rapid traverse has been selected; “0” indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

NOTE

1 The rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. The rapid traverse in manual operation also includes the rapid traverse in reference position return.

2 Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement is started.

Cutting feed signal CUT <F002#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 interpolation, thread cutting, skip cutting, or cutting in 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002		CUT					RPDO	

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<G066#0>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, VRDY OFF, for all axes.

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

· The control unit does not issue servo alarm No. 401, VRDY OFF, even when the servo amplifier ready signal goes off. The control unit, however, sets servo ready signal SA to 0. The SA signal can remain set to 1, depending on the setting of SAK, bit 6 of parameter No. 1804.

Each–Axis VRDY OFF Alarm Ignore Signal IGVRY1 – IGVRY8 <G192>

[Classification] Input signal

[Function] Disables the detection of servo alarm No. 401, VRDY OFF, for the corresponding axis. These signals correspond to the controlled axes. The suffixed number of each signal corresponds to the number of the controlled axis.

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

· The control unit does not issue servo alarm No. 401, VRDY OFF, even when the servo amplifier ready signal for the corresponding axis goes off. The servo ready signal SA, however, is set to 0. The SA signal, however, can remain set to 1 depending on the setting of SAK, bit 6 of parameter No. 1804.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066								IGNVRY
G192	IGVRY8	IGVRY7	IGVRY6	IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1804		SAK						

[Data type] Bit

SAK When the VRDY OFF alarm ignore signal IGNVRY is 1, or when the VRDY OFF alarm ignore signals IGVRY1 to IGVRY8 are 1:

0 : Servo ready signal SA is set to 0.
1 : Servo ready signal SA remains set to 1.

Alarm and Message

Number	Message	Description
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–8) servo amplifier READY signal (DRDY) went off.

Caution

CAUTION

1 When the control enters NOT READY status due to emergency stop or a servo alarm and then the control is reset, reset processing is not terminated until the VRDY OFF alarm ignore signal is set to 0.

2 When the VRDY OFF alarm ignore signal is set to 1 and the servo amplifier ready signal is set to off, the motor is freed from the drive, but follow up is not performed. To perform follow up, set the servo off signal to 1.

Note

NOTE

While the VRDY OFF alarm ignore signal is set to 1, and a servo alarm other than alarm No. 401 occurs, the control unit detects the alarm.

2.10 ABNORMAL LOAD DETECTION

General

Machine collision, defective, and damaged cutters cause a large load torque on the servo and spindle motors, compared with normal rapid traverse or cutting feed. This function detects a load torque on the motors and sends it as an estimated load torque to the PMC via the CNC. If the detected load is abnormally great compared with the value specified in the parameter, the function stops the servo motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize a possible damage to the machine. (The function to reverse motors is effective only for servo motors.)

The abnormal load detection function is further divided as follows:

(1) Estimated load torque output function

The CNC is always calculating the estimated load torque for the motor (excluding acceleration/deceleration torque). The estimated load torque output function enables the PMC to read the calculated torque using the window function.

(2) Abnormal load detection alarm function

This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, if the load torque obtained by the estimated load torque output function is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)

● Parameter setting

The following flowcharts explain how to specify parameters for the abnormal load detection function.

(1) Servo axis

Abnormal load detection function is available.

↓

Abnormal load detection function to be used?

- No → No. 2016#0 = 0
- Yes ↓

No. 2016#0 = 1, No. 2104 = 0

↓

Abnormal load detection alarm function to be used?

- No → Only the estimated load torque output function is enabled.
- Yes ↓

Set parameter bit 5 of No. 2015 to 1 to observe the load torque.

↓ <Adjustment>

Set parameter Nos. 1880, 2050, 2051, 2103, and 2104.

↓

Reset parameter bit 5 of parameter No. 2015 to 0.

↓

The estimated load torque output and abnormal load detection functions are enabled.

(2) Spindle

Abnormal load detection function is available.

↓

Specify parameter Nos. 4247, 4248, 4249, and 4250, and reset parameter No. 4341 to 0.

↓

Abnormal load detection alarm function to be used?

- No → Only the estimated load torque output function is enabled.
- Yes ↓

Specify parameter No. 4341.

↓

The estimated load torque output and abnormal load detection functions are enabled.

Signal

Servo axis abnormal load detected signal ABTQSV <F090#0>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on a servo axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for a servo axis, Cs axis, spindle positioning axis, or spindle axis during rigid tapping.

First–spindle abnormal load detected signal ABTSP1 <F090#1>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the first axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for the first spindle under speed control.

Second–spindle abnormal load detected signal ABTSP2 <F090#2>

[Classification] Output signal

[Function] Informs the PMC that an abnormal load was detected on the second axis.

[Output condition] This signal becomes “1” if:

- An abnormal load is detected for the second spindle under speed control.

The following list summarizes the alarms and signals output by each function.

	Signal output		Alarm	
	ABTQSV	ABTSP1/ ABTSP2	409	754/764
Servo axis	○	—	○	—
Cs contour control	○	—	○	—
Spindle positioning axis	○	—	○	—
Rigid tapping	○	—	—	○
Spindle axis for speed control	—	○	—	○

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F0090						ABTSP2	ABTSP1	ABTQSV

Parameter

(1) Parameter common to servo axes and spindles

1880	Timer for abnormal load detection alarm

[Data type] Word

[Unit of data] msec

[Valid data range] 0 to 32767
(If 0 is set, 200 ms is assumed.)

This parameter specifies the interval between the detection of an abnormal load and the issue of a servo alarm. When the set value is not a multiple of eight, it is rounded up to the nearest multiple of eight.

[Example] When 30 is set, the system assumes 32 ms.

(2) Servo axis parameters

	#7	#6	#5	#4	#3	#2	#1	#0
2015			TDOUT					

[Data type] Bit axis

TDOUT Output to the check board for each axis
0 : Output the torque command to the check board.
1 : Output the estimated load torque to the check board.

	#7	#6	#5	#4	#3	#2	#1	#0
2016								ABNTDT

[Data type] Bit axis

ABNTDT Output of the estimated load torque for each axis
0 : Disabled
1 : Enabled

This parameter must be specified when using the estimated load torque output function or abnormal load detection alarm function.

2050	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3559

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 956 in this parameter.

2051	Velocity control observer

[Data type] Word axis

[Valid data range] 0 to 32767

[Setting value] 3329

When using the velocity loop observer (by setting bit 2 of parameter No. 2003 to 1), set 510 in this parameter.

2103	Retraction distance upon the detection of an abnormal load

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter specifies the amount by which the tool is retracted, by reversing the motor, if an abnormal load is detected. When the motor is rotating at low speed, however, the tool may be retracted too far. To prevent this, the motor is stopped, instead of being reversed, upon the detection of an abnormal load while the specified feedrate is less than the value listed in the table below.

When this parameter is set to value A, the detection of an abnormal load causes the tool to be retracted in the reverse direction by an amount A, then stop , if the specified feedrate is equal to or greater than the value listed below, for each detection unit.

Detection unit	Feedrate
1μ 0.1μ	A / 8 mm/ min A / 80 mm/ min

When this parameter is set to 0, the motor stops immediately upon the detection of abnormal load.

2104	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] Torque command unit (Refer to the digital servo operator's manual for details.)

[Valid data range] 0 to 7282
(The maximum motor torque is 7282, regardless of the motor type.)

This parameter specifies the threshold load torque at which an abnormal load detection alarm is issued.

Monitor the load torque by setting bit 5 of parameter No. 2015 to 1 then, for this parameter (No. 2104), set a value larger than the maximum monitored torque. An output of 4.4 V is equivalent to 7282 in the units of this parameter.

(3) Spindle parameters

	#7	#6	#5	#4	#3	#2	#1	#0
4015							SPLDMT	

[Data type] Bit axis

SPLDMT Spindle load torque monitor function

0 : The spindle load torque monitor function is disabled.
1 : The spindle load torque monitor function is enabled.

4247	Magnetic flux compensation time constant for spindle load torque monitor

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 8192

[Standard setting] Depends of the motor model.

This parameter is used to compensate the delay in the generation of magnetic flux in the spindle motor relative to the specified value. When 0 is set, it is assumed that the generation of magnetic flux is not delayed.

4248	Spindle load torque monitor constant

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] Depends of the motor model.

This constant is determined by the maximum output torque and inertia of the motor. It is used for observer processing.

4249	Observer gain 1 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4250	Observer gain 2 for spindle load torque monitor

[Data type] Word axis

[Valid data range] 0 to 32767

[Standard setting] 500

4341	Threshold for abnormal load detection alarm

[Data type] Word axis

[Unit of data] 0.01 %

[Valid data range] 0 to 10000

This parameter specifies the threshold load torque at which an abnormal load detection alarm is issued for the spindle. Set a percentage (in units of 0.01 %) for the maximum output torque of the motor. When 0 is set, no abnormal load detection alarm is issued for the spindle.

Alarm and message

(1) Servo axis

Number	Message	Description
409	Servo alarm: Abnormal load detected on axis n	An abnormal load was detected on a servo motor, or on a spindle motor during Cs mode. To release the alarm, use RESET.

(2) Spindle

Number	Message	Description
754	Abnormal load detected on the first spindle	An abnormal load was detected on the first spindle motor. To release the alarm, use RESET.
764	Abnormal load detected on the second spindle	An abnormal load was detected on the second spindle motor. To release the alarm, use RESET.

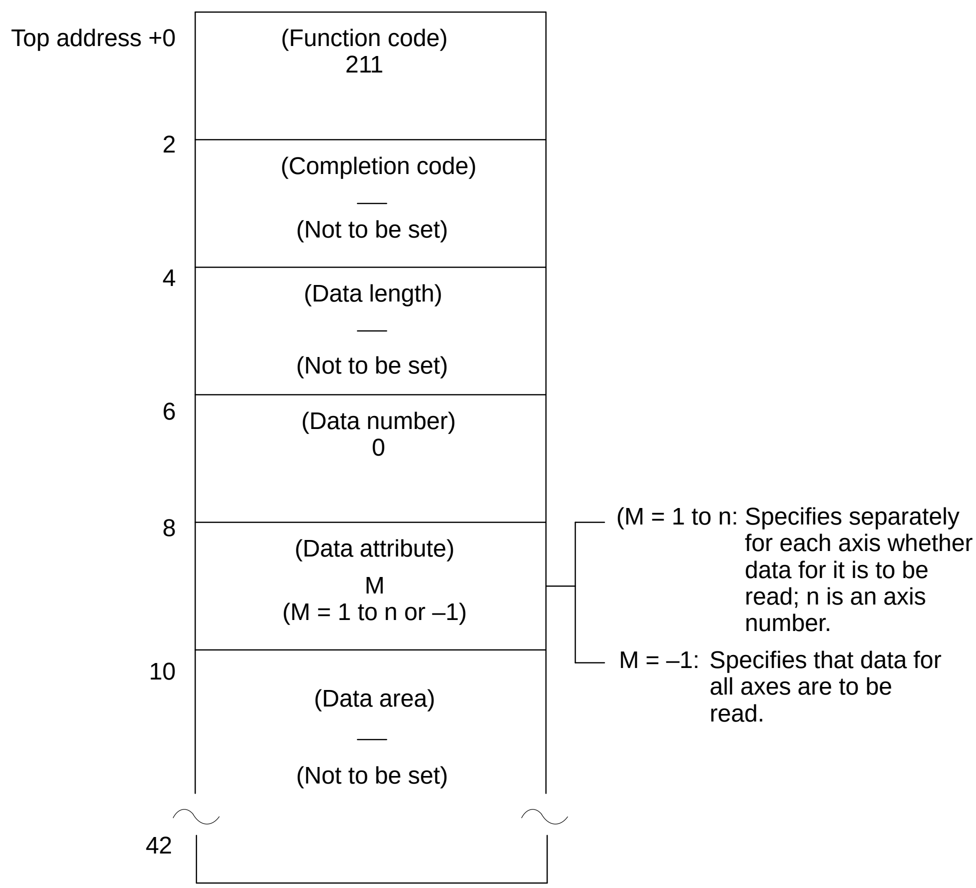
PMC window function

- **Reading the load torque data**

The load torque data can be read at the PMC using its window function.

(1) Servo axis

[Input data structure]



[Completion code]

0: The load torque data was read normally.

4: Incorrect data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than the number of controllable axes was specified.

[Output data structure]

Top address +	Contents	Description of value
0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 0	
8	(Data attribute) M (M: Data at input)	
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are four controllable axes

		Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	
14	Load torque for the third axis (2 bytes)	
16	Load torque for the fourth axis (2 bytes)	

(2) Spindle

[Input data structure]

Address	Content	
Top address +0	(Function code) 211	
2	(Completion code) — (Not to be set)	
4	(Data length) — (Not to be set)	
6	(Data number) 1	
8	(Data attribute) M (M = 1 to n or –1)	(M = 1 to n: Specifies separately for each axis whether data for it is to be read; n is an axis number. M = –1: Specifies that data for all axes are to be read.
10	(Data area) — (Not to be set)	
42		

[Completion code]

0: The load torque data was read normally.

4: Incorrect data was specified as a data attribute, that is a value other than –1 or 1 to n (number of axes) was specified. Alternatively, a value greater than controllable axes was specified.

[Output data structure]

Top address +		
0	(Function code) 211	
2	(Completion code) ? (Refer to the above description about the completion code.)	
4	(Data length) L (L = 2*n, where n is the number of specified axes)	
6	(Data number) 1	
8	(Data attribute) M (M: Data at input)	Description of value
10	Load torque for the specified axis (2 bytes)	Signed binary format (negative value in two's complement)

Or if there are two controllable axes

		Description of value
10	Load torque for the first axis (2 bytes)	Signed binary format (negative value in two's complement)
12	Load torque for the second axis (2 bytes)	

2.11 SERVO/SPINDLE MOTOR SPEED DETECTION

General

The servo axis and spindle motor speeds are monitored. If the speed of an axis exceeds a preset maximum (specified by parameter setting), the corresponding signal is output to a Y address (specified by parameter setting) of the PMC.

(1) Setting a Y address for signal output

In parameter No. 1891, set a Y byte address to which signals are to be output.

(2) Setting a maximum speed

Set the maximum speed for each servo–motor–controlled axis in parameter No. 1890.

Set the maximum speed for the axis controlled by the first serial spindle motor in S1 of parameter No. 4345.

Set the maximum speed for the axis controlled by the second serial spindle motor in S2 of parameter No. 4345.

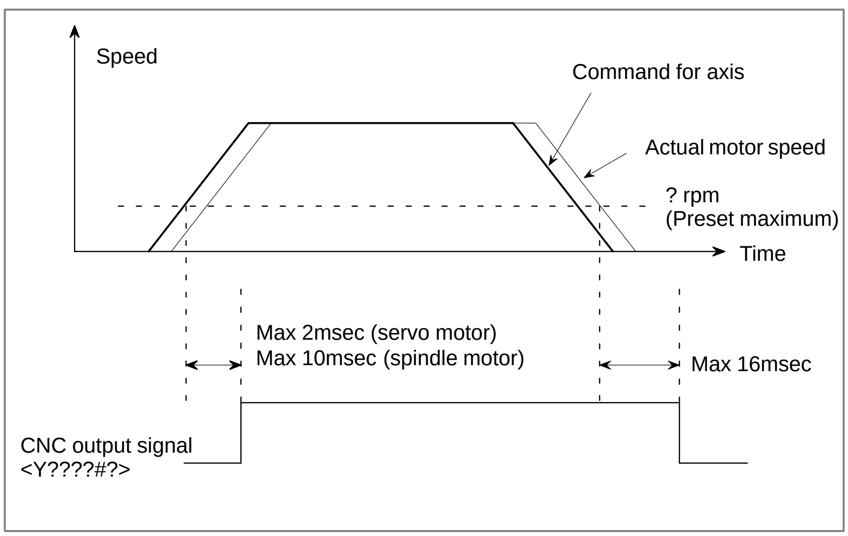
When 0 is specified as the maximum speed for an axis, the speed of that axis is not monitored.

(3) Signal input

Set the motor speed detection function enable signal MSDFON to 1. When the power is turned on, the CNC checks the status of this signal. If the signal is 1, the CNC enables the motor speed detection function.

When MSDFON is 1, and a motor speed exceeds the preset maximum, the corresponding bit of the set Y address is set to 1.

The following diagram illustrates the signal output timing.



NOTE

The status of each signal is updated every 8 msec. (Fluctuations in real speed of less than 8 msec duration cannot be detected.)

Signal

Motor speed detection function enable signal MSDFON <G016#0>

[Classification] Input signal

[Function] Enables the motor speed detection function.

[Operation] When this signal is 1, the motor speed detection function is enabled.

The servo/spindle motor speed detection function allows the CNC to output a detected result to the Y addresses directly. Normally, only the PMC can write to the Y addresses. When this function is used, however, the CNC can also write to the Y addresses. Therefore, care must be taken not to write to the Y addresses from both the CNC and PMC.

When this signal is 1, the CNC enables this function, assuming that the PMC ladder does not use the address set in parameter No. 1891 and address (the setting + 1).

When the signal is 0, the CNC disables this function, assuming that the PMC ladder is using these Y addresses.

Therefore, to use the servo/spindle motor speed detection function, add the processing for setting the motor speed detection function enable signal to 1 to the PMC ladder. When doing so, care must be taken to ensure that the PMC ladder does not use the address set in parameter No. 1891 and address (the setting + 1), as the Y address.

CAUTION

1 Immediately after the PMC ladder starts operating, set the signal to 1.
2 When this signal is set to 0 after the CNC starts, the servo/spindle motor speed detection function is not disabled.

Servo motor speed detection signals DSV1 to DSV8 <Y(n + 0)>

(n is the value set in parameter No. 1891.)

[Classification] Output signal

[Function] Report the motor speed status of each of the axes controlled by servo motors.

[Operation] Each signal is set to 1 when:

· The servo motor speed exceeds the maximum speed set in parameter No. 1890.

Each signal is set to 0 when:

· The servo motor speed does not exceed the maximum speed set in parameter No. 1890.

Spindle motor speed detection signals DSP1, DSP2, DSP3 <Y(n + 1)#0 to #2>

(n is the value set in parameter No. 1891.)

[Classification] Output signal

[Function] Report the motor speed status of each of the axes controlled by spindle motors.

[Operation] Each signal is set to 1 when:

· The spindle motor speed exceeds the maximum speed set in parameter No. 4345.

Each signal is set to 0 when:

· The spindle motor speed does not exceed the maximum speed set in parameter No. 4345.

CAUTION

1 When two–path control is applied, the detection results for the axes in path 2 are output to the addresses set with parameter No. 1891 for path 2, in the same way as described above. Set the parameter for each path so that the difference between the setting made for path 1 and that made for path 2 is at least +2. This prevents the outputs from the paths from overlapping.

2 Some of the servo motor speed detection signals DSV1 to DSV8 and spindle motor speed detection signals DSP1, DSP2, and DSP3 may not function depending on the axis configuration. Even when some signals do not function, the addresses Y(n + 0) #7 to #0 and Y(n + 1) #1 and #0 must not be used by the PMC ladder.

3 When this function is used, Y(n + 1) #7 to #3 are reserved. These bits must not be used by the PMC ladder.

4 A spindle motor may be used for controlling apositioning axis, or a servo motor may be used for a spindle. This function performs motor–related detection.

Example: When Cs contour axis control is performed (the spindle and C–axis are controlled by the first serial spindle motor), the fourth axis is used as the C–axis.
In this case, if the preset maximum speed is exceeded, the following occurs regardless of whether spindle control or C–axis control is being performed:
DSP1 <Y(n + 1) #0, where n = setting in parameter No. 1891> is turned on.
DSV4 <Y(n + 0) #3, where n = setting in parameter No. 1891> is not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016								MSDFON

	#7	#6	#5	#4	#3	#2	#1	#0
Y (n+00	DSV8	DSV7	DSV6	DSV5	DSV4	DSV3	DSV2	DSV1
Y (n+1)						DSP3	DSV2	DSV1

Reserved (#7 to #3 of Y (n+1))

Parameter

1890	Servo motor speed for detection

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

[Data type] Word axis

[Unit of data] rpm

[Valid data range] 0 to 8000

The servo motor speed of each axis is monitored and a motor speed detection signal is output indicating whether the speed of each axis exceeds the value set in this parameter (set in the Y address specified in parameter No. 1891)

NOTE
No motor speed detection signals are output when the servo/spindle motor speed detection function is not used or 0 is set in this parameter.

1891	Initial value of the Y address where motor speed detection signals are output

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

[Data type] Word axis

[Valid data range] 0 to 126, 1000 to 1013, 1020 to 1033

This parameter specifies the Y address where motor speed detection signals are output.

The spindle motor speeds and servo motor speed of each axis are monitored and motor speed detection signals are output to the Y address specified in this parameter and (Y address +1) to indicate whether speeds exceed the values set in the parameters.

WARNING

1 Be sure to specify a Y address that is not used with a PMC sequence program (ladder).

2 When controlling two path lathe, ensure that the same value is not set for path 1 and path 2. (Set a separate address for path 1 and path 2.)

NOTE

No motor speed detection signals are output when the servo/spindle motor speed detection function is not used, the value 0 or a value beyond the allowable data range is specified in this parameter, or an input/output address specified within the allowable data range represents an address where no I/O device is mounted.

4345	Serial spinsle motor detection speed

[Data type] Word type

[Unit of data] rpm

[Valid data range] 0 to 32767

S1 : for First spindle / S2 : for Second spindle / S3 : for Third spindle

This parameter sets the serial spindle motor speed at which the motor speed detection signal is output. The speeds of the serial spindle motors for the first, second, and third spindles are monitored, and the motor speed detection signal, indicating whether the speed of each spindle exceeds the value set in this parameter, is output to the Y address specified with parameter No. 1891.

WARNING

For this parameter, set a motor speed rather than a spindle speed.

NOTE

The motor speed detection signals are not output when the servo/spindle motor speed detection function is not used, or 0 is set for this parameter.

Note

NOTE

1 Spindle motor speed detection is enabled only for serial spindles.
2 The relationship between servo motor speed detection signals DSV1 to DSV8 and the servo motors depends on the servo axis number (servo connector number) set in parameter No. 1023.
3 When two–path control is applied, commands can be switched between the paths by using composite control (T series) or S command selection.
  Even when such command switching is performed, the relationship between detection signals DSV1 to DSV8, DSP1, DSP2, and DSP3 and the motors remains as is. (The relationship is determined as described in Note 2 above.)

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 JAX (No.1002#0).

- **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 is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
With the rapid traverse selection switch the tool can be moved at the rapid traverse rate regardless of the jog feedrate 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, MJ</td><td>MD1, MD2, MD4, MINC</td></tr>
<tr><td>Selection of the axis to move</td><td colspan="2" rowspan="2">+J1, –J1, +J2, –J2, +J3, –J3, ...</td></tr>
<tr><td>Selection of the direction to move the axis</td></tr>
<tr><td>Selection of the move amount</td><td></td><td>MP1, MP2</td></tr>
<tr><td>Selection of feedrate</td><td colspan="2">*JV0 – *JV15, RT, ROV1, ROV2</td></tr>
</table>

The only difference between jog feed and incremental feed is the method of selecting the feed distance. In jog feed, the tool continues to be fed while the following signals selecting the feed axis and direction are “1”: +J1, –J1, +J2, –J2, +J3, –J3, etc. In incremental feed, the tool is fed by one step.
The distance of the step is selected by the manual handle feed move distance select 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 Section 7.1.7.1, “Feedrate Override Signals.”
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 – +J8<G100> –J1 – –J8<G102>

[Classification] Input signal

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

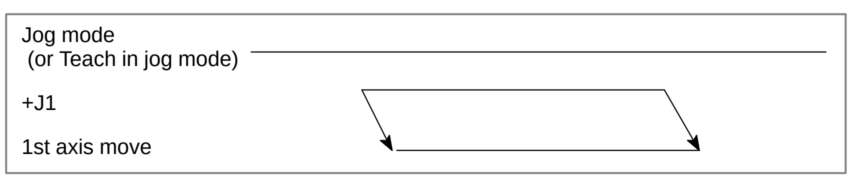
+ J 1

1 First axis is fed
2 Second axis is fed
3 Third axis is fed
: :
8 Eighth axis is fed

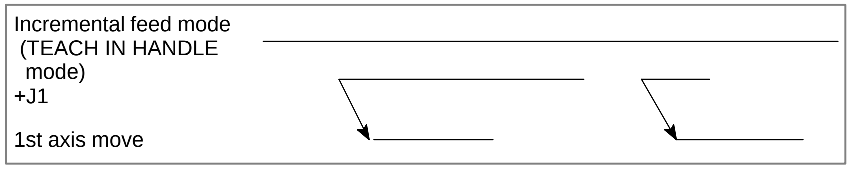
\+ Feed in positive direction
– Feed in negative direction

[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”.



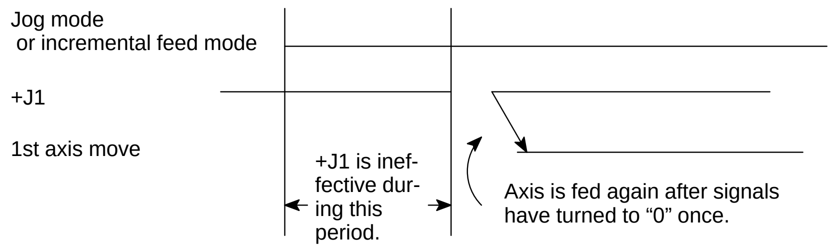
- In incremental feed, the control unit feeds the specified axis by the step distance which is specified by the manual handle feed move distance selection signal MP1, MP2. Then the control unit stops it. Even if the signal is set to “0” while the axis is being fed, the control unit does not stop feeding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



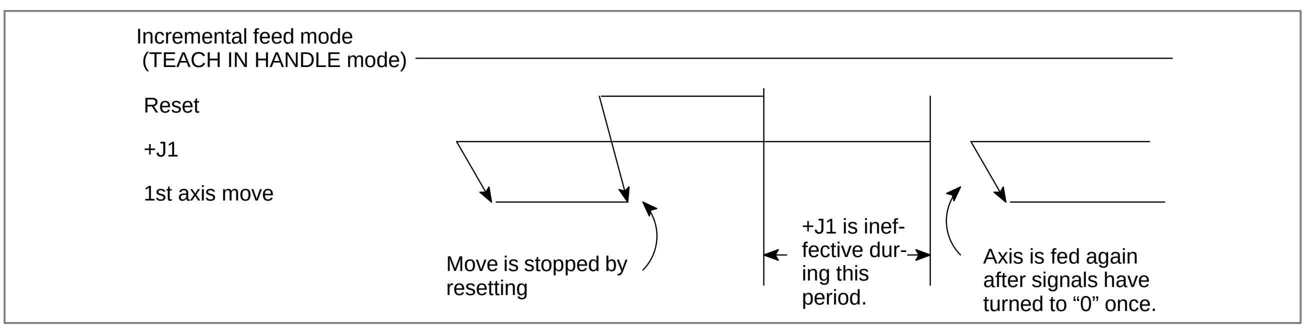
NOTE

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.



Manual Feedrate Override Signal *JV0 – *JV15<G011>

[Classification] Input signal

[Function] Selects a feedrate in jog feed or incremental feed. These signals are sixteen binary code signals, which correspond to the override values as follows:

$$\text{Override value (\%)} = 0.01\% \quad \times \sum_{i=0}^{15} | 2^{1} \times \text{Vi} |$$

where
Vi = 0 when the *JVi signal is “1”
Vi = 1 when the *JVi signal is “0”

The override value is assumed to be zero when all of the signals, (*JV0 to *JV15) are set to “1” or “0”. When this occurs, the feed is stopped. The override value can be specified in the range of 0% to 655.34% in units of 0.01%. Some examples are listed below.

*JV0 – *JV15				Override value (%)
12	8	4	0	
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 0	0.01
1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.10
1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.00
1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.00
1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.00
0 1 1 0	0 0 1 1	1 0 1 1	1 1 1 1	400.00
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 1	655.34
0 0 0 0	0 0 0 0	0 0 0 0	0 0 0 0	0

[Operation] If rapid traverse selection signal RT is “0” during jog feed or incremental feed, the manual feedrate specified by the parameter (no. 1423) is overridden by the value specified by the JVi signal.

NOTE
The JVi signals also serve as the override signals during dry run in automatic operation mode.

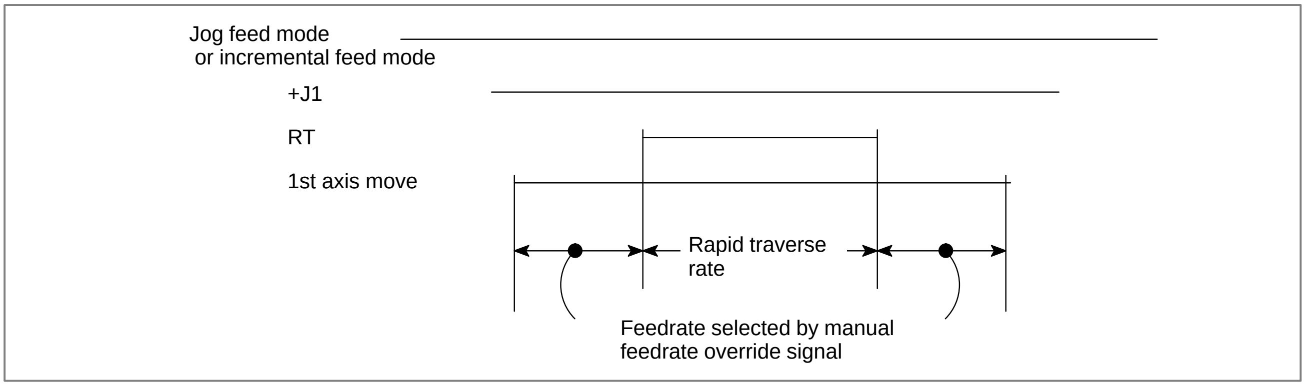
Manual rapid traverse selection signal RT<G019#7>

[Classification] Input signal

[Function] Selects a rapid traverse rate for jog feed or incremental feed.

[Operation] When the signal turns to “1”, the control unit operates as described below:

- The control unit executes the jog feed or incremental feed at a rapid traverse rate. The rapid traverse override is validated.
- When the signal is switched from “1” to “0” or vice versa during jog feed or incremental feed, the feedrate is decelerated until it reaches zero, then increased to the specified value. During acceleration and deceleration, the feed axis and direction selection signal can be kept “1”.



WARNING
After the power is turned on, the stroke limit function does not work until the reference position return is completed. During this period, the control unit ignores the RT signal, if it is set to "1", and keeps moving the tool at a feedrate selected by the manual feedrate override signal. A parameter RPD (No. 1401#0) can be specified so that the rapid traverse is validated before the reference position return is completed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G019	RT							
G100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1
G102	–J8	–J7	–J6	–J5	–J4	–J3	–J2	–J1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in jog feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1401								RPD

[Data type] Bit

RPD Manual rapid traverse during the period from power–on time to the completion of the reference position return.

0 : Disabled (Jog feed is performed.)
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				

[Data type] Bit

JRV Manual continuous feed (jog feed)

0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

NOTE
Specify a feedrate in parameter No. 1423.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Data type] Word axis

(1) In M series, or in T series when JRV, bit 4 of parameter No. 1402, is set to 0 (feed per minute), specify a jog feedrate at feed per minute with an override of 100%.

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	6 – 32767
Inch machine	0.1 inch/min	
Rotation axis	1 deg/min	

(2) When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a jog feedrate (feed per revolution) under an override of 100%.

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 – 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rate of manual rapid traverse when the rapid traverse override is 100% for each axis.

> **NOTE**
> If 0 is set, the rate set in parameter 1420 is assumed.

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				

[Data type] Bit axis

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0: Exponential acceleration/deceleration is applied.
1: Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1624	Time constant of exponential acceleration/deceleration or bell–shaped acceleration/deceleration or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] 1 msec

[Valid data range] 0 to 4000 (for exponential acceleration/deceleration)
0 to 512 (for linear acceleration/deceleration after interporation)

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration or linear acceleration/deceleration after interpolation in jog feed for each axis.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

Warning

> **WARNING**
> For incremental feeding along an axis under diameter programming, the tool moves in units of the diameter.

Note

NOTE

1 Time constant and method of automatic acceleration/deceleration for manual rapid traverse are the same as G00 in programmed command.
2 If a manual pulse generator is provided, the manual handle feed mode is enabled instead of incremental feed mode. However, using parameter JHD (bit 0 of parameter No. 7100) enables both manual handle and incremental feed in the manual handle feed mode.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.3.2	JOG FEED
		III.3.3	INCREMENTAL FEED
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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. 7113 and 7114).
The handle magnifications can be selected by the manual handle feed move distance selection signal.

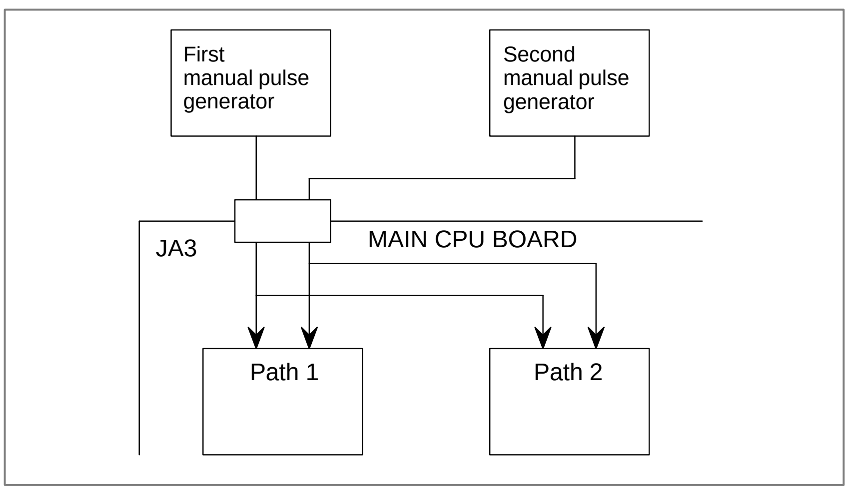
The number of manual pulse generators available depends on the type of an option used as listed below.

(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 path, eight bits are reserved as manual handle feed axis select signals.



- **Availability of manual handle feed in Jog mode**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual handle feed in TEACH IN JOG mode**

Parameter THD (bit 1 of No. 7100) enables or disables the manual handle feed generator in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter (No. 7117) specifies as follows:

SET VALUE 0: The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

Other than 0: The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG**

Parameter HNGx (No. 7102#0) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Signal

Manual Handle Feed Axis Selection Signals

- **(M series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>
  HS3A – HS3D
  <G019#0 – #3>

- **(T series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>

- **(Two–path control)**
  HS1A$^{\#1}$ – HS1D$^{\#1}$
  <G018#0 – #3>
  HS2A$^{\#1}$ – HS2D$^{\#1}$
  <G018#4 – #7>
  HS3A$^{\#1}$–HA3D$^{\#1}$ (M series)
  <G019#0–#3>
  HS1A$^{\#2}$ – HS1D$^{\#2}$
  <G1018#0 – #3>
  HS2A$^{\#2}$ – HS2D$^{\#2}$
  <G1018#4 – #7>
  HS3A$^{\#3}$–HS3D$^{\#2}$ (M series)
  <G1019#0–#3>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of four code signals, A, B, C, and D is provided for each manual pulse generator. (Up to three generators can be used.) (For two–path, these signals are provided for each manual pulse generator and each path.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS 1 A

1 First manual pulse generator
2 Second manual pulse generator
3 Third manual pulse generator (M series)

Code signals A, B, C, and D correspond to the feed axes as listed in the following table:

Manual handle feed axis selection				Feed axis
HSnD	HSnC	HSnB	HSnA	
0	0	0	0	No selection (None of axis is fed)
0	0	0	1	1st axis
0	0	1	0	2nd axis
0	0	1	1	3rd axis
0	1	0	0	4th axis
0	1	0	1	5th axis
0	1	1	0	6th axis
0	1	1	1	7th axis
1	0	0	0	8th axis

<Two–path control>

Manual handle feed axis selection				Feed axis
HSnD#1	HSnC#1	HSnB#1	HSnA#1	
0	0	0	0	No selection (no axis is used for path 1)
0	0	0	1	1st axis of path 1
0	0	1	0	2nd axis of path 1
0	0	1	1	3rd axis of path 1
0	1	0	0	4th axis of path 1
0	1	0	1	5th axis of path 1
0	1	1	0	6th axis of path 1
0	1	1	1	7th axis of path 1

Manual handle feed axis selection				Feed axis
HSnD#2	HSnC#2	HSnB#2	HSnA#2	
0	0	0	0	No selection (no axis is used for path 2)
0	0	0	1	1st axis of path 2
0	0	1	0	2nd axis of path 2
0	0	1	1	3rd axis of path 2
0	1	0	0	4th axis of path 2
0	1	0	1	5th axis of path 2
0	1	1	0	6th axis of path 2
0	1	1	1	7th axis of path 2

Manual Handle Feed Amount Selection Signal MP1, MP2<G019#4, 5> (Incremental Feed Signal)

[Classification] Input signal

[Function] This signal selects the distance traveled per pulse from the manual pulse generator during the manual handle feed or manual handle interrupt. It also selects the distance traveled per incremental feed step.

The table below lists the signal–to–distance correspondence.

Travel distance select signal for manual handle feed		Distance traveled		
MP2	MP1	Manual handle feed	Manual handle interrupt	Incremental feed
0	0	Least input increment × 1	Least command increment × 1	Least input increment × 1
0	1	Least input increment × 10	Least command increment × 10	Least input increment × 10
1	0	Least input increment × m*1	Least command increment × m*1	Least input increment × 100
1	1	Least input increment × n*1	Least command increment × n*1	Least input increment × 1000

*1 Scale factors m and n are specified using parameter Nos. 7113 and 7114.

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 interrupts, and Section 3.1, “Jog Feed/Incremental Feed” for incremental feed.

Signal address

<For 1–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019			MP2	MP1	HS3D	HS3C	HS3B	HS3A

<For 2–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D #1	HS2C #1	HS2B #1	HS2A #1	HS1D #1	HS1C #1	HS1B #1	HS1A #1
G019			MP2#1	MP1#1	HS3D#1	HS3C#1	HS3B#1	HS3A#1
G1018	HS2D #2	HS2C #2	HS2B #2	HS2A #2	HS1D #2	HS1C #2	HS1B #2	HS1A #2
G1019			MP2#2	MP1#2	HS3D#2	HS3C#2	HS3B#2	HS3A#2

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF			THD	JHD

[Data type] Bit

JHD Manual handle feed in JOG mode or incremental feed in the manual handle feed

0: Invalid
1: Valid

THD Manual pulse generator in TEACH IN JOG mode

0: Invalid
1: Valid

HPF When a manual handle feed exceeding the rapid traverse rate is issued,

0: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are ignored. (The graduations of the manual pulse generator may not agree with the distance the machine has traveled.)
1: The rate is clamped at the rapid traverse rate, and the handle pulses corresponding to the excess are not ignored, but stored in the CNC. (If the rotation of the manual pulse generator is stopped, the machine moves by the distance corresponding to the pulses preserved in the CNC, then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Data type] Bit axis

HNGx Axis movement direction for rotation direction of manual pulse generator

0: Same in direction
1: Reverse in direction

7110	Number of manual pulse generators used

[Data type] Byte

[Valid data range] 1, 2, or 3

This parameter sets the number of manual pulse generators.

7113	Manual handle feed magnification m

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 127

This parameter sets the magnification when manual handle feed movement selection signal MP2 is on.

7114	Manual handle feed magnification n

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 1000

This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are “1”.

7117	Allowable number of pulses that can be accumulated during manual handle feed

[Data type] 2–word

[Unit of data] Pulses

[Valid data range] 0 to 99999999

If manual handle feed is specified such that the rapid traverse rate will be exceeded momentarily, those pulses received from the manual pulse generator that exceed the rapid traverse rate are accumulated rather than canceled. This parameter sets the maximum number of pulses which can be accumulated in such a case.

Warning

WARNING

Rotating the handle quickly with a large magnification such as x100 moves the tool too fast or the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator. The feedrate is clamped at the rapid traverse rate.

Caution

CAUTION
Rotate the manual pulse generator at a rate of five rotations per second or lower.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.4	MANUAL HANDLE FEED
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.3.4	MANUAL HANDLE FEED
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.3.4	MANUAL HANDLE FEED
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.3.4	MANUAL HANDLE FEED

3.3 MANUAL HANDLE INTERRUPTION

General

Rotating the manual pulse generator during automatic operation can increase the distance traveled by the amount corresponding to the handle feed. The axis to which the handle interrupt is applied is selected using the manual handle interrupt axis select signal.
The minimum travel distance per graduation is the least command increment. The minimum travel distance can be increased by tenfold or by two scale factors (parameter Nos. 7113 and 7114). Each scale factor can be selected using the manual handle travel distance select signal (Section 3.2, ”Manual Handle Feed”).

Signal

Manual Handle Interrupt Axis Selection Signal

- **(M series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>
  HS3IA – HS3ID
  <G042#0 – #3>

- **(T series)**
  HS1IA – HS1ID
  <G041#0 – #3>
  HS2IA – HS2ID
  <G041#4 – #7>

- **(Two–path control)**
  HS1IA$^{\#1}$ – HS1ID$^{\#1}$
  <G041#0 – #3>
  HS2IA$^{\#1}$ – HS2ID$^{\#1}$
  <G041#4 – #7>
  HS3IA$^{\#1}$–HS3ID$^{\#1}$ (M series)
  <G042#0–#3>
  HS1IA$^{\#2}$ – HS1ID$^{\#2}$
  <G1041#0 – #3>
  HS2IA$^{\#2}$ – HS2ID$^{\#2}$
  <G1041#4 – #7>
  HS3IA$^{\#2}$–HS3ID$^{\#2}$ (M series)
  <G1042#0–#3>

[Classification] Input signal

[Function] These signals select an axis to which the manual handle interrupt is applied. There are three sets of signals, each corresponding to a manual pulse generator (up to three). Each set consists of four code signals A, B, C, and D. (For the T series (two–path control), each manual pulse generator has one set of signals for each tool post.) The number in each signal name corresponds to the number (position) of the manual pulse generator.

HS 1 IA

- 1 Selects the axis for which manual pulse generator No. 1 is used
- 2 Selects the axis for which manual pulse generator No. 2 is used
- 3 Selects the axis for which manual pulse generator No. 3 is used (M series)

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

Signal address

<For 1–path control >

	#7	#6	#5	#4	#3	#2	#1	#0
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042					HS3ID	HS3IC	HS3IB	HS3IA

<For 2–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G041	HS2ID #1	HS2IC #1	HS2IB #1	HS2IA #1	HS1ID #1	HS1IC #1	HS1IB #1	HS1IA #1
G042					HS3ID	HS3IC	HS3IB	HS3IA
G1041	HS2ID #2	HS2IC #2	HS2IB #2	HS2IA #2	HS1ID #2	HS1IC #2	HS1IB #2	HS1IA #2
G1042					HS3ID	HS3IC	HS3IB	HS3IA

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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.4.8	MANUAL HANDLE INTERRUPTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.4.6	MANUAL HANDLE INTERRUPTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.4.7	MANUAL HANDLE INTERRUPTION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.4.6	MANUAL HANDLE INTERRUPTION

3.4 TOOL AXIS DIRECTION HANDLE FEED FUNCTION/ TOOL AXIS DIRECTION HANDLE FEED FUNCTION B

The tool axis direction handle feed function allows the tool to be moved through a specified distance by handle feed in the axis direction of the tool, tilted by rotating the rotation axes.
Tool axis direction handle feed function B provides two functions: tool axis direction handle feed and tool axis perpendicular direction handle feed, which performs handle feed in the direction perpendicular to the tool axis.

3.4.1 Tool Axis Direction Handle Feed Function

General

By using the tool axis direction handle feed function, the tool can be moved in the axis direction of the tool by an amount equal to the manual pulse generator rotation.

Tool axis direction handle feed is enabled when the following four conditions are satisfied:

(1) Handle mode is selected.

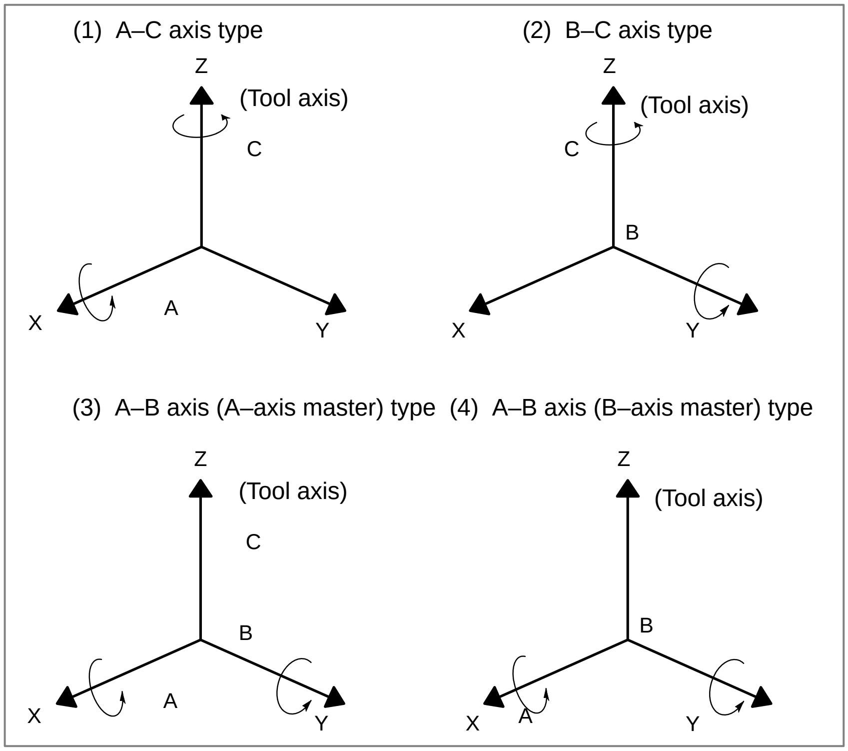
(2) The tool axis direction handle feed mode signal is 1.

> **NOTE**
> When both the tool axis direction handle feed mode signal and tool axis perpendicular direction handle feed mode signal are 1, neither mode is enabled. In this case, normal handle mode is set.

(3) In parameter No. 7121, the axis number for the first manual pulse generator is set as the tool axis direction handle feed mode axis.

(4) A manual handle feed axis is selected for the axis set in parameter No. 7121.

Assume that the rotation axes for basic axes X, Y, and Z are A, B, and C, respectively. Assume also that the Z–axis represents the tool axis in the machine coordinate system. Then, depending on the axis configuration of the machine, four tool axis directions are available. Specify the desired type with parameter No. 7120.



Output pulse (Hp) distribution by the manual pulse generator to the X–axis, Y–axis, and Z–axis for the four types is expressed below.

(1) A–C axis type

$Xp = Hp \times \sin(a) \times \sin(c)$

$Yp = -Hp \times \sin(a) \times \cos(c)$

$Zp = Hp \times \cos(a)$

(2) B–C axis type

$Xp = Hp \times \sin(b) \times \cos(c)$

$Yp = Hp \times \sin(b) \times \sin(c)$

$Zp = Hp \times \cos(b)$

(3) A–B axis type (A–axis master)

$Xp = Hp \times \sin(b)$

$Yp = -Hp \times \cos(b) \times \sin(a)$

$Zp = Hp \times \cos(b) \times \cos(a)$

(4) A–B axis type (B–axis master)

$Xp = Hp \times \cos(a) \times \sin(b)$

$Yp = -Hp \times \sin(a)$

$Zp = Hp \times \cos(a) \times \cos(b)$

In the above expressions, a, b, and c represent the positions (angles) of the A–axis, B–axis, and C–axis relative to the machine zero point; those values that are present when tool axis direction handle feed mode is set, or when a reset occurs, are used. To change the feed direction, reenter tool axis direction handle feed mode, or press the reset key.

For tool axis direction handle feed B, the coordinates (angular displacements) of the rotation axes that determine the direction of the tool axis can be set. These coordinates are set using bits 3 and 4 (3D1X and 3D2X) of parameter No. 7104, and parameter Nos. 7144 and 7145.

Signal

Tool axis direction handle feed mode signal ALNGH <G023#7>

[Classification] Input signal

[Function] This signal selects tool axis direction handle feed mode. When the following conditions are all satisfied, tool axis direction handle feed mode is set:

1. This signal is 1.
2. The value of the manual handle feed axis selection signal for the first manual pulse generator matches the value set in parameter No. 7121.
3. Handle mode is set.

NOTE
When both tool axis direction handle feed mode signal ALNGH and tool axis perpendicular direction handle feed mode signal RGHTH are set to 1, neither mode is set.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G023	ALNGH							

Parameter

- **Settings for tool axis direction handle feed**

	#7	#6	#5	#4	#3	#2	#1	#0
7104				3D2	3D1	CXC		TLX

[Data type] Bit

TLX When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, this parameter selects the tool axis direction when the rotation axes for the three basic axes in the basic coordinate system are positioned to the machine zero point:

0 : Z–axis direction

1 : X–axis direction

CXC Tool axis direction handle feed or tool axis perpendicular direction handle feed is performed with:

0 : 5–axis machine.
1 : 4–axis machine.

3D1 When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, the coordinates of the first rotation axis are:

0: The machine coordinates when the tool axis direction handle feed mode or tool axis perpendicular direction handle feed mode is entered, or upon a reset.
1: The value set in parameter No. 7144.

3D2 When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, the coordinates of the second rotation axis are:

0: The machine coordinates when the tool axis direction handle feed mode or tool axis perpendicular direction handle feed mode is entered, or upon a reset.
1: The value set in parameter No. 7145.

- **Axis configuration for using the tool axis direction handle feed function**

7120	Axis configuration for using the tool axis direction handle feed or tool axis perpendicular direction handle feed function

[Data type] Byte

[Valid data range] 1 to 4

When using the tool axis direction handle feed or tool axis perpendicular direction handle feed function, suppose that the rotation axes for the three basic axes X, Y, and Z in the basic coordinate system are axes A, B, and C, respectively. Suppose also that the Z–axis represents the tool axis direction when the rotation axes are positioned to the machine zero point. Then, depending on the axis configuration of the machine, all four of the following types are available. When a 4–axis machine is used, and when the tool axis perpendicular direction handle feed function is used, however, only types (1) and (2) are available.

(1) A–C axis type

(2) B–C axis type

(3) A–B axis (A–axis master) type

(4) A–B axis (B–axis master) type

This parameter selects a type. Values of 1 to 4 are assigned to these types, in order, from top to bottom. When the X–axis represents the tool axis direction, the above types are changed to B–A axis type, C–A axis type, B–C axis (B–axis master) type, and B–C axis (C–axis master) type.

- **Axis selection in the tool axis direction handle feed mode**

7121	Axis selection in tool axis direction handle feed mode

[Data type] Byte

[Valid data range] 1 to number of controlled axes

This parameter sets an axis number for the manual handle feed axis selection signal for the first manual pulse generator to enable tool axis direction handle feed mode. When the value set in this parameter matches the value of the manual handle feed axis selection signal, tool axis direction handle feed mode is enabled.

- **Rotation axis for using the tool axis direction handle feed function**

7144	Coordinates of the first rotation axis for using the tool axis direction handle feed or tool axis perpendicular direction handle feed function
7145	Coordinates of the second rotation axis for using the tool axis direction handle feed or tool axis perpendicular direction handle feed function

[Data type] Two–word

[Unit of data] 0.001 degree

[Valid data range] –360000 to 360000

When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, and 3D1 and 3D2 (bits 3 and 4 of parameter No. 7104) are set to 1, parameter Nos. 7144 and 7145 are set the coordinates (angular displacements) of the first and second rotation axes, respectively. If CXC (bit 2 of parameter No. 7104) is set to 1, however, the coordinates of the second rotation axis is assumed to be 0 regardless of the settings of 3D2 and parameter No. 7145.

Alarm and message

No.	Message	Meaning
5015	* No message appears.	In tool axis direction handle feed or tool axis perpendicular direction handle feed mode, a specified rotation axis does not exist.

Note

NOTE

1 The basic axes X, Y, and Z are determined by parameter No. 1022 (plane selection). The rotation axes A, B, and C are determined by parameter No. 1020 (axis name).

2 If one of the two axes specified by a type set depending on the axis configuration does not exist, alarm P/S 5015 is issued.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.6	Tool axis direction handle feed / Tool axis direction handle feed B

3.4.2 Tool Axis Perpendicular Direction Handle Feed Function

General

This function moves the tool by an amount corresponding to the rotation of the manual pulse generator handle, in the direction specified relative to the tool axis, tilted due to movement of the rotation axis.

Tool axis perpendicular direction handle feed is enabled when the following four conditions are satisfied:

(1) Handle mode is selected.

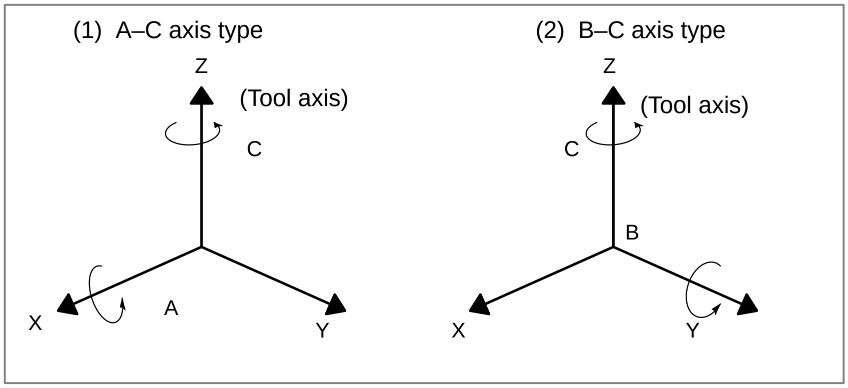
(2) The tool axis perpendicular direction handle feed mode signal is 1.

> **NOTE**
> When both the tool axis direction handle feed mode signal and tool axis perpendicular direction handle feed mode signal are 1, neither mode is enabled. In this case, normal handle mode is set.

(3) In parameter Nos. 7141 and 7142, the axis number for the first manual pulse generator is set as the tool axis perpendicular direction handle feed mode axis.

(4) A manual handle feed axis is selected for the axis set in parameter Nos. 7141 and 7142.

Assume that the rotation axes for basic axes X, Y, and Z are A, B, and C, respectively. When the direction of the tool axis corresponds to the direction of the Z–axis in the machine coordinate system, either rotation axis A or B is used with the tool axis, depending on the machine axis configuration type. For each type, feed in the X–axis direction and that in the T–axis direction are defined as described below. Specify the desired type with parameter No. 7120.



Output pulse (Hp) distribution by the manual pulse generator to the X–axis, Y–axis, and Z–axis for the four types is expressed below.

(1) A–C axis type (X axis direction)

$Xp = Hp \times \cos(c)$

$Yp = -Hp \times \sin(c)$

$Zp = \phi$

(2) A–C axis type (Y axis direction)

$Xp = -Hp \times \cos(a) \times \sin(c)$

$Yp = Hp \times \cos(a) \times \cos(c)$

$Zp = Hp \times \sin(a)$

(3) B–C axis type (X axis direction)

$Xp = Hp \times \cos(b) \times \cos(c)$

$Yp = Hp \times \cos(b) \times \sin(c)$

$Zp = -Hp \times \sin(b)$

(4) B–C axis type (Y axis direction)

$Xp = -Hp \times \sin(c)$

$Yp = Hp \times \cos(c)$

$Zp = \phi$

In the above expressions, a, b, and c represent the positions (angles) of the A–axis, B–axis, and C–axis relative to the machine zero point; those values that are present when tool axis direction handle feed mode is set, or when a reset occurs, are used. To change the feed direction, reenter tool axis direction handle feed mode, or press the reset key.

The coordinates (angular displacement) of the rotation axis, required to determine the direction of the tool axis, can be specified by setting bits 3D1 and 3D2 (bits 3 and 4 of parameter No. 7104) and parameters No. 7144 and 7145.

Signal

Tool axis perpendicular direction handle feed mode signal RGHTH <G023#6>

[Classification] Input signal

[Function] This signal selects tool axis perpendicular direction handle feed mode. When the following conditions are all satisfied, tool axis direction handle feed mode is set:

1. This signal is 1.
2. The value of the manual handle feed axis selection signal for the first manual pulse generator matches the value set in parameter Nos. 7141 and 7142.
3. Handle mode is set.
4. The value of the axis configuration type specified in parameter No. 7120 is 1 or 2.

> **NOTE**
> When tool axis direction handle feed mode signal ALNGH and tool axis perpendicular direction handle feed mode signal RGHTH are both set to 1, neither mode is set.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G023		RGHTH						

Parameter

- **Settings for tool axis perpendicular direction handle feed**

	#7	#6	#5	#4	#3	#2	#1	#0
7104				3D2	3D1	CXC		TLX

[Data type] Bit

TLX When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, this parameter selects the tool axis direction when the rotation axes for the three basic axes in the basic coordinate system are positioned to the machine zero point:

0 : Z–axis direction
1 : X–axis direction

CXC Tool axis direction handle feed or tool axis perpendicular direction handle feed is performed with:

0 : 5–axis machine.
1 : 4–axis machine.

3D1 When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, the coordinates of the first rotation axis are:

0: The machine coordinates when the tool axis direction handle feed mode or tool axis perpendicular direction handle feed mode is entered, or upon a reset.
1: The value set in parameter No. 7144.

3D2 When the tool axis direction handle feed or tool axis perpendicular direction handle feed function is used, the coordinates of the second rotation axis are:

0: The machine coordinates when the tool axis direction handle feed mode or tool axis perpendicular direction handle feed mode is entered, or upon a reset.
1: The value set in parameter No. 7145.

- **Axis configuration for using the tool axis perpendicular direction handle feed function**

7120	Axis configuration for using the tool axis direction handle feed or tool axis perpendicular direction handle feed function

[Data type] Byte

[Valid data range] 1 to 4

When using the tool axis direction handle feed or tool axis perpendicular direction handle feed function, suppose that the rotation axes for the three basic axes X, Y, and Z in the basic coordinate system are axes A, B, and C, respectively. Suppose also that the Z–axis represents the tool axis direction when the rotation axes are positioned to the machine zero point. Then, depending on the axis configuration of the machine, all four of the following types are available. When a 4–axis machine is used, and when the tool axis perpendicular direction handle feed function is used, however, only types (1) and (2) are available.

(1) A–C axis type

(2) B–C axis type

(3) A–B axis (A–axis master) type

(4) A–B axis (B–axis master) type

This parameter selects a type. Values of 1 to 4 are assigned to these types, in order, from top to bottom. When the X–axis represents the tool axis direction, the above types are changed to B–A axis type, C–A axis type, B–C axis (B–axis master) type, and B–C axis (C–axis master) type.

- **Axis selection setting in handle feed mode, in a direction perpendicular to the tool axis**

7141	Direction of the X–axis in handle feed mode, in a direction perpendicular to the tool axis
7142	Direction of the Y–axis in handle feed mode, in a direction perpendicular to the tool axis

[Data type] Byte

[Valid data range] 1 to 8

Specify the status of the axis selection signal of the first manual pulse generator to enable handle feed mode in a direction perpendicular to the tool axis. When these parameter settings correspond to the manual handle feed axis selection signal, handle feed mode in a direction perpendicular to the tool axis is enabled.

- **Rotation axis setting for handle feed function in a direction perpendicular to the tool axis**

7144	Coordinates of the first rotation axis for handle feed in the tool axis direction or handle feed in a direction perpendicular to the tool axis
7145	Coordinates of the second rotation axis for handle feed in the tool axis direction or handle feed in a direction perpendicular to the tool axis

[Data type] Two–word

[Unit of data] 0.001 degree

[Valid data range] –360000 to 360000

Specify the coordinates (angular displacement) of the first and second rotation axes used when bits 3D1 and 3D2 (bits 3 and 4 of parameter No. 7104) are both set to 1 in handle feed mode in the direction of the tool axis, or in a direction perpendicular to the tool axis. When the CXC bit (bit 2 of parameter No. 7104) is set to 1, the coordinates of the second rotation axis are assumed to be 0, irrespective of the values of bit 3D2X or these parameters.

Alarm and message

No.	Message	Meaning
5015	* No message appears.	In tool axis direction handle feed or tool axis perpendicular direction handle feed mode, a specified rotation axis does not exist.

Note

NOTE

1 The basic axes X, Y, and Z are determined by parameter No. 1022 (plane selection). The rotation axes A, B, and C are determined by parameter No. 1020 (axis name).

2 If one of the two axes specified by a type set depending on the axis configuration does not exist, alarm P/S 5015 is issued. In handle feed mode in a direction perpendicular to the tool axis, either the A–C axis type or B–C axis type must be selected as the axis configuration type.

Reference item

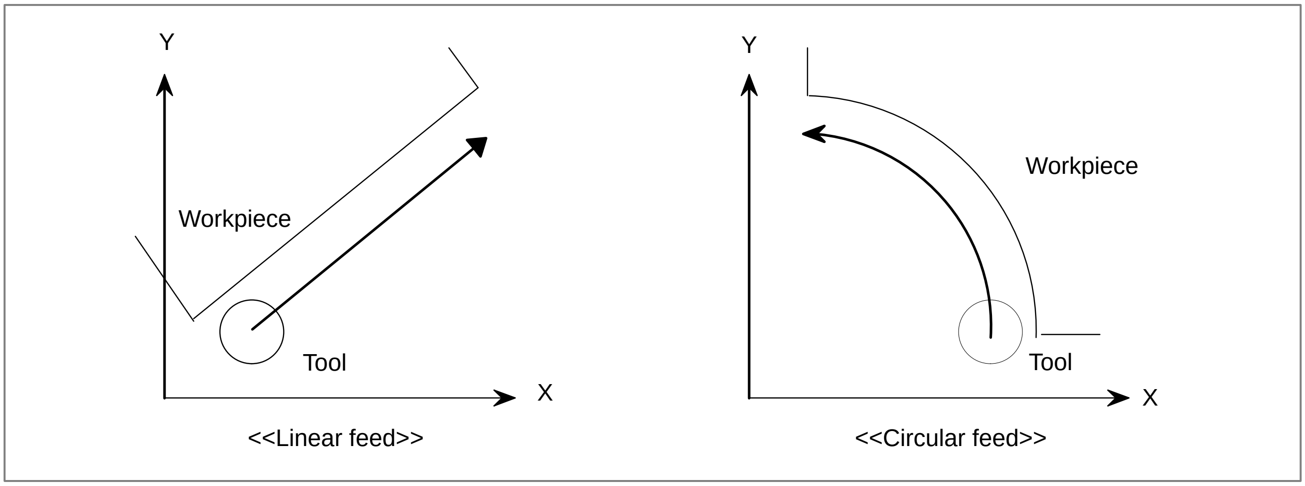
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.6	Tool axis direction handle feed / Tool axis direction handle feed B

3.5 MANUAL LINEAR/CIRCULAR INTERPOLATION

General

In manual handle feed or jog feed, the following types of feed operations are enabled in addition to the conventional feed operation along a specified single axis (X–axis, Y–axis, Z–axis, and so forth) based on simultaneous 1–axis control:

- Feed along a tilted straight line in the XY plane (M series) or ZX plane (T series) based on simultaneous 2–axis control (linear feed)
- Feed along a circle in the XY plane (M series) or ZX plane (T series) based on simultaneous 2–axis control (circular feed)



> **NOTE**
> The X–axis and Y–axis (M series) or Z–axis and X–axis (T series) must be the first controlled axis and second controlled axis, respectively. The following description applies to the X–Y plane for the M Series. For the T Series, read the X–Y plane as the Z–X plane.

Explanations

• Line/circle definition

A line or circle definition is not required for axial feed. For linear or circular feed, however, a line and circle must be defined, using the interface described later. (For circular feed, for example, the center and radius of the circle must also be specified.)

• Interface area

The R area of the PMC–RB/RC is used partly for line and circle definitions. Set the necessary data in this area, using the PMC or macro executor. See the following descriptions for what data is to be set at each address.

• Data setting

(a) Input data (PMC–RB/RC –> CNC)

Lines and circles are defined by setting the data listed below.

<table>
<tr><th rowspan="2"></th><th rowspan="2">Data name</th><th rowspan="2">Number of bytes</th><th colspan="2">Setting</th></tr>
<tr><th>Linear feed</th><th>Circular feed</th></tr>
<tr><td></td><td>R960</td><td>1</td><td colspan="2">(Reserve) Do not use.</td></tr>
<tr><td>(1)</td><td>R961</td><td>1</td><td colspan="2">Linear or circular feed selection
<table><tr><th>Set value</th><th>Description</th></tr><tr><td>0
1
2
3</td><td>Neither linear nor circular feed is carried out.
Linear feed is carried out.
Clockwise circular feed is carried out. CW
Counterclockwise circular feed is carried out. CCW</td></tr></table></td></tr>
<tr><td>(2)</td><td>R962 to R965</td><td>4</td><td>Approach direction (X–axis direction)</td><td>Center of the circle (Xo)</td></tr>
<tr><td>(3)</td><td>R966 to R969</td><td>4</td><td>Approach direction (Y–axis direction)</td><td>Center of the circle (Yo)</td></tr>
<tr><td>(4)</td><td>R970 to R973</td><td>4</td><td>Distance (P) between the origin and a given line</td><td>Radius (R) of the circle</td></tr>
<tr><td>(5)</td><td>R974</td><td>1</td><td>Direction of cutting (amount of travel corresponding to the forward rotation of the guidance handle)
<table><tr><th>Set value</th><th>Description</th></tr><tr><td>0</td><td>Direction diverted +90° from the approach direction</td></tr><tr><td>1</td><td>Direction diverted –90° from the approach direction</td></tr></table></td><td>Which is to be machined, the inside or outside of the circle?
<table><tr><th>Set value</th><th>Description</th></tr><tr><td>0</td><td>Inside</td></tr><tr><td>1</td><td>Outside</td></tr></table></td></tr>
<tr><td>(6)</td><td>R975</td><td>1</td><td colspan="2">Control flags
<table><tr><td>bit 0 to bit 6</td><td>Must be 0.</td></tr><tr><td>bit 7</td><td>Whether the limit function is enabled
0: Disabled
1: Enabled</td></tr></table></td></tr>
<tr><td></td><td>R976 to R978</td><td>3</td><td colspan="2">(Reserve) Do not use.</td></tr>
<tr><td>(7)</td><td>R979</td><td>1</td><td colspan="2">Notice of changes in the setting</td></tr>
</table>

(b) Output data (CNC –> PMC–RB/RC)

The data listed below will be output. Do not change this data.

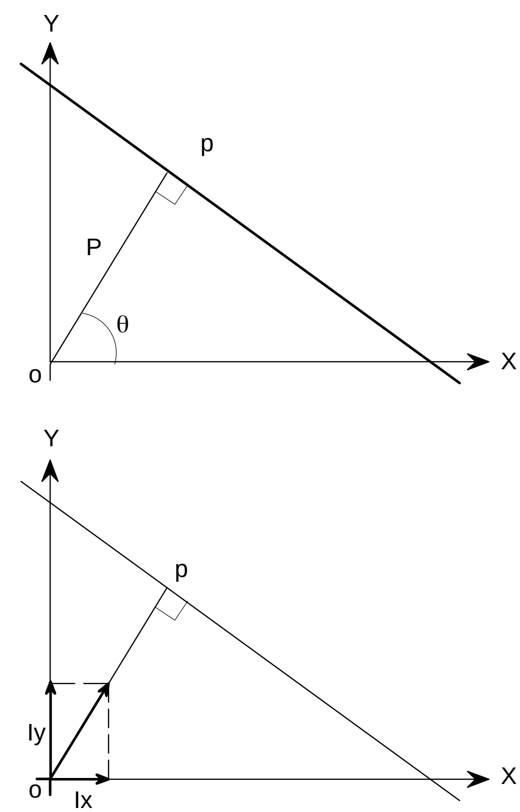
<table>
<tr><th rowspan="2"></th><th rowspan="2">Data name</th><th rowspan="2">Number of bytes</th><th colspan="2">Description</th></tr>
<tr><th>Linear feed</th><th>Circular feed</th></tr>
<tr><td>(8)</td><td>R980 to R983</td><td>4</td><td>A distance from the current position to a given line is output.</td><td>A distance from the current position to a given circle is output.</td></tr>
<tr><td></td><td>R984 to R989</td><td>6</td><td colspan="2">(Reserve) Do not use.</td></tr>
</table>

(1) Setting for linear feed

Let us assume that P is the length of a line segment starting at the origin and perpendicular to a given line, and θ is an angle formed by the perpendicular line and the positive X–axis. The given line is defined as:

$$X \cdot \cos\theta + Y \cdot \sin\theta = P$$

NOTE
The origin mentioned above is that of the absolute coordinate system. X and Y used in this description refer to coordinates in the absolute coordinate system. These coordinates represent the center of the tool.



Specify the following data:

Data	
1) R961	: Linear feed
2) R962	: Approach directionIx · 2^{30}
3) R966 to R 969	: Approach direction Iy · 2^{30}
4) R970 to R973	: Distance P between the origin and a given line
5) R974	: Direction in which the guidance handle moves
6) R975	: Whether the limit function is enabled
7) R979	: Notice of changes in the setting

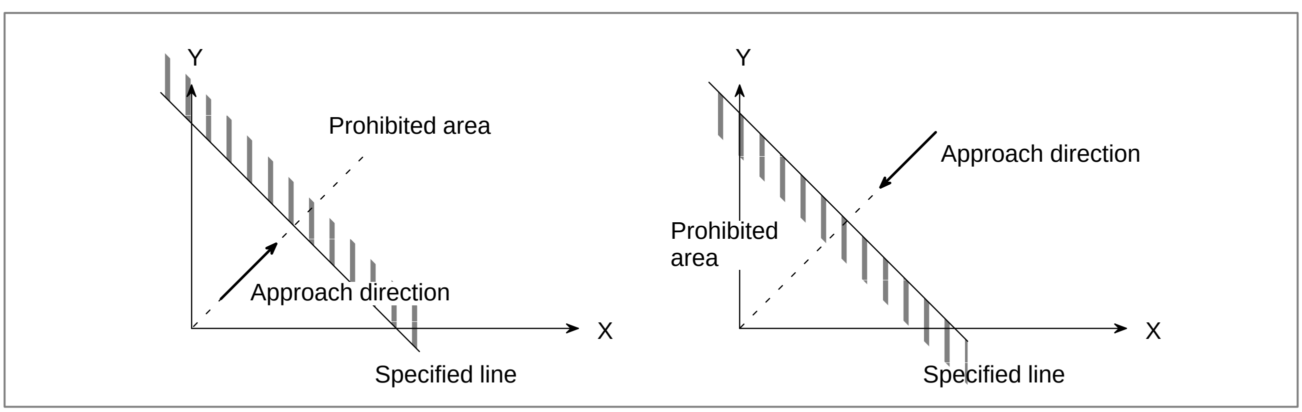
1) Select linear feed. (R961)
   Set R961 to 1.

2), 3) Specify the approach direction. (R962 to R969)
   Specify the X and Y components (Ix, Iy) of a unit vector (+cosθ , +sinθ) or (–cosθ , –sinθ), which is parallel to perpendicular op, with four bytes, using the values multiplied by 230.
   R962 to R965 = Ix 2^{30}
   R966 to R969 = Iy 2^{30}
   The tool moves in the direction indicated with this vector, when the approach handle is rotated in the forward direction.

4) Specify the length (P) of perpendicular op (line segment beginning at origin o and perpendicular to the given line) with 4 bytes. (R970 to R973)
   Length P must satisfy the following equation:
   P= $+|\overrightarrow{op}|$, where $+|\overrightarrow{op}|$ for (+cosθ , +sinθ) or $-|\overrightarrow{op}|$ for (–cosθ, –sinθ)
   The unit of P is the least input increment. (Example: For IS–B with metric input, the unit of P is 0.001 mm.)
   R970 to R973 = P

5) Specify the cutting direction. (R974)
   Specify the direction of travel, corresponding to the forward rotation of the guidance handle, with R974. The meaning of setting is as follows:
   0: Direction diverted +90° from the approach direction
   1: Direction diverted –90° from the approach direction

6) Specify whether to enable the limit function. (Bit 7 of R975)
   To disable the limit function, reset bit 7 of R975 to 0.
   To enable the limit function, set bit 7 of R975 to 1.
   When the limit function is enabled, it sets up an area which the tool is not allowed to enter. The area is delimited with a specified line. When you attempt to bring the tool into the prohibited area, using manual handle or jog feed, the tool decelerates and stops.
   The prohibited area is set up as shown below, according to the setting of the approach direction (R962 to R965, R966 to R969).



7) Notify of changes in the setting (R979).
Reset R979 to 0.

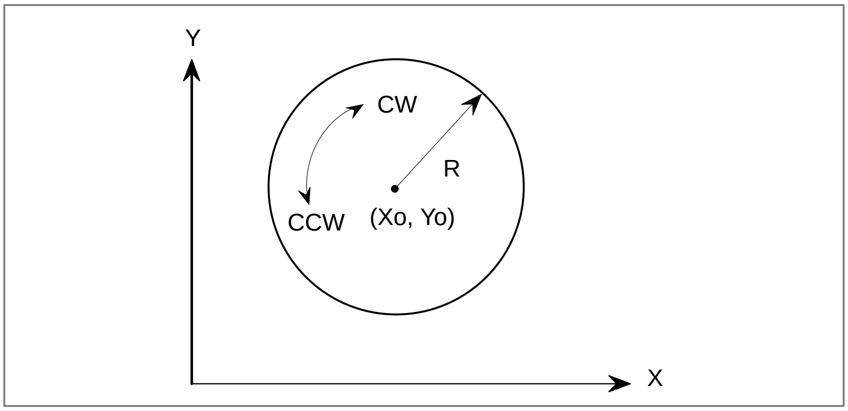
CAUTION
1 Line and circle definitions (data items 1 to 6) can be set or changed during manual operation mode (manual handle or jog feed mode). This data notifies the CNC when the definitions are changed.
After setting data items 1 to 6, reset R979 (notice of changes in the setting) to 0. When the R979 value becomes 0, the CNC assumes that data items 1 to 6 (R961 to R975) are changed, and reads them, then sets R979 back to 1. Until R979 becomes 0 again, the CNC continues to carry out linear or circular feed according to the read data.
2 Do not set R979 (notice of changes in the setting) when any axis is running.

This is the end of line definition.

8) During linear feed, a distance to a given line is calculated, using the following equation, and output to R980 to R983 (4 bytes). The unit of the data is the least input increment.
$f(X, Y) = P - (Ix \cdot X + Iy \cdot Y)$
where X, Y: Current X– and Y–axis positions

(2) Setting for circular feed

Specify the data according to the procedure below.



Data

1)	R961	: Circular feed and direction of rotation (CW/CCW)
2)	R962	: Center of the circle Xo
3)	R966 to R 969	: Center of the circle Yo
4)	R970 to R973	: Radius R
5)	R974	: Which is to be machined, inside or outside?
6)	R975	: Whether the limit function is enabled
7)	R979	: Notice of changes in the setting

1) Specify circular feed and the direction of circle rotation. (R961)

Set R961 to 2 or 3.

If R961 is 2, the tool moves along the circle clockwise, when the guidance handle is rotated in the forward direction. If R961 is 3, the tool moves along the circle counterclockwise, when the guidance handle is rotated in the forward direction.

2), 3) Specify the coordinates (Xo, Yo) of the center of the circle. (R962 to R965, R966 to R969)

NOTE
X and Y used in this description refer to coordinates in the absolute coordinate system.

R962 to R965 = Xo
R966 to R969 = Yo

Each coordinate is four bytes. The unit of the data is the least input increment.

4) Specify radius R. (R970 to R973)
R970 to R973 = R

The radius R is four bytes. The unit of the data is the least input increment.

5) Specify which is to be machined, the inside or outside of the circle. (R974)
Set R974 to 0 or 1.

If R974 is 0, the inside of the circle is machined. If R974 is 1, the outside of the circle is machined.

When the approach handle is rotated, the tool moves along a straight line normal to the specified circle. The direction of the tool movement is determined according to the setting of R974. When the approach handle is rotated in the forward direction, the direction of the tool movement (approach direction) is as follows:

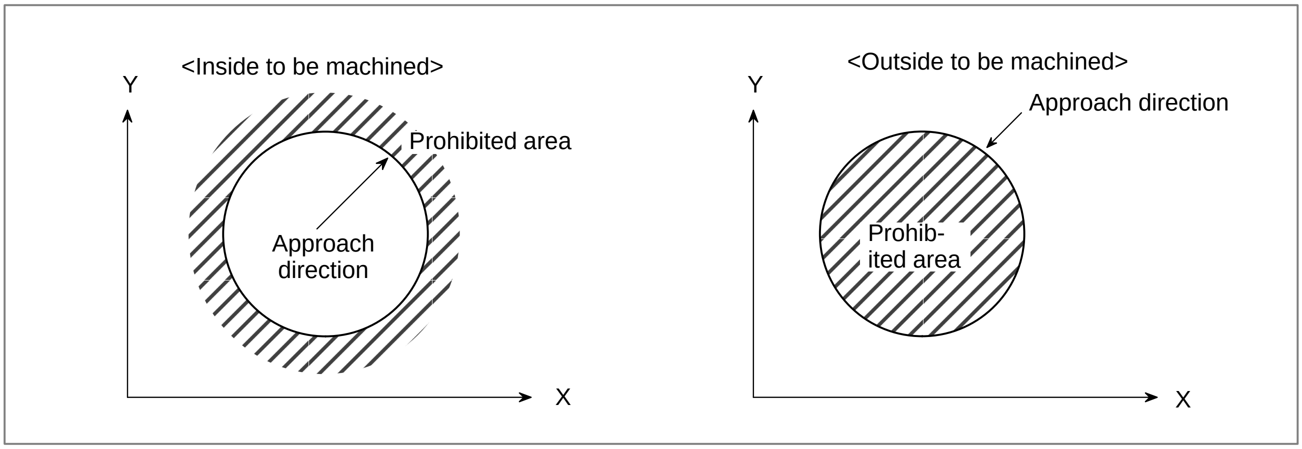
· When the inside of the circle is to be machined (R974 = 0), the tool moves from the center of the circle to the circumference.

· When the outside of the circle is to be machined (R974 = 1), the tool moves toward the center of the circle.

6) Specify whether to enable the limit function. (bit 7 of R975)
To disable the limit function, set bit 7 of R975 to 0. To enable the limit function, set bit 7 of R975 to 1.

When the limit function is enabled, it sets up an area which the tool is not allowed to enter. The area is either inside or outside of the specified circle. When you attempt to bring the tool into the prohibited area, using manual handle or jog feed, the tool decelerates and stops.

Where (inside or outside of the circle) the prohibited area is set is determined according to the setting of R974 (which is to be machined, the inside or outside of the circle). If the inside of the circle is to be machined, the prohibited area is outside the circle. If the outside of the circle is to be machined, the prohibited area is inside the circle.



7) Notify of changes in the setting (R979).
Reset R979 to 0.
This is the end of circle definition.

8) During circular feed, a distance to a given circle is calculated, using the following equation, and output to R980 to R983 (4 bytes). The unit of the data is the least input increment.
– When the inside is to be machined:

$$f(X, Y) = R - \sqrt{(X - X_0)^2 + (Y - Y_0)^2}$$

– When the outside is to be machined:

$$f(X, Y) = \sqrt{(X - X_0)^2 + (Y - Y_0)^2} - R$$

(3) Setting for linear and circular feed

Specify the following data:

Data		
1)	R961	: 0 (Neither linear nor circular feed is carried out.)
2)	R962	: (Need not be specified.)
3)	R966 to R 969	: (Need not be specified.)
4)	R970 to R973	: (Need not be specified.)
5)	R974	: (Need not be specified.)
6)	R975	: (Need not be specified.)
7)	R979	: Notice of changes in the setting

1) Linear or circular feed selection
Reset R961 to 0.
If R961 is 0, both the guidance and approach handles become ineffective. The tool will not move, even if these handles are rotated.

2) to 6) It is unnecessary to set R962 to R975.

7) Notify of changes in the setting (R979).
Reset R979 to 0.

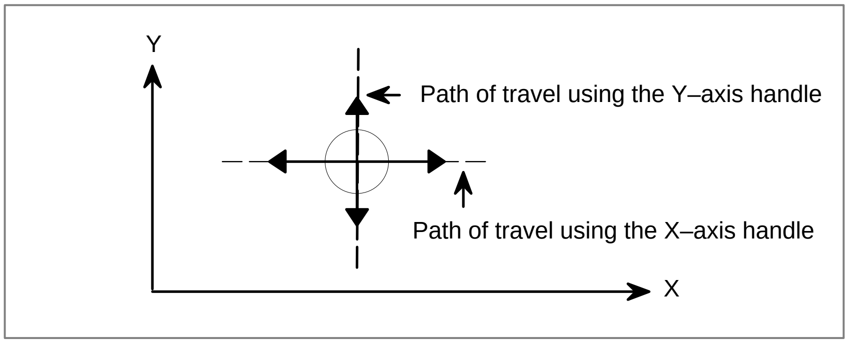
8) The values of R980 to R983 (distance to a given line or circle) are output as 0.

- **Manual handle feed**

In manual handle feed, the tool can be moved along a specified axis (X–axis, Y–axis, Z–axis, ..., or the 8th axis), or can be moved along a tilted straight line (linear feed) or a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1–axis control)

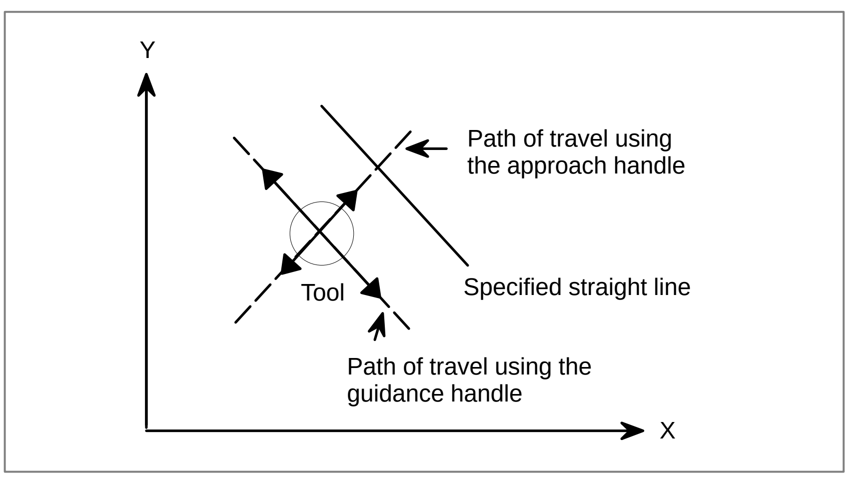
By turning a manual handle, the tool can be moved along the desired axis (such as X–axis, Y–axis, and Z–axis) on a simultaneous 1–axis control basis. (This mode of feed is the conventional type of manual handle feed.)



Feed along a specified axis

(2) Linear feed (simultaneous 2–axis control)

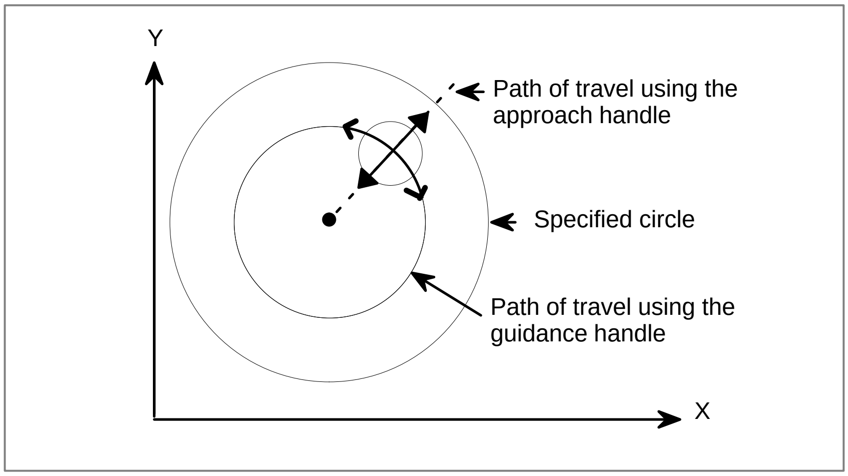
By turning a manual handle, the tool can be moved along the straight line parallel to a specified straight line on a simultaneous 2–axis control basis. This manual handle is referred to as the guidance handle. Moreover, by turning another manual handle, the tool can be moved at right angles to a specified straight line on a simultaneous 2–axis control basis. This manual handle is referred to as the approach handle. When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Linear feed

(3) Circular feed (simultaneous 2–axis control)

By turning a manual handle, the tool can be moved from the current position along the concentric circle that has the same center as a specified circle on a simultaneous 2–axis control basis. This manual handle is referred to as the guidance handle. Moreover, by turning another manual handle, the tool can be moved along the normal to a specified circle on a simultaneous 2–axis control basis. This manual handle is referred to as the approach handle. When the guidance handle or approach handle is turned clockwise or counterclockwise, the tool travels forward or backward along the respective path.



Circular feed

- **Feedrate for manual handle feed**

The feedrate depends on the speed at which a manual handle is turned. A distance to be traveled by the tool (along a tangent in the case of linear or circular feed) when a manual handle is turned by one pulse can be selected using the manual handle feed travel distance magnification switch, MP1 or MP2.

- **Manual handle selection**

The Series 16/18–M has three (The Series 16/18–T has two) manual pulse generator interfaces to allow up to three (in T–series two) manual handles to be connected. The use of the manual handles connected to the interfaces (whether to use each manual handle as a handle for feed along an axis, as a guidance handle, or as an approach handle), can be selected by Manual handle feed axis selection signal in each interface. And by switching them, one manual handle can be used in plural purposes.

- **Direction of movement using manual handles**

The user can specify the direction of the tool moved along a straight line or circle (for example, whether to make a clockwise or counterclockwise movement along a circle) when the guidance handle or approach handle is turned clockwise or counterclockwise. For details, refer to the relevant manual provided by the machine tool builder.

- **Jog feed (JOG)**

In jog feed, the tool can be moved along a specified axis (X–axis, Y–axis, Z–axis, ..., or the 8th axis), or can be moved along a tilted straight line (linear feed) or a circle (circular feed).

(1) Feed along a specified axis (simultaneous 1–axis control)

While a feed axis and its direction are specified with the feed axis direction select switch, the tool moves in the specified axis direction at the feedrate specified in parameter No. 1423. The feedrate can be overridden using the manual feedrate override dial.

(2) Linear feed (simultaneous 2–axis control)

By defining a straight line beforehand, the tool can be moved as follows:

- While a feed axis and its direction are selected using the feed axis direction select switch, the tool moves along a straight line parallel to the specified straight line on a simultaneous 2–axis control basis.
- While a feed axis and its direction are selected using the feed axis direction select switch, the tool moves at right angles to the specified straight line on a simultaneous 2–axis control basis.

The feedrate in the tangential direction is specified in parameter No. 1410. The feedrate can be overridden (0.01% ~ 655.34%) using the manual feedrate override dial. (*JV0 ~ *JV15)

(3) Circular feed (simultaneous 2–axis control)

By defining a circle beforehand, the tool can be moved as follows:

- While a feed axis and its direction are selected using the feed axis γ direction select switch (+Jg, −Jg), the tool moves from the current position along the concentric circle that has the same center as the specified circle.
- While a feed axis and its direction are selected using the feed axis γ direction select switch (+Jg, −Jg), the tool moves along the normal to the specified circle.

The feedrate in the tangential direction is specified in parameter No. 1410. The feedrate can be overridden (0.01% ~ 655.34%) using the manual feedrate override dial. (*JV0 ~ *JV15)

• Manual handle feed in JOG mode

Even in JOG mode, manual handle feed can be enabled using bit 0 (JHD) of parameter No. 7100. In this case, however, manual handle feed is enabled only when the tool is not moved along any axis by jog feed.

• Basic procedure

(1) Select manual operation mode.

To perform manual handle feed, select manual handle feed mode. To perform jog feed, select jog feed mode.

(2) Define a line or arc.

See the relevant descriptions in Explanation.

(3) Move the tool by means of manual handle feed or jog feed.

To perform manual handle feed, select the axis along which the tool will move when the manual handle is turned (single–axis feed along the X–, Y–, or Z–axis, or simultaneous two–axis feed along a specified line or arc, involving both the X– and Y–axes), using manual handle feed axis selection signals HSnA, HSsB, HSsC, and HSsD.

Subsequently, turning the manual handle will move the tool along the selected axis. The feedrate varies with the speed at which the manual handle is turned.

The amount by which the tool will be moved when the manual handle is turned through the angle corresponding to one pulse can be specified using manual handle feed amount selection signals MP1 and MP2.

To perform jog feed, select the feed axis and the direction in which the tool is to be moved, using the feed axis and direction selection signals (+J1, –J1, +J2, –J2, ... +J8, –J8). While the feed axis and direction are specified, the tool is moved along the specified axis, or specified line or arc, at the parameter–set feedrate (jog feedrate).

Manual feedrate override signals (*JV0 to *JV15) can be used to apply an override (0.01% to 655.34%) to the feedrate.

Limitations

- **Mirror image**

Never use the mirror image function when performing manual operation. (Perform manual operation when the mirror image switch (MI1 ~ MI3) is off, and mirror image setting is off.)

- **Two–path control**

Manual linear/circular interpolation can not used with two–path control.

Signal

The following signals determine the way in which jog feed or manual handle feed is executed.

Selection	Jog feed	Manual handle feed
Mode selection	MD1, MD2, MD4	MD1, MD2, MD4
Selection of the axis to move	+J1, –J1, +J2, –J2, +J3, –J3, ... +J8, –J8, +Jg, –Jg, +Ja, –Ja, ...	HS1A, HS1B, HS1C, HS1D, HS2A, HS2B, HS2C, HS2D, HS3A, HS3B, HS3C, HS3D
Selection of the direction to move the axis		
Selection of the move amount		MP1, MP2
Selection of feedrate	*JV0 – *JV15	

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 manual feedrate override signals *JV0 – *JV15, see Section 3.1, “Jog feed/incremental feed.”
Other signals are described below.

Feed Axis and Direction Selection Signal +J1 – +J8<G100> –J1 – –J8<G102> +Jg, –Jg, +Ja, –Ja<G086>

[Classification] Input signal

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

+ J 1

1 First axis is fed
2 Second axis is fed
3 Third axis is fed
: :
8 Eighth axis is fed
g :Both X– and Y–axes (M Series) or Z– and X–axes (T Series) are fed simultaneously. (Guidance jog: Linear or circular feed)
a :Both X– and Y–axes (M Series) or Z– and X–axes (T Series) are fed simultaneously. (Approach jog: Feed along a normal)

\+ Feed in positive direction
– Feed in negative direction

[Operation] See Section 3.1 for the operation.

Manual Handle Feed Axis Selection Signals

- **(M series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>
  HS3A – HS3D
  <G019#0 – #3>

- **(T series)**
  HS1A – HS1D
  <G018#0 – #3>
  HS2A – HS2D
  <G018#4 – #7>

[Classification] Input signal

[Function] Selects the axis of manual handle feed. A set of four code signals, A, B, C, and D is provided for each manual pulse generator. (Up to three generators can be used.) (For two–path, these signals are provided for each manual pulse generator and each path.) The number in the signal name indicates the number of the manual pulse generator to be used.

HS 1 A

1 First manual pulse generator
2 Second manual pulse generator
3 Third manual pulse generator (M series)

Code signals A, B, C, and D correspond to the feed axes as listed in the following table:

Manual handle feed axis selection				Feed axis
HSnD	HSnC	HSnB	HSnA	
0	0	0	0	No selection (None of axis is fed)
0	0	0	1	1st axis
0	0	1	0	2nd axis
0	0	1	1	3rd axis
0	1	0	0	4th axis
0	1	0	1	5th axis
0	1	1	0	6th axis
0	1	1	1	7th axis
1	0	0	0	8th axis
1	1	1	0	XY simultaneous 2 axes (M serios) ZX simultaneous 2 axes (T series) (Guidance handle)
1	1	1	1	XY simultaneous 2 axes (M series) ZX simultaneous 2 axes (T series) (Approach handle)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1
G102	–J8	–J7	–J6	–J5	–J4	–J3	–J2	–J1
G086					–Ja	+Ja	–Jg	+Jg

<For 1–path control>

	#7	#6	#5	#4	#3	#2	#1	#0
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019					HS3D	HS3C	HS3B	HS3A

Parameter

1410	Dry run rate/Jog feedrate (linear feed, circular 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 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the dry run with auto operation rate or jog feedrate with linear feed, circular feed when the manual feedrate is overridden by 100%.

1423	Feedrate in manual continuous feed (jog feed) for each axis

[Data type] Word axis

(1) In M series, or in T series when JRV, bit 4 of parameter No. 1402, is set to 0 (feed per minute), specify a jog feedrate at feed per minute with an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	6 – 32767
Inch machine	0.1 inch/min	
Rotation axis	1 deg/min	

(2) When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a jog feedrate (feed per revolution) under an override of 100%.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 – 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

	#7	#6	#5	#4	#3	#2	#1	#0
7100							THD	JHD

[Data type] Bit

JHD Manual handle feed in JOG mode or incremental feed in the manual handle feed

0 : Invalid
1 : Valid

THD Manual pulse generator in TEACH IN JOG mode

0 : Invalid
1 : Valid

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

[Data type] Bit axis

HNGx Axis movement direction for rotation direction of manual pulse generator

0: Same in direction
1: Reverse in direction

7110	Number of manual pulse generators used

[Data type] Byte

[Valid data range] 1, 2, or 3

This parameter sets the number of manual pulse generators.

7113	Manual handle feed magnification m

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 127

This parameter sets the magnification when manual handle feed movement selection signal MP2 is on.

7114	Manual handle feed magnification n

[Data type] Word

[Unit of data] One time

[Valid data range] 1 to 1000

This parameter sets the magnification when manual handle feed movement selection signals MP1 and MP2 are “1”.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.3.7	Manual linear/circular interpolation
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.3.6	Manual linear/circular interpolation

3.6 MANUAL RIGID TAPPING (M SERIES)

General

For execution of rigid tapping, set rigid mode, then switch to handle mode and move the tapping axis with a manual handle.
Manual rigid tapping is enabled by setting bit 0 (HRG) of parameter No. 5203 to 1.

• Basic Procedure

1 Stop the spindle and servo axes, then set MDI mode by pressing the MDI switch among the mode selection switches.

2 Enter and execute the following program:

M29 S1000 ;
G91 G84 Z0 F1000 ;

The program above is required to determine a screw lead and set rigid tapping mode. In this program, a tapping axis must always be specified. Specify a value that does not operate the tapping axis.

> **WARNING**
> In this MDI programming, never specify commands to position the tool at a drilling position and at point R. Otherwise, the tool moves along an axis.

3 When the entered program is executed, rigid tapping mode is set.

4 After rigid mode is set upon completion of MDI program execution, switch to the handle mode by pressing the handle switch among the mode selection switches.

> **CAUTION**
> At this time, never press the reset key. Otherwise, rigid mode is canceled.

5 To perform rigid tapping, select a tapping axis with the handle feed axis select switch, and move the tapping axis with the manual handle.

• Cancellation of rigid mode

To cancel rigid mode, specify G80 as same the normal rigid tapping. When the reset key is pressed, rigid mode is canceled, but the canned cycle is not canceled.

When the rigid mode switch is to be set to off for rigid mode cancellation (when bit 2 (CRG) of parameter No. 5200 is set to 0), the G80 command ends after the rigid mode switch is set to off.

• Spindle rotation direction

The rotation direction of the spindle is determined by a specified tapping cycle G code and the setting of bit 1 (HRM) of parameter No. 5203. For example, when the HRM parameter is set to 0 in G84 mode, the spindle makes forward rotations as the tapping axis moves in the minus direction. (When the tapping axis moves in the plus direction, the spindle makes reverse rotations.)

- **Arbitrary tapping axis**

By setting bit 0 (FXY) of parameter No. 5101 to 1, an arbitrary tapping axis can be selected. In this case, specify a G code for plane selection and tapping axis address when rigid mode is set in MDI mode.

- **Specification of M29 and G84 in the same block**

In an MDI program for setting rigid mode, G84 can be used as a rigid tapping G code, or M29 and G84 can be specified in the same block.

- **Acceleration/ deceleration type**

When manual rigid tapping is executed, the acceleration/deceleration type and acceleration/deceleration time constant specified in the rigid tapping parameters become valid.

- **Specification of manual handle feed faster than the rapid traverse rate**

Set bit 0 (HPF) of parameter No. 7100 to 0 so that when manual handle feed is specified which is faster than the rapid traverse rate, the handle pulses beyond the rapid traverse rate are ignored.

Limitations

- **Check for excessive error**

In manual rigid tapping mode, only excessive error during movement is checked.

- **Tool–axis direction handle feed**

Tool–axis direction handle feed cannot be performed.

- **Extraction override**

In manual rigid tapping mode, an extraction override or extraction acceleration/deceleration time constant cannot be used.

- **Repetition count**

Do not specify K0 or L0 (which sets the repetition count to 0, such that a G84 block is not executed) in the MDI program. Otherwise, rigid tapping mode cannot be set.

- **Positioning to the drilling position**

Positioning to the drilling position must be performed in handle mode, by selecting the X– or Y–axis using the axis selection signal. Do not perform positioning to the drilling position in MDI or MEM mode because any such attempt may cause the tapping axis to move.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5203							HRM	HRG

HRG Rigid tapping by the manual handle is:
0 : Disabled.
1 : Enabled.

HRM When the tapping axis moves in the negative direction during rigid tapping controlled by the manual handle, the direction in which the spindle rotates is determined as follows:
0 : In G84 mode, the spindle rotates in a normal direction. In G74 mode, the spindle rotates in reverse.
1 : In G84 mode, the spindle rotates in reverse. In G74 mode, the spindle rotates in a normal direction.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.3.8	MANUAL RIGID TAPPING
	OPERATOR’S MANUAL (This manual)	9.11	RIGID TAPPING

3.7 MANUAL NUMERIC COMMAND

General

The manual numeric command function allows data programmed through the MDI to be executed in jog mode. Whenever the system is ready for jog feed, a manual numeric command can be executed. The following eight functions are supported:

(1) Positioning (G00)
(2) Liner interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)
(5) M codes (miscellaneous functions)
(6) S codes (spindle functions)
(7) T codes (tool functions) (M series)
(8) B codes (second auxiliary functions)

By setting the following parameters, the commands for axial motion and the M, S, T, and B functions can be disabled:

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)

Bit 0 (JAXx) of parameter No. 7010

(5) M codes (miscellaneous functions):
Bit 0 (JMF) of parameter No. 7002
(6) S codes (spindle functions):
Bit 1 (JSF) of parameter No. 7002
(7) T codes (tool functions) (M series):
Bit 2 (JTF) of parameter No. 7002
(8) B codes (second auxiliary functions):
Bit 3 (JBF) of parameter No. 7002

Explanations

• Positioning

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified.

The tool moves along each axis independently at the rapid traverse rate. Linear interpolation type positioning (where the tool path is linear) can also be performed by setting bit 1 (LRP) of parameter No. 1401.

	Manual rapid traverse selection switch	
	Off	On
Feedrate (parameter)	Jog feed rate for each axis (No. 1423)	Rapid traverse rate for each axis (No. 1420)
Automatic acceleration/deceleration (parameter)	Exponential acceleration/deceleration in jog feed for each axis (No. 1624)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Manual feed override	Rapid traverse override

> **NOTE**
> When the manual rapid traverse selection signal RT is 0, the jog feedrate for each axis is clamped such that a parameter–set feedrate determined by bit 1 (LRP) of parameter No. 1401 as shown below, is not exceeded.
> LRP = 0 : Manual rapid traverse rate for each axis (parameter No. 1424)
> LRP = 1 : Rapid traverse rate for each axis (parameter No. 1420)

- **Linear interpolation (G01)**

An amount of travel is given as a numeric value, preceded by an address such as X, Y, or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified. Axial movement is always performed in incremental mode even during scaling or polar coordinate interpolation. In addition, movement is always performed in feed per minute mode regardless of the specification of G94 or G95.

Feedrate (parameter)	Dry run feedrate (No. 1410)
Automatic acceleration/deceleration (parameter)	Exponential acceleration/deceleration in cutting feed for each axis (No. 1622)
Override	Manual feed override

> **NOTE**
> Since the feedrate is always set to the dry run feedrate, regardless of the setting of dry run signal DRN, the feedrate cannot be specified using F. The feedrate is clamped such that the maximum cutting feedrate, set in parameter No. 1422, is not exceeded.

- **Automatic reference position return (G28)**

The tool returns directly to the reference position without passing through any intermediate points, regardless of the specified amount of travel. For those axes for which no move command is specified, however, a return operation is not performed.

Feedrate (parameter)	Rapid traverse rate for each axis (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

- **2nd, 3rd, or 4th reference position return (G30)**

The tool returns directly to the 2nd, 3rd, or 4th reference position without passing through any intermediate points, regardless of the specified amount of travel. To select a reference position, specify P2, P3, or P4 in address P. If address P is omitted, a return to the second reference position is performed.

Feedrate (parameter)	Rapid traverse rate for each axis (No. 1420)
Automatic acceleration/deceleration (parameter)	Linear acceleration/deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

NOTE

The function for 3rd/4th reference position return is optional.

- When the option is not selected

Return to the 2nd reference position is performed, regardless of the specification of address P.

- When the option is selected

If none of P2, P3, or P4 is specified in address P, a “START IMPOSSIBLE” warning is generated, and the entered data cannot be processed.

- **M codes (miscellaneous functions)**

After address M, specify a numeric value of no more than the number of digits specified by parameter No. 3030. When M98 or M99 is specified, it is executed but not output to the PMC.

NOTE

Neither subprogram calls nor custom macro calls can be performed using M codes.

- **S codes (spindle functions)**

After address S, specify a numeric value of no more than the number of digits specified by parameter No. 3031.

NOTE

Subprogram calls cannot be performed using S codes.

- **T codes (M series) (tool functions)**

After address T, specify a numeric value of no more than the number of digits specified by parameter No. 3032.

NOTE

Subprogram calls cannot be performed using T codes.

- **B codes (second auxiliary functions)**

After address B, specify a numeric value of no more than the number of digits specified by parameter No. 3033.

> **NOTE**
>
> 1 B codes can be named U, V, W, A, or C by setting parameter No. 3460. If the new name is the same as an axis name address, B is used. When B is used, and axis name B exists, B is used as the axis address. In this case, no second auxiliary function can be specified.
> 2 Subprogram calls cannot be performed using B codes.

- **Constant surface speed control**

S codes cannot be specified in constant surface speed control mode.

- **M, S, T, and B functions**

While automatic operation is halted, manual numeric commands can be executed. In the following cases, however, a ***F.314–5*** warning is output, and command execution is disabled.

(1) While an M, S, T, or B function is being executed, a manual numeric command containing an M, S, T, or B function cannot be executed.

(2) While an M, S, T, or B function is being executed, and that function is specified alone, or a block specifying that function also contains any other function (such as a move command or dwell function) that has already been completed, a manual numeric command cannot be executed.

- **Jog feed**

When a manual numeric command is specified while the tool is being moved along an axis using the feed axis direction selection signal ±Jx, the axial movement is interrupted, and the manual numeric command is executed. Therefore, the tool cannot be moved along an axis using the tool direction selection signal ±Jx during the execution of a manual numeric command.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001						JSL		

[Data type] Bit

JSL Specifies whether to output automatic operation signal STL during automatic operation based on a manual numeric command.

0 : Not output.
1 : Output.

	#7	#6	#5	#4	#3	#2	#1	#0
7002					JBF	JTF	JSF	JMF

[Data type] Bit

JMF Specifies whether to support the M function for the manual numeric command.

0 : Supported.
1 : Not supported.

JSF Specifies whether to support the S function for the manual numeric command.

0 : Supported.
1 : Not supported.

JTF Specifies whether to support the T function for the manual numeric command.

0 : Supported.
1 : Not supported.

JBF Specifies whether to support the B function for the manual numeric command.

0 : Supported.
1 : Not supported.

	#7	#6	#5	#4	#3	#2	#1	#0
7010								JAXx

[Data type] Bit axis

JAXx Specifies whether to support axis movement commands for the manual numeric command.

0 : Supported.
1 : Not supported.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.9	Manual numeric command
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.3.9	Manual numeric command

4 REFERENCE POSITION ESTABLISHMENT

4.1 MANUAL REFERENCE POSITION RETURN

General

The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) 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**

When the option for a workpiece coordinate system is not used, Bit 0 of parameter 1201 (ZPR) can be set to automatically determine the coordinate system at manual reference position return. Parameter 1250 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.

> **NOTE**
> Automatic coordinate system setting is not performed if the option for a workpiece coordinate system is used. In that case, manual reference position return always establishes a workpiece coordinate system based on the workpiece origin offsets, specified with parameters No. 1220 to 1226.

The following signals relate with the manual reference position return:

	Manual Reference Position Return
Mode selection	MD1, MD2, MD4
Selection of reference position return	ZRN, MREF
Selection of axis to be moved	+J1, –J1, +J2, –J2, +J3, –J3, ...
Selection of direction to be moved	
Selection of speed to be moved	ROV1, ROV2
Deceleration signal for reference position return	*DEC1, *DEC2, *DEC3, ...
Completion signal for reference position return	ZP1, ZP2, ZP3, ...
Reference position establishment signal	ZRF1, ZRF2, ZRF3, ...

Basic Procedure for Manual Reference Position Return

(1) Select 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 (+J1, –J1, +J2, –J2,…) “1”.

(3) While the feed axis and direction selection signal is “1”, rapid traverse takes place along that axis. Although the rapid traverse override signals (ROV1, ROV2) are valid, the override is generally set to 100%.

(4) When the reference position is approached, a limit switch installed on the machine is turned on, making the deceleration signal (*DEC1, *DEC2, *DEC3,…) 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. 1425) setting).

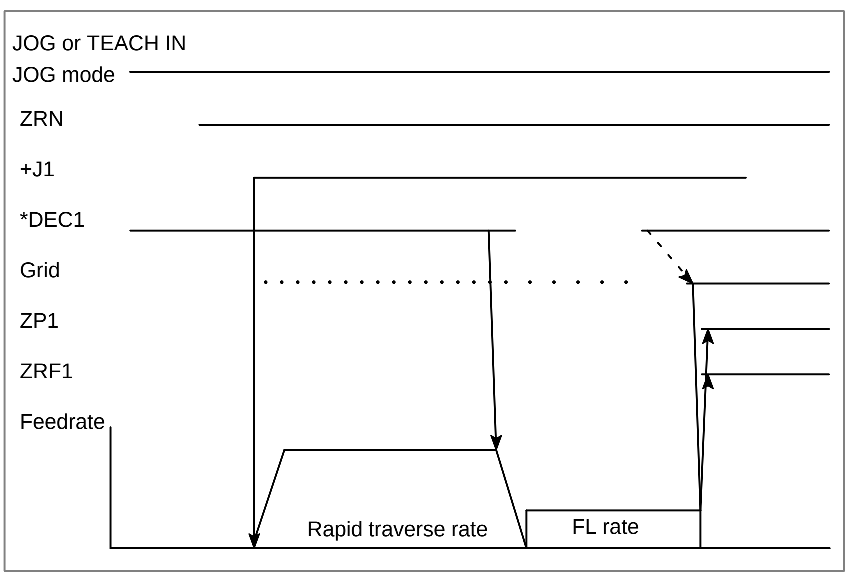
(5) When the deceleration signal turns to “1” again after the limit switch for deceleration is passed, the tool is fed with the feedrate unchanged, then the tool stops at the first grid point (electric grid point).

(6) Upon confirmation that the current position is in the in–position area, the reference position return end signal (ZP1, ZP2, ZP3,…) and the reference position establishment signal (ZRF1, ZRF2, ZRF3,…) 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 JAX (No. 1002#0).

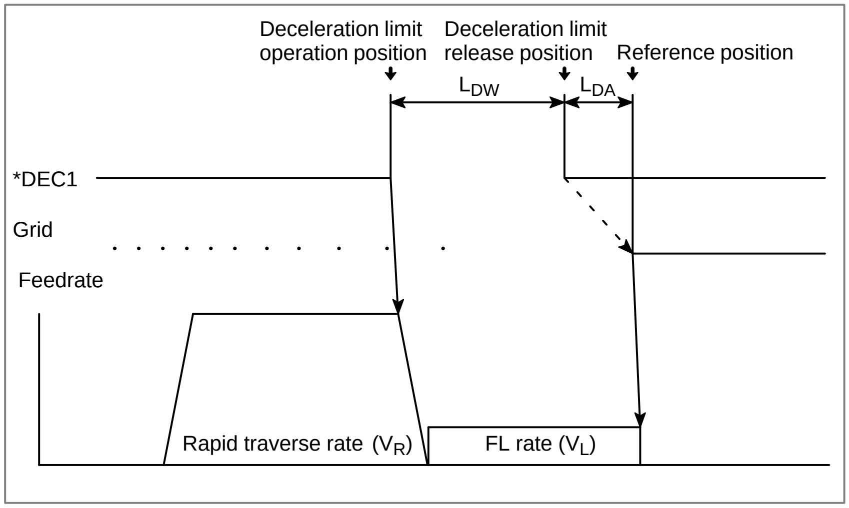
If the feed axis direction selection signal (+J1, –J1, +J2, –J2,…) turns to “0” between step (2) and (5) , the tool is stopped at once, and reference position return is assumed to be canceled. If the signal turn to “1” again, operation resumes from step (3) (rapid traverse).

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 exceeds the value set in parameter No. 1836. 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}, the detection unit is 1 μm, and parameter No. 1836 is set to 128, the tool must be fed in the reference position return direction at a speed of at least 230 mm/min before manual reference position return.

Grid shift

The grid can be shifted by the distance set in parameter 1850, thus shifting the reference position. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821) (grid interval).

Signal

Manual reference position return selection signal (ZRN) <G043#7>

[Classification] Input signal

[Function] This signal selects manual reference position return. Manual reference position return is a kind of jog feed. Therefor, to select manual reference position return, it is required that the jog mode be selected and that the manual reference position return selection signal be set to “1”.

[Operation] When the manual reference position return selection signal is set to “1”, the control unit 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. In this case, manual reference position return selection check signal MREF turns to “1”.

> **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”.

Manual reference position return selection check signal MREF <F004#5>

[Classification] Output signal

[Function] This signal reports that manual reference position return has been selected once.

[Output condition] This signal turns to "1" when:

· Manual reference position return has been selected.

The signal turns to "0" when:

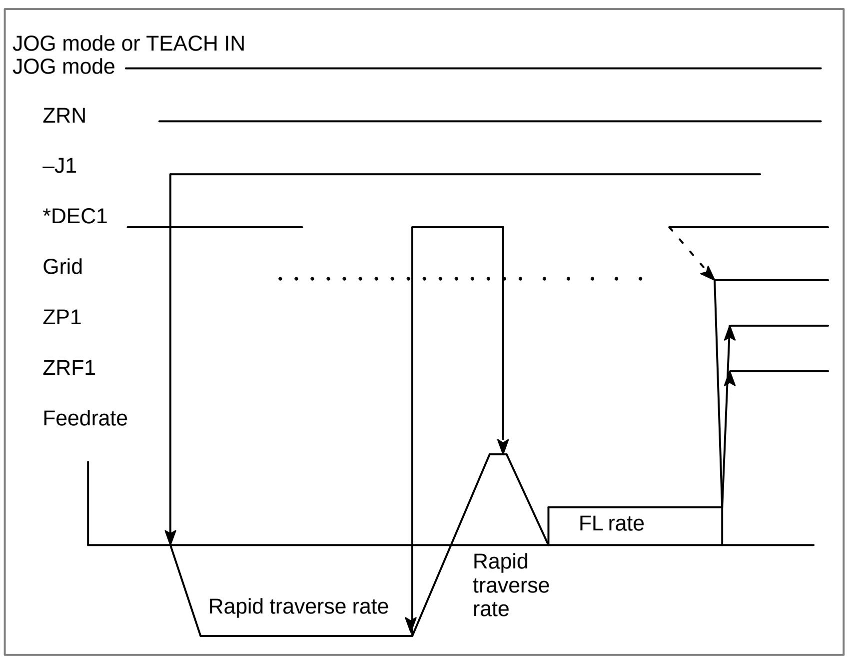
· The selection of manual reference position return has terminated.

Feed Axis and Direction Selection Signal

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

NOTE

The direction of reference position return is predetermined for each axis by parameter ZMI (No. 1006#5). 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**
> 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 *DEC1 to *DEC8 <X009>

[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 appended to a deceleration signal represents a controlled axis number.

*DEC 1

1 : Reference position return deceleration signal for the first axis
2 : Reference position return deceleration signal for the second axis
3 : Reference position return deceleration signal for the third axis
:
:

[Operation] For 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 end signals ZP1 to ZP8 <F094>

[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 appended to a signal represents a controlled axis number.

ZP 1

1 : Reference position return end signal for the first axis
2 : Reference position return end signal for the second axis
3 : Reference position return end signal for the third axis
:
:

[Output condition] These signals 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 ZRF1 to ZRF8<F120>

[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 of the controlled axis.

ZRF 1

1 : 1st–axis reference position establishment signal
2 : 2nd–axis reference position establishment signal
3 : 3rd–axis reference position establishment signal
:
:

[Output condition] The signals are set to 1 in the following case:

· When the reference position is established after manual reference position return

· When the reference position is established using the absolute–position detector at initial power–on

The signals are set to 0 in the following case:

· When the reference position is lost

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

	#7	#6	#5	#4	#3	#2	#1	#0
G043	ZRN							

	#7	#6	#5	#4	#3	#2	#1	#0
F004			MREF					
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
F120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002								JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in JOG feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

	#7	#6	#5	#4	#3	#2	#1	#0
1005								ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement except for G28 is issued in automatic operation (MEM, RMT, or MDI) when a return to the reference position has not been performed since the power was turned on

0 : An alarm is generated. (P/S alarm 224).
1 : An alarm is not generated.

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

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

[Data type] Bit axis

ZMIx The direction of reference position return

0 : Positive direction
1 : Negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL	ZPI	ZPR

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed

0 : Not set automatically
1 : Set automatically

ZPI Coordinates at the reference position when a coordinate system is set automatically

0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

ZCL Local coordinate system when the manual reference position return is performed

0 : The local coordinate system is not canceled.
1 : The local coordinate system is canceled.

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

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

1250	Coordinate value of the reference position on each axis used for setting a coordinate system automatically

[Data type] Two–word axis

[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.

1251	Coordinate value of the reference position on each axis used for setting a coordinate system automatically when input is performed in inches

[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 ZPI in parameter 1201#1 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual reference position return

0 : The stroke limit 1 is checked.

1 : The stroke limit 1 is not checked

	#7	#6	#5	#4	#3	#2	#1	#0
1401						JZR		

[Data type] Bit

JZR The manual reference position return at JOG feedrate

0 : Not performed

1 : Performed

1425	FL rate of the reference position return for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in the halt state during automatic operation (feed hold stop state) under any of the conditions listed below:

0 : Manual reference position return is not performed, with P/S alarm No. 091.

1 : Manual reference position return is performed without an alarm occurring.

< Conditions >

- When there is a remaining distance to travel.
- When an auxiliary function (miscellaneous function, spindle–speed function, tool function) is being executed.
- When a dwell or cycle such as a canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

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.

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets a servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

WARNING
When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.
Example:When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

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

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

	#7	#6	#5	#4	#3	#2	#1	#0
3003			DEC					

[Data type] Bit

DEC Deceleration signal (*DEC1 to *DEC8) for manual reference position return

0 : Deceleration is applied when the signal is 0.
1 : Deceleration is applied when the signal is 1.

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	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 point before cycle start. (Only when parameter ZRNx (No. 1005#0). Do reference position return.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.1	MANUAL REFERENCE POSITION RETURN
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.3.1	MANUAL REFERENCE POSITION RETURN

Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.3.1	MANUAL REFERENCE POSITION RETURN
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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

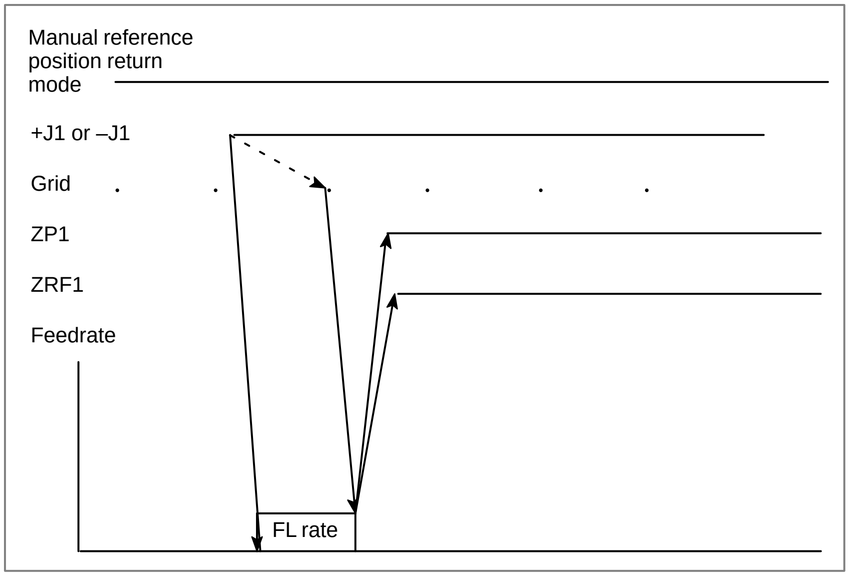
(1) Feed the tool, along the axis for which the reference position is to be set, by manual continuous feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.

(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.

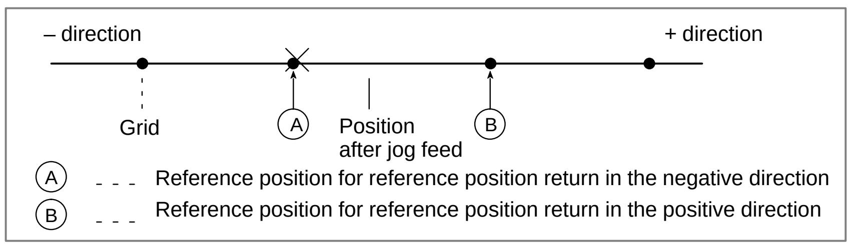
(3) The CNC positions the tool to the nearest grid line (based on one–rotation signals from the position detector) in the reference position return direction specified with bit 5 (ZMIx) of parameter No. 1006. The point to which the tool is thus positioned becomes the reference position.

(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 exceeds the value set in parameter No. 1836. The tool must cross the grid line corresponding to a one–rotation signal from the position detector.

Section 4.1 explains how to calculate the servo position error.

Grid shift

To shift the reference position, the grid can be shifted by the distance set in parameter No. 1850. The grid shift to be set in the parameter must not exceed the reference counter capacity (parameter No. 1821).

Reference position return

When the feed axis and direction selection signal is set to 1 in manual reference position return mode after the reference position has been established, the tool is positioned to the reference position regardless of the direction specified 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
1002							DLZ	JAX

[Data type] Bit

JAX Number of axes controlled simultaneously in manual continuous feed, manual rapid traverse and manual reference position return

0 : 1 axis
1 : 3 axes

DLZ Function setting the reference position without dog

0 : Disabled
1 : Enabled

NOTE
This function can be specified for each axis by DLZx, bit 1 of parameter No. 1005.

	#7	#6	#5	#4	#3	#2	#1	#0
1005							DLZx	ZRNx

[Data type] Bit axis

ZRNx When a command specifying the movement except for G28 is issued in automatic operation (MEM, RMT, or MDI) and when a return to the reference position has not been performed since the power was turned on

0 : An alarm is generated (P/S alarm 224).
1 : An alarm is not generated.

DLZx Function for setting the reference position without dogs

0 : Disabled
1 : Enabled

NOTE
When DLZ of parameter No. 1002#1 is 0, DLZx is enabled. When DLZ of parameter No. 1002#1 is 1, DLZx is disabled, and the function for setting the reference position without dogs is enabled for all axes.

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

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction
1 : Negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1201						ZCL	ZPI	ZPR

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed

0 : Not set automatically
1 : Set automatically

ZPI Coordinates at the reference position when a coordinate system is set automatically

0 : Value set in parameter No. 1250 is used.
1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

ZCL Local coordinate system when the manual reference position return is performed

0 : The local coordinate system is not canceled.
1 : The local coordinate system is canceled.

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

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

[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.

1251	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 axis

[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 ZPI in parameter 1201#1 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1300		LZR						

[Data type] Bit

LZR Checking of stored stroke limit 1 during the time from power–on to the manual position reference return

0: The stroke limit 1 is checked.

1: The stroke limit 1 is not checked

1425	FL rate of the reference position return for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

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

[Data type] Bit

OZR When manual reference position return is attempted in 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, B function) is being executed.
- When a dwell or cycle such as a canned cycle is being executed.

1821	Reference counter size for each axis

[Data type] Two–word axis

[Valid data range] 0 to 99999999

Set the size of the reference counter.

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.

1836	Servo error amount where reference position return is possible

[Data type] Byte axis

[Unit of data] Detection unit

[Valid data range] 0 to 127

This parameter sets a servo error used to enable reference position return in manual reference position return.

In general, set this parameter to 0. (When 0 is set, 128 is assumed as the default.)

> **WARNING**
> When bit 0 of parameter No. 2000 is set to 1, a value ten times greater than the value set in this parameter is used to make the check.
> Example:When the value 10 is set in this parameter, and bit 0 of parameter No. 2000 is set to 1, reference position return operation is enabled when a servo error of 100 or more occurs.

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A grid shift is set for each axis.

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

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

Alarm and message

Number	Message	Description
090	REFERENCE RETURN INCOMPLETE	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 parameter ZRNx (No. 1005#0) =0). Do reference position return.

Note

> **NOTE**
> P/S alarm No. 090 is issued when G28 is specified if the reference position has not yet be established.

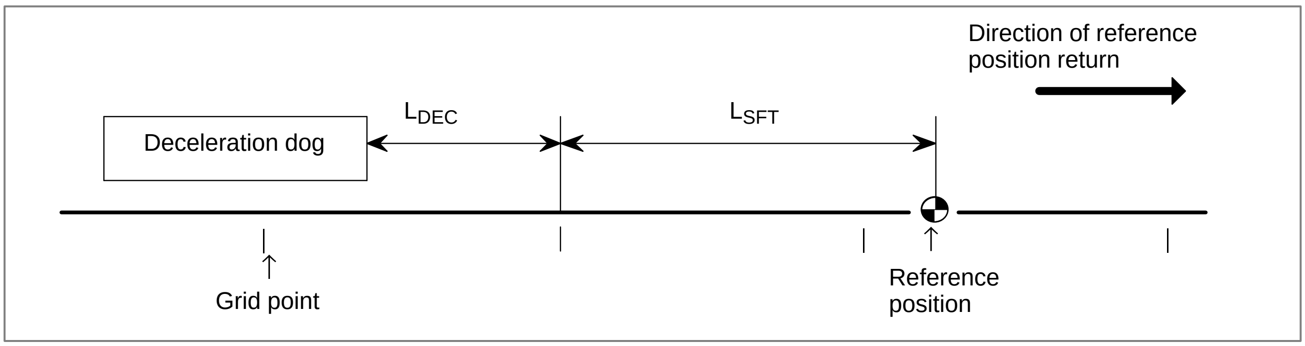
4.3 REFERENCE POSITION SHIFT

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 2 of parameter No. 1002 (SFD) to 1. When distance L_{SFT}, shown below, is set in parameter No. 1850, the reference position can be shifted.

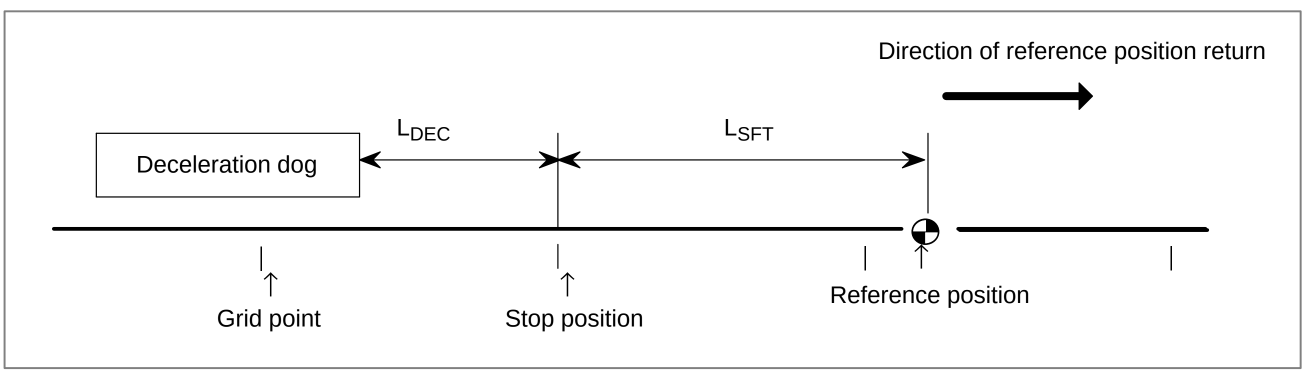
Distance L_{DEC}', shown below, for the axis along which reference position return was last made is indicated on the diagnostic screen (No. 0302).



L_{SFT}: Reference position shift amount
L_{DEC}: Distance from the position where the deceleration dog is turned off to the first grid point (grid point when the shift amount is 0)

- **How to adjust the reference position**

(1) Set the SFD bit (bit 2 of parameter No. 1002) to 1, and set the reference position shift amount to 0. Then, perform reference position return.

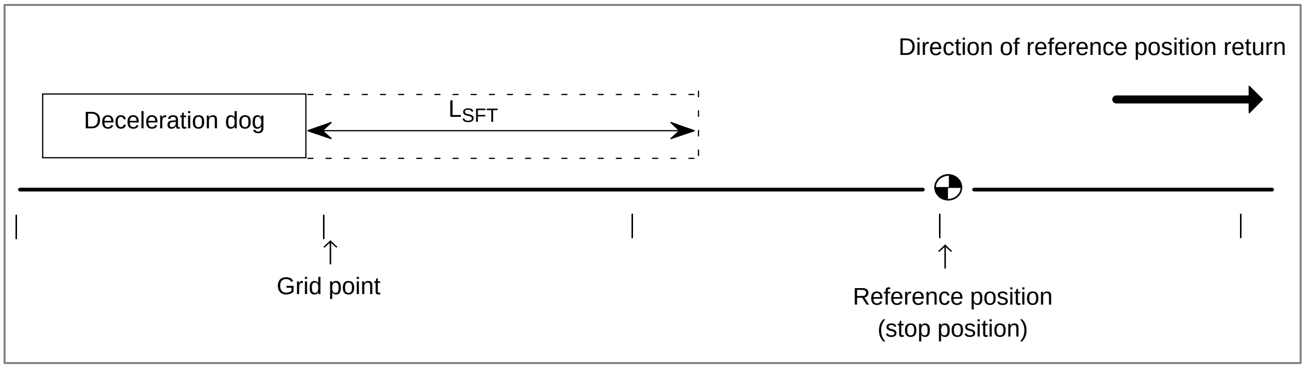


After the deceleration dog is turned off, the tool stops when the first grid point is reached. Distance L_{DEC} is indicated on the diagnostic screen (No. 0302).

(2) Determine the distance L_{SFT} (reference position shift amount) from the stop position to the reference position, and set it in parameter No. 1850.

This completes the adjustment of the reference position.

(3) Perform reference position return again. Then, the tool stops when it reaches the reference position.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1002						SFD		

[Data type] Bit

SFD The function for shifting the reference position is

0 : Not used
1 : Used

1850	Reference position shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] –99999999 to 99999999

A reference position shift is set for each axis.

CAUTION
When bit 2 of parameter No. 1002, SFD, is set to 0, this parameter is used for reference position shift.

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

Alarm and message

- **Diagnostic display**

0302	Distance from the position where the deceleration dog is turned off to the first grid point

[Data type] Two–word axis

[Unit of data] 0.001 mm (metric output), 0.0001 inch (inch output)

[Valid data range] –99999999 to 99999999

Note

NOTE

1 The reference position can be shifted only in the direction of reference position return.

2 When the SFD bit (bit 2 of parameter No. 1002) is 0, only the distance from the position where the deceleration dog is turned off to the first grid point (the grid point after grid shift) is indicated.

4.4 REFERENCE POSITION RETURN

General

The G28 command positions the tool to the reference position, via the specified intermediate point, along the specified axis, then sets the completion signal for reference position return (see Section 4.1) to 1.

The reference position must be set in parameter No. 1240 with the coordinates specified in the machine coordinate system, before issuing the G28 command.

The tool moves to the intermediate point or reference position at the rapid traverse rate.

Parameter

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

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction
1 : Negative direction

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

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

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

Alarm and message

Number	Message	Description
405	SERVO ALARM: (WRONG ZRN)	Position control system fault. Due to an CNC or servo system fault in the reference position return, there is 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 bit 5 (ZMIx) of parameter No. 1006. 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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.6	REFERENCE POSITION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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 No. 1241, 1242, or 1243 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 end signals ZP21 to ZP 28 <F096> Third reference position return end signals ZP31 to ZP38 <F098> Fourth reference position return end signals ZP41 to ZP48 <F100>

[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 appended to the end of a signal represents a controlled axis number, and a numeric character immediately following ZP represents a reference position number.

ZP 2 1

1 : Return end signal for the first axis
2 : Return end signal for the second axis
3 : Return end signal for the third axis
:
:

2 : Second reference position return
3 : Third reference position return
4 : Fourth reference position return

[Output condition] These signals turn to "1" when:

- The second, third, or fourth reference position return (G30) is completed, and the current position is in the in–position area.

These signals turn to "0" when:

- The tool moved from the reference position.
- An emergency stop is applied.
- A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
F098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
F100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41

Parameter

1241	Coordinate value of the second reference position on each axis in the machine coordinate system
1242	Coordinate value of the third reference position on each axis in the machine coordinate system
1243	Coordinate value of the fourth reference position on each axis in the machine coordinate system

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

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

Alarm and message

Number	Message	Description
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command. Correct program.

Caution

CAUTION

1 If the G30 command is issued in machine lock status, the completion signal for 2nd, 3rd, or 4th reference position return is not set to 1.

2 If millimeter input is selected for an inch–system machine, the completion signal for 2nd, 3rd, or 4th reference position return may be set to 1, even when the programmed tool position deviates from the 2nd, 3rd, or 4th reference position by the least input increment. This is because the least input increment is smaller than the least command increment for the machine.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.6	REFERENCE POSITION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.6	REFERENCE POSITION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.6	REFERENCE POSITION

4.6 FLOATING REFERENCE POSITION RETURN

General

It is possible to return the tool to the floating reference position by commanding the G30.1.

The floating reference position is located on the machine and can be a reference position of some sort of machine operation. It is not always a fixed position but may vary in some cases. The floating reference position can be set using the soft keys of MDI and can be memorized even if the power is turned off.

Generally, the position where the tools can be replace on machining center or milling machine is a set position on the machinery. The tools cannot be replaced at any position. Normally the tool change position is at any of the No. 1 to No. 4 reference position. The tool can be restored to these positions easily by G28 or G30 command. However, depending on the machine, the tools can be replaced at any position as long as it does not contact the workpiece.

In lathes, the tool can generally be changed at any position unless it touches the workpiece or tailstock.

For machinery such as these, in order to reduce the cycle time, it is advantageous to replace tools at a position as close as possible to the workpiece. For this purpose, change position should be changed for each workpiece and this feature can be easily realized by this function. Namely, the tool change position which is suitable for workpieces can be memorized as the floating reference position and it is possible to return the tool to the tool change position easily by commanding the G30.1.

When the G30.1 is commanded, the axis commanded goes to the specified intermediate position with rapid traverse at first and then goes to the floating reference position from the intermediate point with rapid traverse. The positioning to the intermediate position or to the floating reference position is performed at rapid traverse for each axis (non–linear positioning). The floating reference position return completion signal turns to “1” after completing the floating reference position return.

Signal

Floating reference position return end signal FRP1 to FRP8 <F116>

[Classification] Output signal

[Function] Notify the system that the tool is at the floating reference position on a controlled axis.

A floating reference position return end signal is provided for each axis. The number appended to each signal name indicates the number of the controlled axis.

FRP 1

1 : Floating reference position return end signal of 1st axis
2 : Floating reference position return end signal of 2nd axis
3 : Floating reference position return end signal of 3rd axis
:
:

[Output condition] The signals are set to 1 in the following case:

· When the tool is positioned to within the in–position area after floating reference position return (G30.1)

These signals are set to “0” when:

· The tool is moved from the floating reference position

· An emergency stop is applied.

· A servo alarm is raised.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F116	FRP8	FRP7	FRP6	FRP5	FRP4	FRP3	FRP2	FRP1

Parameter

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

[Data type] Bit

FPC When the floating reference position is specified using soft keys on the current position display screen

0 : The value of the displayed relative position is not preset. (In other words, the value does not change.)
1 : The value of the displayed relative position is preset to 0.

1244	Coodinates of the floating reference positon for each axis

[Data type] Two–word axis

[Unit of data]

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

[Valid data range] –99999999 to 99999999

This parameter specifies the coordinates of the floating reference position for each axis. The parameter is automatically set when the floating reference position is specified using soft keys on the current position display screen.

Reference Item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.6.2	FLOATING REFERENCE POSITION RETURN (G30.1)
		III.11.1.7	Setting the Floating Reference Position
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.6.2	FLOATING REFERENCE POSITION RETURN (G30.1)
		III.11.1.7	Setting the Floating Reference Position

4.7 BUTT–TYPE REFERENCE POSITION SETTING

General

This function automates the procedure of butting the tool against a mechanical stopper on an axis to set a reference position. The purpose of this function is to eliminate the variations in reference position setting that arise depending on the operator, and to minimize work required to make fine adjustments after reference position setting.

Select the axis for which the reference position is to be set, then perform cycle start. Then, the following operations are performed automatically:

1. The torque (force) of the selected axis is reduced to make the butting feedrate constant, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter–set distance from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter–set distance from the mechanical stopper.
3. The withdrawal point on the axis is set as the reference position.

Basic procedure for butt–type reference position setting

(1) First, set the parameters required for butt–type reference position setting.

ZMIx, bit 5 of parameter (No. 1006):	Direction of reference position setting
Parameter No. 7181:	Withdrawal distance
Parameter No. 7182:	Reference position setting distance
Parameter No. 7183:	Butting feedrate 1
Parameter No. 7184:	Butting feedrate 2
Parameter No. 7185:	Withdrawal feedrate in reference position setting
Parameter No. 7186:	Torque limit

(2) Select manual reference position return mode.

(3) By using a manual handle feed axis select signal, select the axis on which the reference position is to be set.

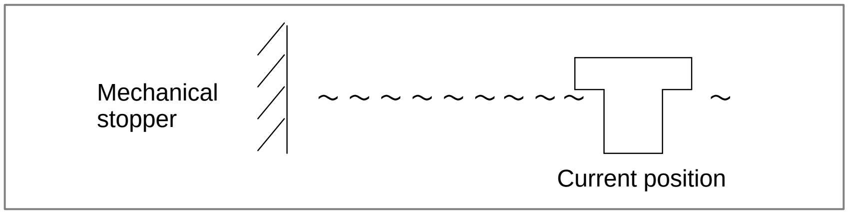
(4) Perform cycle start.

This starts the cycle operation for reference position setting.

(5) During the cycle operation, the automatic operation start signal OP is 1.

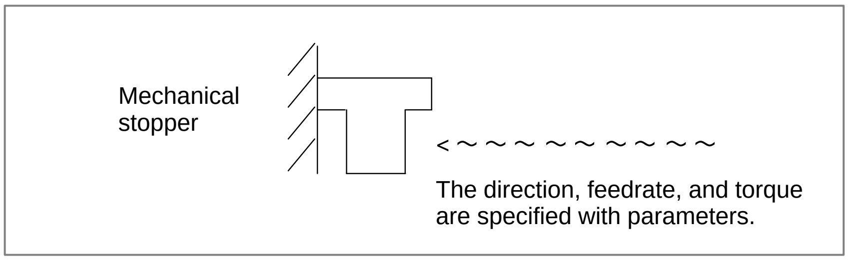
Cycle operation

When no reference position has been set (APZx, bit 4 of parameter No. 1815, is 0), operations (A) to (E), below, are performed automatically to set a reference position.



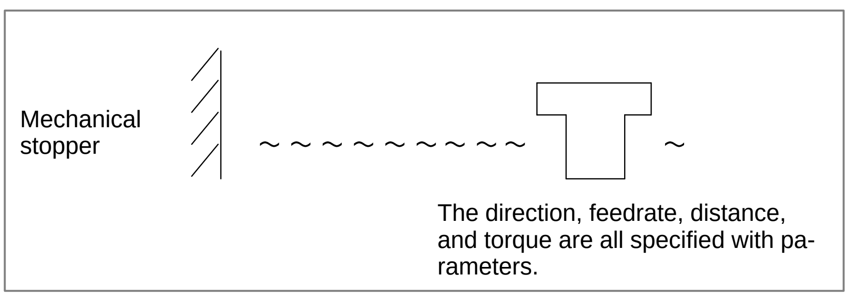
(A)The tool is moved along a specified axis with a limited torque until it butts against the mechanical stopper.

The tool is moved in the direction specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7183, at the torque specified with parameter No. 7186 (until the tool strikes the mechanical stopper).



(B)After the tool strikes the mechanical stopper, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter–set distance.

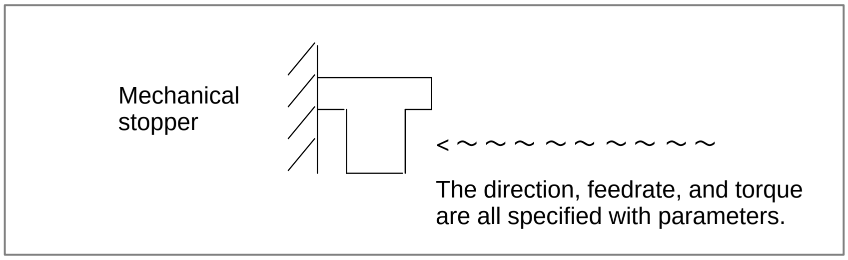
The tool is moved in the direction opposite to that specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7181.



(C)Operations (D) and (E) are performed from the withdrawal point, such that the tool is butted against the mechanical stopper at a constant feedrate in reference position setting.

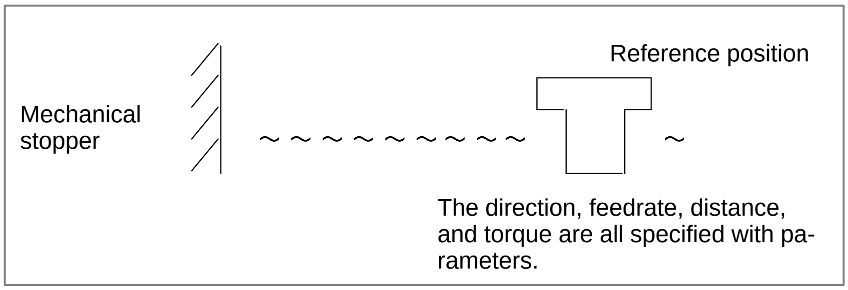
(D)The tool moves along the specified axis at a specified torque until it butts against the mechanical stopper.

The tool moves in the direction specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7184, at the torque specified with parameter No. 7186 (until the tool strikes the mechanical stopper).



(E) After the tool strikes the mechanical stopper end on the axis, the tool is withdrawn in the direction opposite to the butting direction, along the axis for a parameter–set distance.

The tool is moved in the direction opposite to that specified with ZMIx (bit 5 of parameter No. 1006), at the feedrate specified with parameter No. 7185, for the distance specified with parameter No. 7182.



For parameter Nos. 7183 and 7184, set the feedrates at which the tool is moved toward the mechanical stopper with a limited torque, considering the machine accuracy.

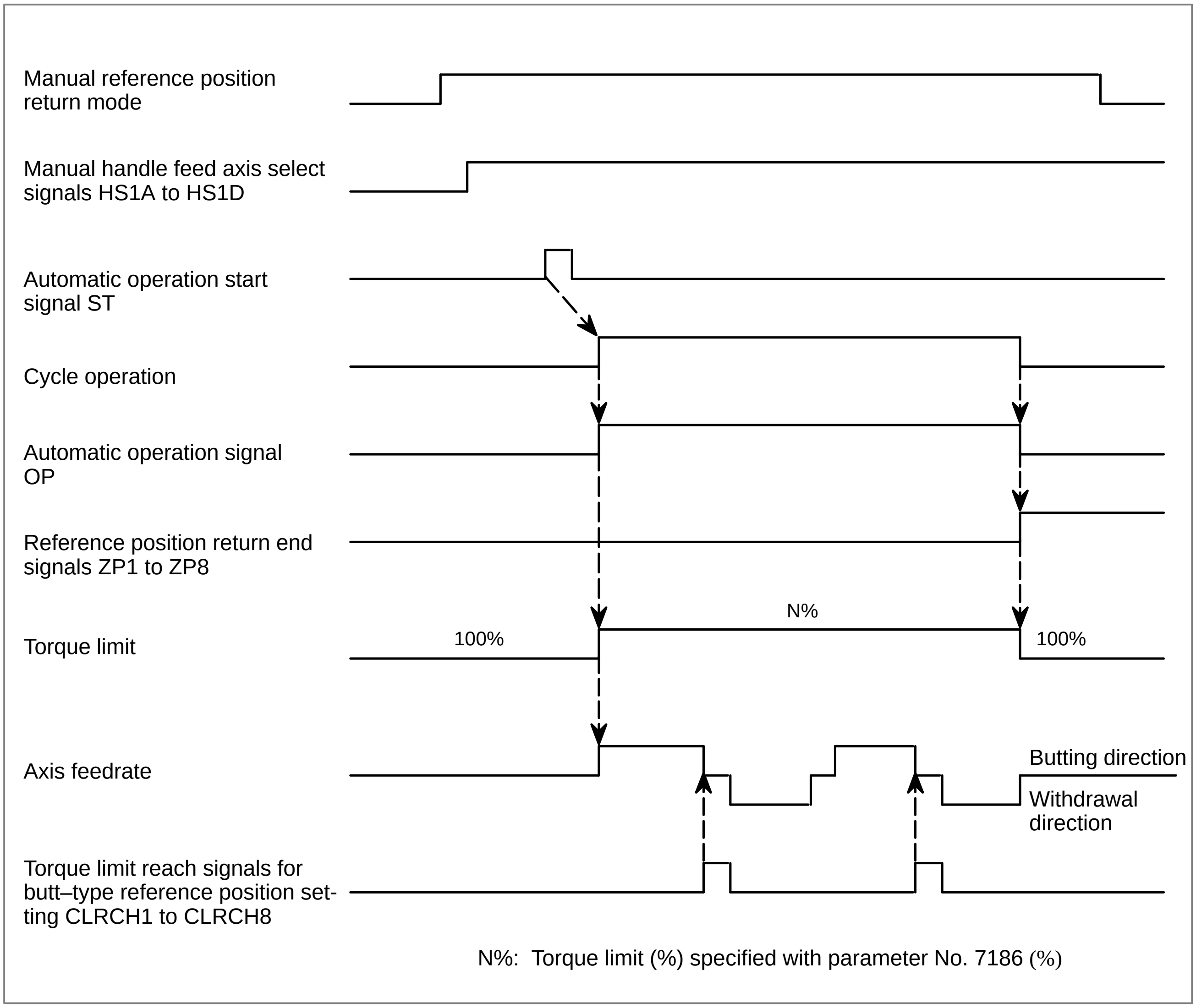
After the tool strikes the mechanical stopper, and the tool is withdrawn the distance specified with parameter No. 7182, the withdrawal point is set as the reference position on the specified axis. Then, the reference position return end signal and reference position establishment signal are set to 1.

After reference position return has been completed, alarm PS000 is issued. Turn the power off then back on before continuing operation.

After the reference position is set

When the reference position has already been set (when APZx, bit 4 of parameter No. 1815, is 1), performing butt–type reference position setting causes the tool to be positioned to the reference position at the rapid traverse rate without the cycle operation. Upon the completion of positioning, the reference position return end signal is set to 1.

The timing chart for the cycle operation is shown below.



Signal

Torque limit reach signals for butt–type reference position setting CLRCH1 to CLRCH8 <F180>

[Classification] Output signal

[Function] These signals are used to post notification of the torque limit having been reached for each corresponding axis during cycle operation for butt–type reference position setting.

[Operation] Each signal is set to 1 when:

· The torque limit is reached for the corresponding axis during cycle operation for butt–type reference position setting.

Each signal is set to 0 when:

· The torque limit is not reached for the corresponding axis during cycle operation for butt–type reference position setting.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F180	CLRCH8	CLRCH7	CLRCH6	CLRCH5	CLRCH4	CLRCH3	CLRCH2	CLRCH1

Parameter

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

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on.

0 : Positive direction
1 : Negative direction

7181	First withdrawal distance in butt–type reference position setting

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch

[Valid data range] –99999999 to 99999999

When the butt–type reference position setting function is used, this parameter sets a distance an axis, along which withdrawal is performed after the mechanical stopper is hit (distance from the mechanical stopper to the withdrawal point).

NOTE
Set the same direction as the direction set for ZMIx, bit 5 of parameter No. 1006. If the opposite direction is set, the cycle operation will not start.

7182	Second withdrawal distance in butt–type reference position setting

[Data type] 2–word axis

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Millimeter machine	0.01	0.001	0.0001	mm
Inch machine	0.001	0.0001	0.00001	inch

NOTE
Set the same direction as the direction set for ZMIx, bit 5 of parameter No. 1006. If the opposite direction is set, the cycle operation will not start.

[Valid data range] –99999999 to 99999999

When the butt–type reference position setting function is used, this parameter sets a distance an axis, along which withdrawal is performed after the mechanical stopper is hit (distance from the mechanical stopper to the withdrawal point).

7183	First butting feedrate in butt–type reference position setting

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate first used to hit the stopper on an axis.

7184	Second butting feedrate in butt–type reference position setting

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate used to hit the stopper on an axis for a second time.

7185	Withdrawal feedrate (common to the first and second butting operations) in butt–type reference position setting)

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30–15000	30–12000
Inch machine	0.1 inch/min	30–6000	30–4800

When the butt–type reference position setting function is used, this parameter sets the feedrate used for withdrawal along an axis after the mechanical stopper has been hit.

7186	Torque limit value in butt–type reference position setting

[Data type] Byte axes

[Unit of data] %

[Valid data range] 0 to 100

This parameter sets a torque limit value in butt–type reference position setting.

> **NOTE**
> When 0 is set in this parameter, 100% is assumed.

Alarm and message

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.

Note

> **NOTE**
> This function is supported only when an absolute–position detector is installed.

4.8 LINEAR SCALE WITH ABSOLUTE ADDRESSING REFERENCE MARKS

General

The linear scale with absolute addressing reference marks has reference marks (one–rotation signals) at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. The CNC makes a small movement along an axis to measure the one–rotation signal interval, then calculates the absolute position. The reference position can be established without performing positioning to the reference position.

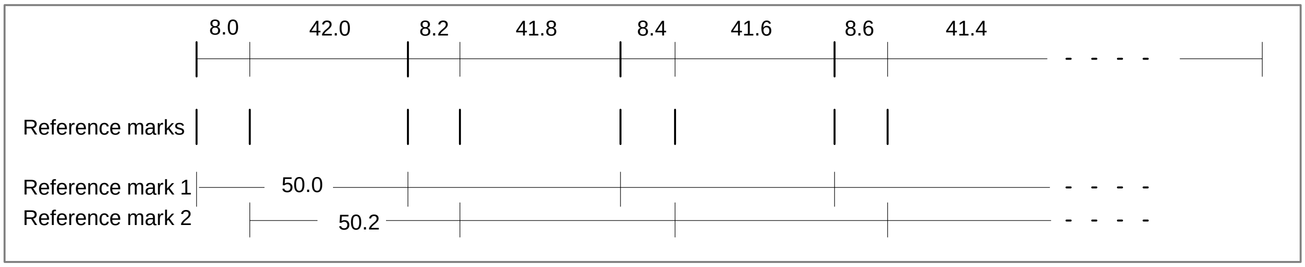


Fig. 4.8 (a) Sample linear scale with absolute addressing reference marks

Basic procedure for establishing the reference position

(1) Select JOG mode, then set manual reference position return selection signal ZRN to 1.

(2) Set the feed axis direction selection signal (+J1, –J1, +J2, –J2, etc.), corresponding to the axis for which the reference position is to be established, to 1, then feed the tool along that axis.

(3) The tool is fed along the selected axis at the reference position return FL feedrate (parameter No. 1425).

(4) Upon the detection of a reference mark sent from the linear scale, the tool is stopped briefly, then fed again at the reference position return FL feedrate.

(5) Step (4) is repeated until three or four reference marks are detected. Then, the absolute position is calculated and the reference position established signal (ZRF1, ZRF2, ZRF3, etc.) is set to 1.

If the feed axis direction selection signal (+J1, –J1, +J2, –J2, etc.) is set to 0 at any point in steps 2 to 5, feed does not stop and establishment of the reference position is continued.
A time chart for the above basic procedure is shown below.

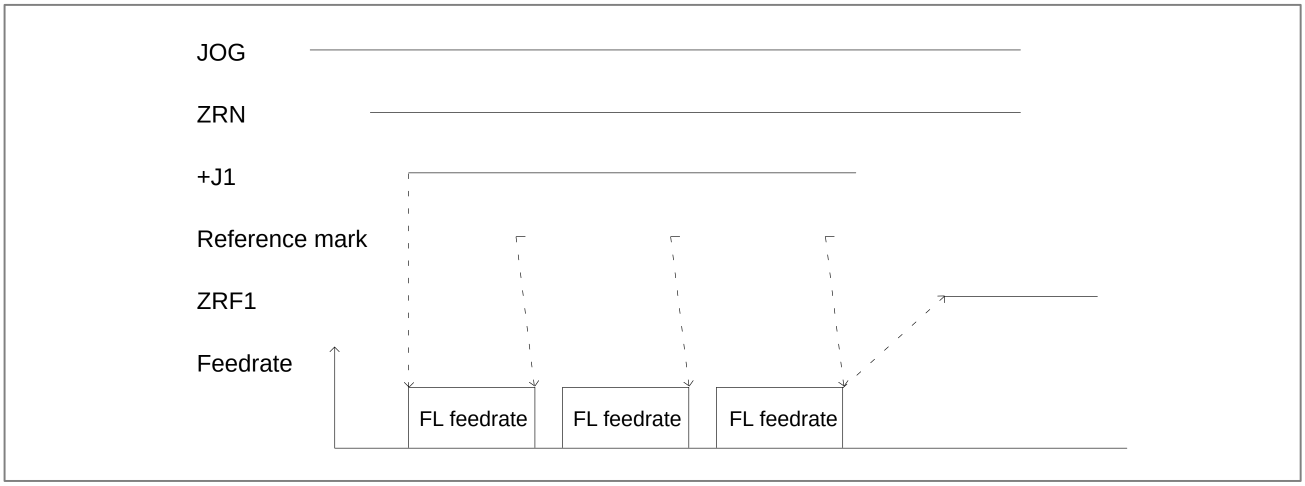


Fig. 4.8 (b) Time chart for reference position establishment

Parameter

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

[Data type] Bit axis

OPTx As a position detector:

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

(To use a linear scale with absolute addressing reference marks, set this bit to 1.)

DCLx As a separate position detector:

0 : A linear scale with absolute addressing reference marks is not used.
1 : A linear scale with absolute addressing reference marks is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1802							DC4	

[Data type] Bit

DC4 When establishing the reference position using the linear scale with absolute addressing reference marks:

0 : Three reference marks are detected to determine the absolute position.
1 : Four reference marks are detected to determine the absolute position.

1821	Reference counter capacity for each axis

[Data type] 2–word axis

[Unit of data] Detection units

[Valid data range] 0 to 99999999

Sets the intervals of mark 1 of the linear scale with absolute addressing reference marks.

1882	Intervals of mark 2 of the linear scale with absolute addressing reference marks.

[Data type] 2–word axis

[Unit of data] Detection units

[Valid data range] 0 to 99999999

Sets the intervals of mark 2 of the linear scale with absolute addressing reference marks.

1883	Distance between the reference position and the origin of the linear scale with absolute addressing reference marks

[Data type] 2–word axis

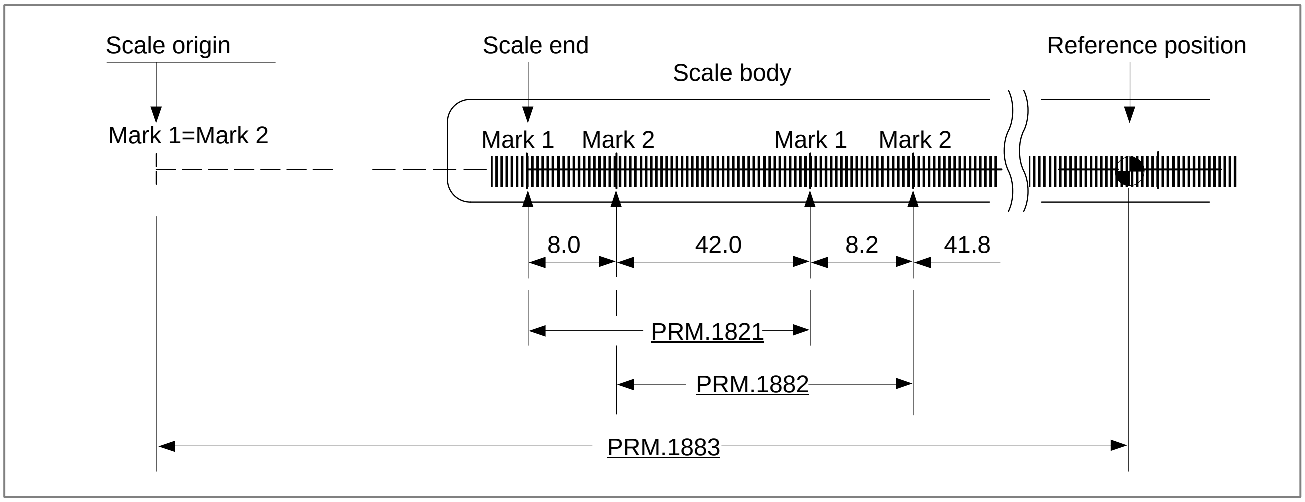
[Unit of data] Detection units

[Valid data range] –99999999 to 99999999

Sets the distance between the reference position and the origin of the linear scale with absolute addressing reference marks. The origin of the scale is that point where marks 1 and 2 coincide.

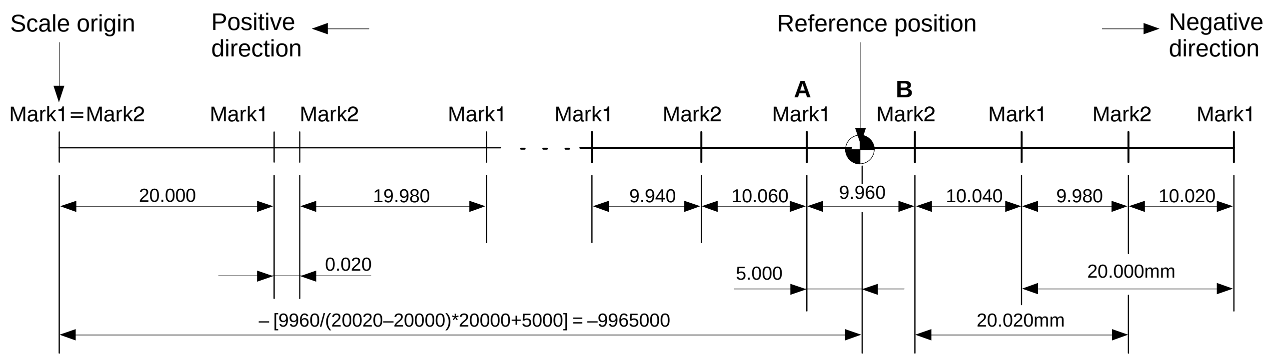
Usually, the origin is an imaginary point which does not physically exist (see the figure below).

If the reference position exists in the positive direction as viewed from the scale origin, specify a positive value for this parameter.
If the reference position exists in the negative direction as viewed from the scale origin, specify a negative value for this parameter.



[Example parameter setting]

When the following scale is used with an IS–B, millimeter machine



Parameter No. 1821 (mark 1 interval) = "20000"

No. 1882 (mark 2 interval) = "20020"

No. 1883 (reference position) = Position of point A +5.000

$$= \frac{\text{Distance between A and B}}{\text{Mark 2 interval – Mark 1 interval}} \text{ * Mark 1 interval+5.000}$$

$$= \frac{9960}{20020–20000} \text{ * 20000+5000}$$

= 9965000

→ "–9965000"(reference position in the negative direction)

[Method of setting parameter No. 1883]

If it is difficult to measure the distance between the scale origin and reference position (parameter No. 1883), the parameter can be set as follows:

(1) Set parameter No. 1815 to enable this function.
Set parameters No. 1821 and 1882 appropriately.
Set parameter No. 1240 to 0.
Set parameter No. 1883 to 0.

(2) At an appropriate position, establish the reference position as described in "Basic procedure for establishing the reference position" (as a result, the machine coordinate becomes the distance between the scale origin and current position).

(3) Position the machine to the reference position, by jog feed or handle feed.

(4) Convert the current machine coordinate (displayed with No. 301 on the diagnosis screen) to a value in detection units (by multiplying the value of No. 301 by CMR), then set the result for parameter No. 1883.

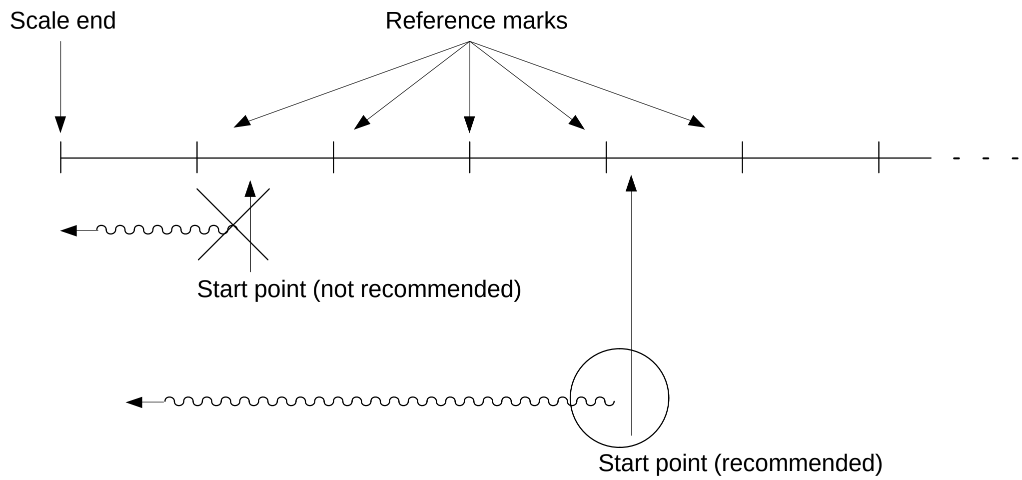
(5) If necessary, set parameter No. 1240.

Caution

CAUTION

1 In the following cases, the machine moves to the reference position without establishing the reference position as described above:
- Axial movement is performed in REF mode when the reference position has already been established.
- Automatic reference position return (G28) is specified when the reference position has already been established.

2 Axial movement is continued until three or four reference marks are detected. If the establishment of the reference position is started at a point near the end of the scale, therefore, axial movement does not stop until overtravel occurs. Start establishing the reference position at a point sufficiently distant from the end of the scale.



Note

NOTE

1 If automatic reference position return (G28) is specified when the reference position has not been established, the reference position is first established as described above, after which the machine is positioned to the reference position.

2 A P/S alarm (No. 090) occurs in the following case:
   - The actual reference mark interval differs from that specified with the parameter.

3 This function cannot be used for axes that are subjected to any of the following functions:
   - Simple synchronous control
   - Angular axis control
   - Straightness deviation compensation

4 This function is invalid if any of the following conditions are satisfied:
   - Either parameter No. 1821 (mark 1 interval) or No. 1882 (mark 2 interval) is set to 0.
   - Parameters No. 1821 and No. 1882 are set to the same value.
   - The value of parameter No. 1821 is more than twice as large as that of parameter No. 1882, or vice versa.

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 (MEM) mode, DNC operation mode (RMT), or manual data input (MDI) mode, the CNC enters the automatic operation start state then starts operating.

Signal ST, however, is ignored in the following cases:

1. When the mode is other than MEM, RMT, or MDI
2. When the feed hold signal (*SP) is set to 0
3. When the emergency stop signal (*ESP) is set to 0
4. When the external reset signal (ERS) is set to 1
5. When the reset and rewind signal (RRW) is set to 1
6. When MDI RESET key is pressed
7. When the CNC is in the alarm state
8. When the CNC is in the NOT READY state
9. When automatic operation is 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 operation mode (JOG, INC, HND, REF, TJOG, or THND).

The CNC enters the automatic operation stop state and stops operating in the following cases during automatic operation:

1. When a single command block is completed during a single block operation
2. When operation in manual data input (MDI) mode has been completed
3. When an alarm occurs in the CNC
4. When a single command block is completed after the mode is changed to other automatic operation mode or memory edit (EDIT)

The CNC enters the reset state and stops operating in the following cases during automatic operation:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

The state of the CNC (automatic operation start, feed hold , automatic operation stop, or reset) is posted to the PMC with status output signals OP, SPL, and STL. See the table in the “Signal” section for details.

- **Halt of automatic operation (feed hold)**

When the feed hold signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. At the same time, cycle start lamp signal STL is set to 0 and feed hold lamp signal SPL is set to 1. Re–setting signal *SP to 1 in itself will not restart automatic operation. To restart automatic operation, first set signal *SP to 1, then set signal ST to 1 and then to 0.

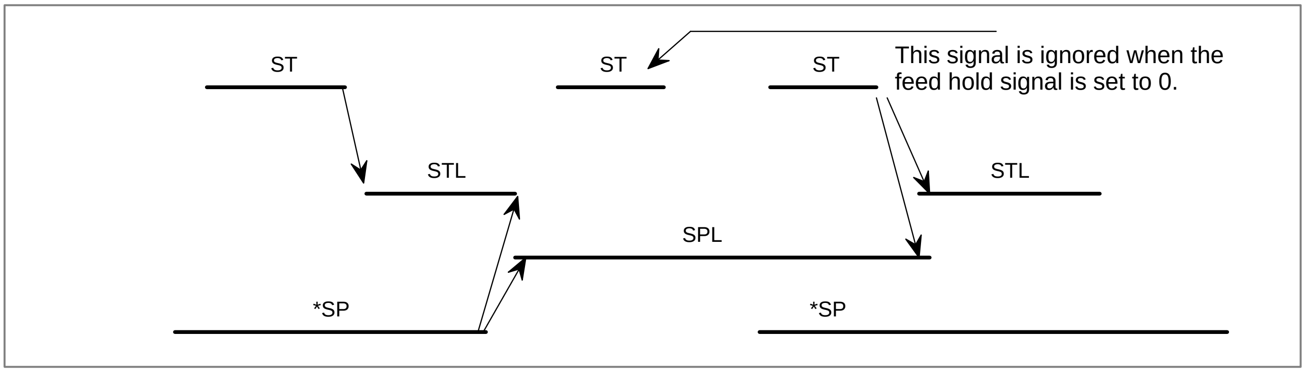


Fig. 5.1 Time chart for automatic operation

When signal *SP is set to 0 during the execution of a block containing only the M, S, T, or B function, signal STL is immediately set to 0, signal SPL is set to 1, and the CNC enters the feed hold state. If the FIN signal is subsequently sent from the PMC, the CNC executes processing up until the end of the block that has been halted. Upon the completion of that block, signal SPL is set to 0 (signal STL remains set to 0) and the CNC enters the automatic operation stop state.

(a) During threading
When signal *SP is set to 0 during threading, the CNC enters the feed hold state after executing a non–threading block after the threading blocks.
When signal *SP is set to 0 during threading with the G92 command (threading cycle), signal SPL is immediately set to 1 but operation continues up until the end of the retraction block following threading.
When signal *SP is set to 0 during threading with the G32 (M series: G33) command, signal SPL is immediately set to 1 but operation continues until the end of a non–threading block following the threading blocks. (Stopping feeding during threading is dangerous because the amount of cutting will increase.)

(b) During tapping in a 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.

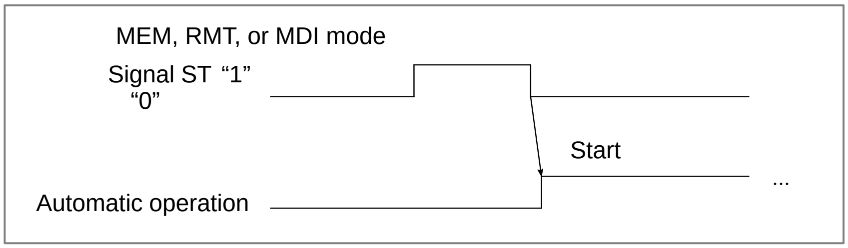
Signal

Cycle start signal ST <G007#2>

[Classification] Input signal

[Function] Starts automatic operation.

[Operation] When signal ST is set to 1 then 0 in memory (MEM) mode, DNC operation mode (RMT) or manual data input (MDI) mode, the CNC enters the cycle start state and starts operation.

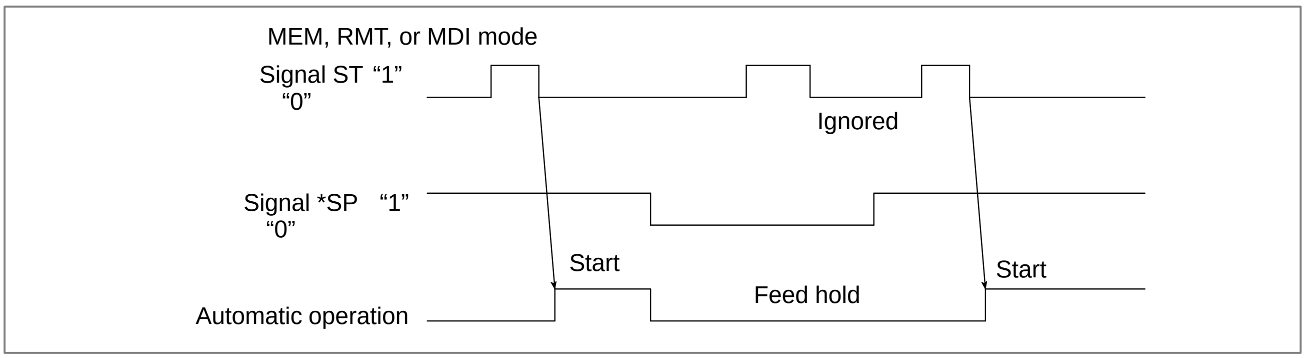


Feed hold signal *SP <G008#5>

[Classification] Input signal

[Function] Halts automatic operation.

[Operation] When signal *SP is set to 0 during automatic operation, the CNC enters the feed hold state and stops operation. Automatic operation cannot be started when signal *SP is set to 0.



Automatic operation signal OP <F000#7>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation is in progress.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Cycle start lamp signal STL <F000#5>

[Classification] Output signal

[Function] Notifies the PMC that automatic operation start is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Feed hold lamp signal SPL <F000#4>

[Classification] Output signal

[Function] Notifies the PMC that feed hold state is entered.

[Output condition] This signal is set to 1 or 0, according to the state of the CNC, as listed in Table 5.1.

Table 5.1 Status of Operation

Signal name / State of the operation	Cycle start lamp STL	Feed hold lamp SPL	Automatic operation lamp OP
Cycle start state	1	0	1
Feed hold state	0	1	1
Automatic operation stop state	0	0	1
Reset state	0	0	0

· Cycle start state
The CNC is executing memory operation or manual data input operation commands.

· Feed hold state
The CNC is not executing memory operation nor manual data input operation commands while the commands to be executed remain.

· Automatic operation stop state
Memory operation or manual data input operation has been completed and stopped.

· Reset state
The automatic operation has been forcibly terminated.

> **NOTE**
> If the sequence number search is performed through MDI panel during Memory mode (MEM), the signal OP turns to “1”.

- **Signal address**

	#7	#6	#5	#4	#3	#2	#1	#0
G007						ST		
G008			*SP					

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

Alarm and message

- **Self–diagnosis information**

During automatic operation, the machine may sometimes show no movement while no alarm is detected. In that case, the CNC may be performing processing or waiting for the occurrence of an event. The state of the CNC can be obtained using the CNC self–diagnosis function (diagnosis numbers 000 to 015).

Detailed information on the automatic operation stop or feed hold state can also be displayed (diagnosis numbers 020 to 025).

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.9	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.3	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.4.1	MEMORY OPERATION
		III.4.2	MDI OPERATION
		III.4.9	DNC OPERATION
		III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

5.2 RESET AND REWIND

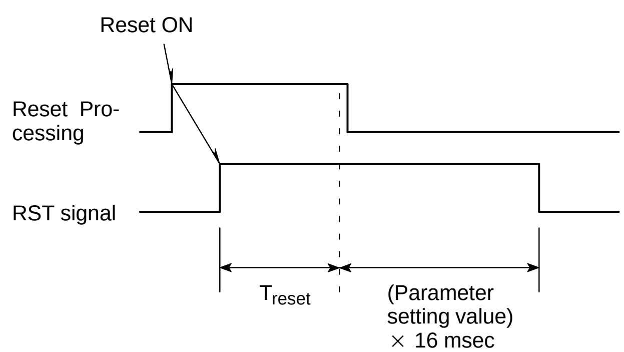
General

The CNC is reset and enters the reset state in the following cases:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset and rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

When the CNC is reset, the resetting signal (RST) is output to the PMC. The resetting signal (RST) is set to 0 when the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released.

RST signal output time = T_{reset} (Reset processing time) + (parameter setting value) $\times$ 16 msec.



CAUTION
T_{reset} requires at least 16 msec. This time will be longer on optional configurations.

When the CNC is reset during automatic operation, automatic operation is stopped and tool movement along the controlled axis is decelerated and stopped(*1). When the CNC is reset during the execution of the M, S, T, or B function, signal MF, SF, TF, or BF is set to 0 within 100 ms.

Tool movement along the controlled axis is also decelerated and stopped(*1) in manual operation (jog feed, manual handle feed, incremental feed, or etc).

CAUTION
*1 When the emergency stop signal (*ESP) is set to 0, the tool is stopped by an emergency stop.

Bit 6 (CLR) of parameter No. 3402 is used to select whether the CNC internal data (such as modal G codes) is cleared or reset when the CNC is reset. Refer to the Appendix E, ”Status when turning on power, when cleared, and when reset” in the Operator’s manual for the state of the internal data when cleared or reset.

The following parameters are also used to select how to handle processing for CNC data when the CNC is reset.

- Bit 7 (MCL) of parameter No. 3203
  Whether programs created in MDI mode are erased or stored
- Bit 6 (CCV) of parameter No. 6001
  Whether custom macro variables #100 to #149 are cleared or stored
- Bit 7 (CLV) of parameter No. 6001
  Whether custom macro local variables #1 to #33 are cleared or stored

- **Reset & Rewind**

When the reset & rewind signal (RRW) is set to 1, reset is performed and the following rewinding operation is also performed.

1. When the DNC operation mode, and a portable tape reader is connected as the current input/output device, the tape reader is rewound.

   While the tape reader is being rewound, the rewinding–in–progress signal (RWD) is output. This signal goes 0 when the tape reader has been rewound.
2. In cases other than case 1, the head of the selected main program is searched for. Setting RWM, bit 2 of parameter no. 3001, determines whether the rewinding–in–progress signal is output.

   When RWM is set to 1:

   The rewinding–in–progress signal is output. It is set to 1, then set to 0 after about 100 ms. Since searching for the main program in memory takes little time, when the rewinding–in–progress signal (RWD) is set to 0, the main program has already been searched for.

Signal

External reset signal ERS<G008#7>

[Classification] Input signal

[Function] Reset the CNC.

[Operation] Turning the signal ERS to 1 resets the CNC and enters the reset state. While the CNC is reset, the resetting signal RST turns to 1.

Reset & rewind signal RRW<G008#6>

[Classification] Input signal

[Function] CNC is reset and a program under an automatic operation is rewound.

[Operation] As described in the item, “Reset & Rewind”.

Resetting signal RST <F001#1>

[Classification] Output signal

[Function] Notifies the PMC that the CNC is being reset. This signal is used for reset processing on the PMC.

[Output condition] This signal is set to 1 in the following cases:

1. When the emergency stop signal (*ESP) is set to 0
2. When the external reset signal (ERS) is set to 1
3. When the reset & rewind signal (RRW) is set to 1
4. When MDI RESET key is pressed

This signal is set to 0 in the following case:
When the resetting signal output time, set with parameter No. 3017, has elapsed after the above conditions have been released and the CNC is reset

Rewinding signal RWD <F000#0>

[Classification] Output signal

[Function] Notifies the PMC that the CNC is being rewound.

[Output condition] As described in the item, “Reset and Rewind”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G008	ERS	RRW						

	#7	#6	#5	#4	#3	#2	#1	#0
F000								RWD
F001							RST	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001						RWM		

[Data type] Bit

RWM RWD signal indicating that rewinding is in progress

0 : Output only when the tape reader is being rewound by the reset and rewind signal RRW

1 : Output when the tape reader is being rewound or a program in memory is being rewound by the reset and rewind signal RRW

3017	Output time of reset signal RST

[Data type] Byte

[Unit of data] 16 ms

[Valid data range] 0 to 255

To extend the output time of reset signal RST, the time to be added is specified in this parameter.
RST signal output time = time required for reset + parameter value × 16 ms

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL							

[Data type] Bit

MCL Whether a program prepared in the MDI mode is cleared by reset

0: Not deleted
1: deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR						

[Data type] Bit

CLR Reset key on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal

0: Cause reset state.
1: Cause clear state.

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

[Data type] Bit

CCV Custom macro’s common variables Nos. 100 through 149

0: Cleared to “vacant” by reset
1: Not cleared by reset

CLV Custom macro’s local variables Nos. 1 through 33

0: Cleared to “vacant” by reset
1: Not cleared by reset

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	APPEN-DIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	APPEN-DIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	APPEN-DIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	APPEN-DIX 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, or each–axis machine lock signals MLK1 to MLK8 are set to 1, output pulses (move commands) to the servo motors are stopped in manual or automatic operation. The commands are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the commands are correct by monitoring the position display.

Signal

All–axis machine lock signal MLK <G044#1>

[Classification] Input signal

[Function] Places all controlled axes in the machine lock state.

[Operation] When this signal is set to 1, pulses (move commands) are not output to the servo motors for all axes in manual or automatic operation.

Each–axis machine lock signals MLK1 to MLK8 <G108>

[Classification] Input signal

[Function] Place the corresponding controlled axes in the machine lock state. These signals are provided for each controlled axis. The signal number corresponds to the number of the controlled axis.

MLK 1

1. Machine lock for the 1st axis
2. Machine lock for the 2nd axis
3. Machine lock for the 3rd axis

⋮

[Operation] When these signals are set to 1, pulses (move commands) are not output to the servo motors for the corresponding axes (1st to 8th) in manual or automatic operation.

All–axis machine lock check signal MMLK <F004#1>

[Classification] Output signal

[Function] Notifies the PMC of the state of the all–axis machine lock signal.

[Output condition] This signal is set to 1 in the following case:
– When all–axis machine lock signal MLK is set to 1

This signal is set to 0 in the following case:
– When all–axis machine lock signal MLK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G044							MLK	
G108	MLK8	MLK7	MLK6	MLK5	MLK4	MLK3	MLK2	MLK1
F004							MMLK	

Note

NOTE

1 **Automatic operation in the machine lock state (M, S, T, and B commands)**
Machine lock applies only to move commands along controlled axes. Updating modal G codes or setting a coordinate system is performed normally. M, S, T, and B (2nd auxilialy function) commands are also performed normally.

2 **Reference position return in the machine lock state (G27, G28, and G30)**
When the reference position return command (G28), or 2nd to 4th reference position return command (G30), is executed for an axis in the machine lock state, distribution and position updating are performed. The tool, however, is not returned to the reference position. The reference position return completion signals (ZP1 to ZP4) are not output.
The reference position return check command (G27) is ignored in the machine lock state.

3 **Turning on/off the machine lock signal during movement along an axis**
When the machine lock signal for an axis is set to 1 during movement along the axis that is not in the machine lock state, the axis is immediately placed in the machine lock state and output pulses (move commands) to the servo motor are stopped. The tool is decelerated and stopped with the automatic acceleration/deceleration function.
On the other hand, when the machine lock signal for an axis is set to 0 during distribution of the move command along the axis in the machine lock state, pulse (move command) output for the axis is immediately restarted. The tool is accelerated with the automatic acceleration/deceleration function.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK

5.3.2 Dry Run

General

Dry run is valid only for automatic operation.
The tool is moved at a constant feedrate(*1) regardless of the feedrate specified in the program. This function is used, for example, to check the movement of the tool without a workpiece.

CAUTION
This feedrate depends on the specified parameters, the manual rapid traverse switching signal (RT), manual feedrate override signals (*JV0 to *JV15), and whether the command block specifies rapid traverse or cutting feed, as listed in the table below.

Manual rapid traverse switching signal (RT)	Program command	
	Rapid traverse	Feed
1	Rapid traverse rate	Dry run feedrate × JV_{max} *2
0	Dry run speed × JV,or rapid traverse rate *1	Dry run feedrate × JV *2

Max. cutting feedrate Setting by parameter No.1422
Rapid traverse rate Setting by parameter No.1420
Dry run feedrate Setting by parameter No.1410
JV Manual feedrate override
JV_{max} Maximum value of manual feedrate override
*1:Dry run feedrate × JV when parameter RDR (bit 6 of No. 1401) is 1.
Rapid traverse rate when parameter RDR is 0.
*2 Clamped by max. cutting feedrate.

Signal

Dry run signal DRN <G046#7>

[Classification] Input signal

[Function] Enables dry run.

[Operation] When this signal is set to 1, the tool is moved at the feedrate specified for dry run.
When this signal is set to 0, the tool is moved normally.

CAUTION
When the dry run signal is changed from 0 to 1 or 1 to 0 during the movement of the tool, the feedrate of the tool is first decelerated to 0 before being accelerated to the specified feedrate.

Dry run check signal MDRN <F002#7>

[Classification] Output signal

[Function] Notifies the PMC of the state of the dry run signal.

[Output condition] This signal is set to 1 in the following case:
– When dry run signal DRN is set to 1

This signal is set to 0 in the following case:
– When dry run signal DRN is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046	DRN							
F002	MDRN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR					

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84; rigid tapping)
0 : Enabled
1 : Disabled

RDR Dry run for rapid traverse command
0 : Disabled
1 : Enabled

1410	Dry run rate

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the dry run rate when the manual feedrate is overridden by 100%.

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	6 – 100000
Inch machine	0.1 inch/min	30 – 96000	6 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

> **NOTE**
> To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.5.4	Dry run
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.5.4	Dry run
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.5.4	Dry run
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.5.4	Dry run

5.3.3 Single Block

General

Single block operation is valid only for automatic operation.
When the single block signal (SBK) is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal (SBK) is set to 0, normal automatic operation is restored.

Single block operation during the execution of custom macro statements depends on the setting of bit 5 (SBM) of parameter No. 6000, as follows:

SBM = 0: Operation does not stop in the custom macro statements but stops once the next NC command has been executed.

SBM = 1: Operation stops after each block in the custom macro statements.

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 (INC), or jog feed (JOG), by using the mode select signals (MD1, MD2, and MD4).

Signal

Single block signal SBK <G046#1>

[Classification] Input signal

[Function] Enables single block operation.

[Operation] When this signal is set to 1, single block operation is performed. When this signal is set to 0, normal operation is performed.

Single block check signal MSBK <F004#3>

[Classification] Output signal

[Function] Notifies the PMC of the state of the single block signal.

[Output condition] This signal is set to 1 in the following case:
– When single block signal SBK is set to 1

This signal is set to 0 in the following case:
– When single block signal SBK is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046							SBK	
F004					MSBK			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					

[Data type] Bit

SBM Custom macro statement

0: Not stop the single block

1: Stops the single block

> **NOTE**
> SBM is valid when NOP (bit 0 of parameter No. 3404) is set to 0.

Caution

> **CAUTION**
>
> 1 **Operation in thread cutting**
> When the SBK signal turns to "1" during thread cutting, operation stops after execution of the first non-thread cutting block after the thread cutting command.
>
> 2 **Operation in canned cycle**
> When the SBK signal turns to "1" during canned cycle operation, the operation stops at each positioning, approach, drilling and retreat instead of the end of the block. The SPL signal turns to "1" while the STL signal turns to "0", showing that the end of the block has not been reached. When the execution of one block is completed, the STL and SPL signals turn to "0" and the operation is stopped.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.5.5	Single block
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.5.5	Single block
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.5.5	Single block
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.5.5	Single block

5.4 MANUAL ABSOLUTE ON/OFF

General

This function selects whether the movement of the tool with manual operation (such as jog feed and manual handle feed) is counted for calculating the current position in the workpiece coordinate system. A check signal is also output to indicate whether the manual absolute function in the CNC is turned on or off.

When manual absolute turns on (manual absolute signal *ABSM =0)

When manual operation interrupts during automatic operation:

i) At the end of the block where manual operation interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command.

ii) In subsequent blocks, the parallel-moved tool position remains unchanged until an absolute command block appears. Therefore, if all blocks are programmed by incremental commands, the tool keeps the parallel-moved position until machining ends.

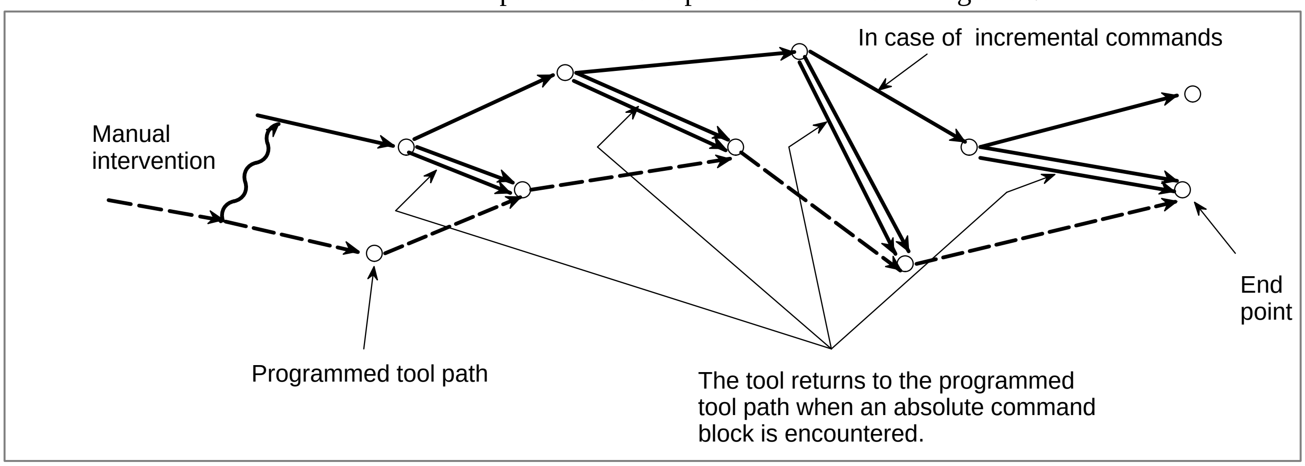


Fig. 5.4 (a) Manual absolute ON

> **CAUTION**
> If the machining end position has shifted by the manual move amount because all blocks are programmed by incremental commands only, the present position is displayed shifted by the manual move amount.

When manual absolute turns off (manual absolute signal *ABSM=1)

The manual move amount is not counted to the present position on the workpiece coordinate system. The present position display on the CRT includes the manual move amount. The display is reset to the initial value (before manual operation) when the control is reset, or when operation in the automatic operation mode MEM, RMT, or MDI is started after the manual operation.

During automatic operation, if manual intervention of a block interrupts, the tool position moves in parallel by the manual move amount, regardless of the absolute or incremental command at the end point of that block, as well as at the end point of subsequent blocks.

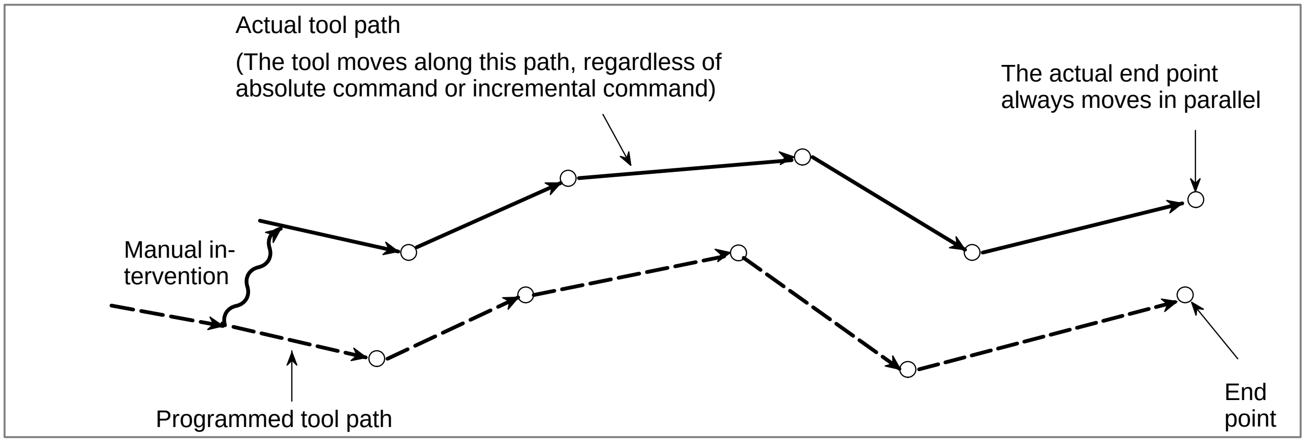


Fig. 5.4 (b) Manual absolute OFF

The present position display at the finish of the operation shows an end point value on the program as if manual intervention had not been executed. However, the tool position moves in parallel.

Signal

Manual absolute signal *ABSM <G006#2>

[Classification] Input signal

[Function] Turns the manual absolute function on or off.

[Operation] When this signal is set to 1, the control unit operates as follows:
– Turns off the manual absolute function.

When this signal is set to 0, the control unit operates as follows:
– Turns on the manual absolute function.

Manual absolute check signal MABSM <F004#2>

[Classification] Output signal

[Function] Notifies the PMC of the state of the manual absolute signal.

[Output condition] This signal is set to 1 in the following case:
– When the manual absolute signal *ABSM is set to 0

This signal is set to 0 in the following case:
– When manual absolute signal *ABSM is set to 1

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006						*ABSM		
F004						MABSM		

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.3.5	Manual absolute ON/OFF
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.3.5	Manual absolute ON/OFF
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.3.5	Manual absolute ON/OFF
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.3.5	Manual absolute ON/OFF

5.5 OPTIONAL BLOCK SKIP/ADDITION OF OPTIONAL BLOCK SKIP

General

When a slash followed by a number (/n, where n = 1 to 9) is specified at the head of a block, and optional block skip signals BDT1 to BDT9 are set to 1 during automatic operation, the information contained in the block for which /n, corresponding to signal BDTn, is specified is ignored (from /n to the end of the block).

(Example) /2 N123 X100. Y200. ;

Input signal	Code specified at the head of a block
BDT1	/ or /1 (Note 1)
BDT2	/2
BDT3	/3
BDT4	/4
BDT5	/5
BDT6	/6
BDT7	/7
BDT8	/8
BDT9	/9

NOTE

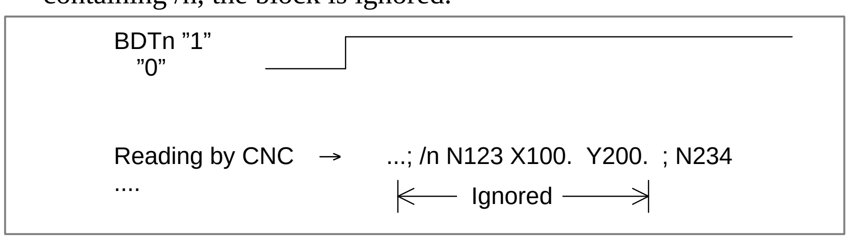
1 Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used in one block, number 1 for /1 cannot be omitted.

(Example)

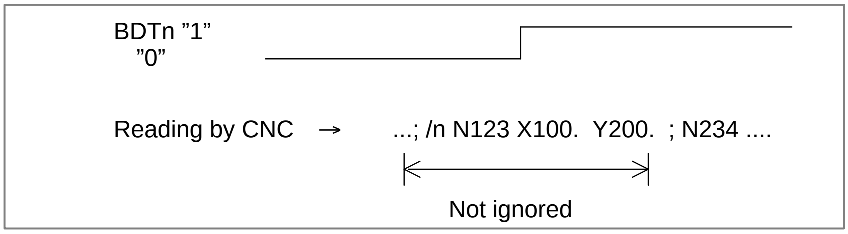
//3 N123 X100. Y200. ; — Invalid
/1 /3 N123 X100. Y200. ;— Valid

The following figures show the relationship between the timing, when optional block skip signals (BDT1 to BDT9) are set to 1, and the ignored information:

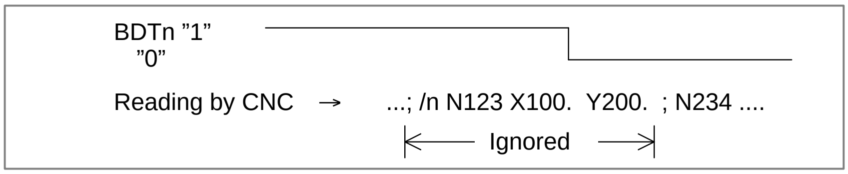
1. When BDTn is set to 1 before the CNC starts reading a block containing /n, the block is ignored.



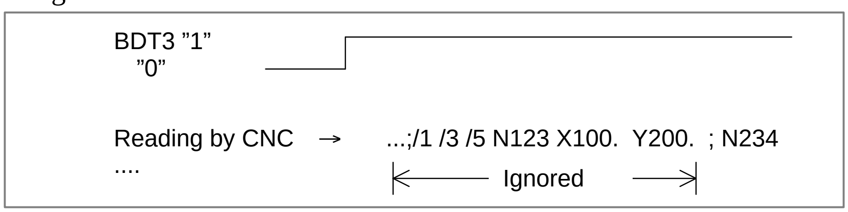
2. When BDTn is set to 1 while the CNC is reading a block containing /n, the block is not ignored.



3. When BDTn, currently set to 1, is set to 0 while the CNC is reading a block containing /n, the block is ignored.



4. When two or more optional block skip switches are specified in a block and BDTn, corresponding to one of them, is set to 1, the block is ignored.



Signal

Optional block skip signals BDT1 <G044#0> BDT2 to BDT9 <G045>

[Classification] Input signal

[Function] Select whether a block containing /n is to be executed or ignored.

[Operation] During automatic operation, a block containing /n in the program is ignored when the corresponding optional block skip signal is set to 1. It is executed normally when the signal is set to 0.

Optional block skip check signals MBDT1 <F004#0> MBDT2 to MBDT9 <F005>

[Classification] Output signal

[Function] Notify the PMC of the states of the optional block skip signals BDT1 to BDT9. Nine signals are provided, corresponding to the nine optional block skip signals. Signal MBDTn corresponds to signal BDTn.

[Output condition] Signal MBDTn is set to 1 in the following case:
– When the corresponding optional block skip signal (BDTn) is set to 1

Signal MBDTn is set to 0 in the following case:
– When the corresponding optional block skip signal (BDTn) is set to 0

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G044								BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
F004								MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2

Note

NOTE

1 This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip signal is set.
Programs stored in memory can be output, regardless of how the optional block skip signals are set.
Optional block skip is effective even during sequence number search operation.

2 Position of a slash
A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

3 TV and TH check
When an optional block skip signal is “1”. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is “0”.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.12.2	Program section configuration
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.12.2	Program section configuration
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.12.2	Program section configuration
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.12.2	Program section configuration

5.6 SEQUENCE NUMBER COMPARISON AND STOP

General

During program execution, this function causes a single block stop right after a block with a specified sequence number is executed.

To use this function, first specify the program number (1 to 9999) of a program that contains a sequence number where operation is to be stopped and the sequence number on the setting data screen:
With this setting, a single block stop occurs after the execution of the block with the specified sequence number during automatic operation.

Parameter

Setting data

– SEQUENCE STOP (PROGRAM NO.)
  Specify the program number (1 to 9999) of a program to which a sequence to be stopped belongs.
– SEQUENCE STOP (SEQUENCE NO.)
  Specify the sequence number (1 to 99999) of a sequence to be stopped.

Note

NOTE
After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.4	Sequence Number Comparison and Stop
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.8	Sequence Number Comparison and Stop
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.4.4	Sequence Number Comparison and Stop
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.4.8	Sequence Number Comparison and Stop

5.7 PROGRAM RESTART

General

A program may be restarted at a block by specifying the sequence number of the block, after automatic operation is stopped because of a broken tool or for holidays. This function can also be used as a high–speed program check function.

There are two types of restart methods.

P type: Restart after a tool is broken down
Q type: Restart after holidays

Signal

Program restart signal SRN<G006#0>

[Classification] Input signal

[Function] Selects program restart.

[Operation] When the program restart signal is set to “1” to search for the sequence number of the block to be restarted, the CRT screen changes to the program restart screen. When the program restart signal is set to “0”, and automatic operation is activated, the tool is moved back to the machining restart point at dry run speed along the axes one by one in the sequence specified in parameter No. 7310. When the tool is set to the restart point, machining restarts.

Program restart under way signal SRNMV<F002#4>

[Classification] Output signal

[Function] Indicates the program is being restarted.

[Output condition] The program restart under way signal becomes “1” when:

– The program restart signal is set to “0” after the CRT screen changes to the program restart screen.

The signal is set to “0” when:

– The program restart sequence ends (the tool has been moved to the restart point on all controlled axes).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006								SRN
F002				SRNMV				

Parameter

7310	Movement sequence to program restart position

Setting entry is accepted.

[Data type] Byte axis

[Valid data range] 1 to no. of controlled axes

This parameter sets the axis sequence when the machine moves to the restart point by dry run after a program is restarted.

[Example]
The machine moves to the restart point in the order of the fourth, first, second, and third axes one at a time when the first axis = 2, the second axis = 3, the third axis = 4, and the fourth axis = 1 are set.

Alarm and message

Number	Message	Description
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.)
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be specified when the program is restarted. (After power ON, after emergency stop or P / S alarm 94 to 97 were reset, no automatic operation was performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.

Warning

WARNING

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.4.5	PROGRAM RESTART
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.4.3	PROGRAM RESTART
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.4.4	PROGRAM RESTART
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.4.3	PROGRAM RESTART

5.8 TOOL RETRACTION AND RETURN

General

The tool can be retracted from a workpiece to replace the tool, if damaged during machining, or to check the status of machining. Then, the tool can be returned to restart machining efficiently.

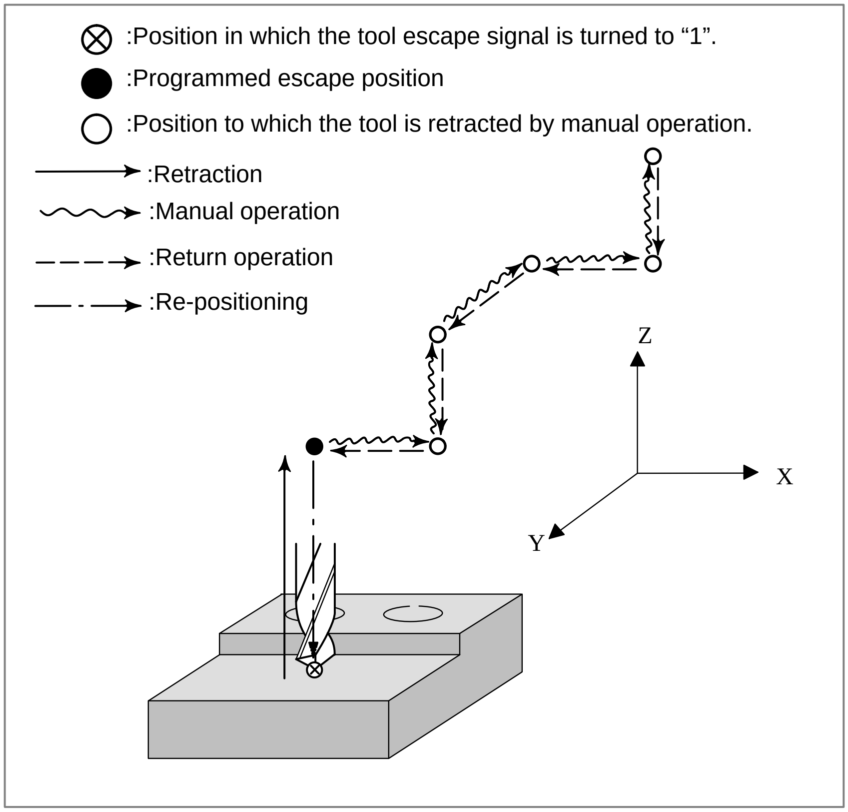
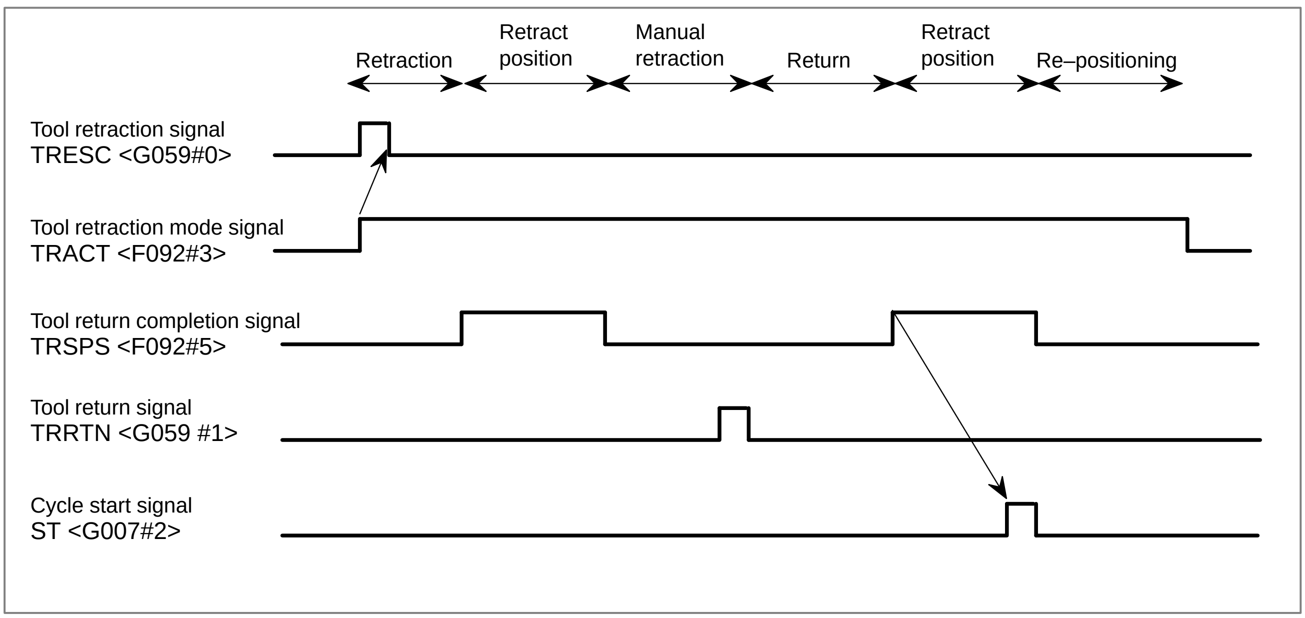


FIg. 5.8 Path of tool retraction and return for M series

Basic procedure for tool retraction and return

- When tool retraction signal TRESC input from the PMC turned to “1” while automatic operation is started, stopped, or held , the tool is retracted by the distance specified in the program. The position to which the tool retracts is called the retraction position. When TRESC turned to “1” while automatic operation is started, if a block is being executed, the tool is retracted after the block is suspended. When retraction ends, the machine enters the automatic operation hold state. The clearance and direction can be specified by the program. If these values are not specified by the program, the tool is not retracted.
When TRESC turned to “1”, the machine enters the tool retraction mode, and tool retraction mode signal TRACT turned to “1” to notify the PMC that the machine has entered the tool retraction mode.

- In the manual mode, when it is necessary to replace the tool or measure workpieces, the tool can be moved manually, such as by manual continuous feed, or manual handle feed. This operation is called manual retraction. The path along which the tool retracts is automatically stored in the control unit. The number of paths which can be stored, however, is limited.
- Set automatic operation mode again. Set the tool return signal TRRTN to 1, then to 0. Then, the control unit traces back the path of the manually moved tool to automatically return the tool to the retract position. This operation is called return. When the tool has been returned to the retraction position, tool retraction completion signal TRSPS is turned to “1”.
- When the automatic operation is started at the retraction position, the tool is first returned to the position where TRESC was turned to “1”. This operation is called repositioning. When repositioning completes, TRACT is turned to “0” to notify the PMC of the end of the tool retraction mode. Then the operation differs according to the status of automatic operation when the machine entered the tool retraction mode.
  - ❏ When the machine enters the tool retraction mode while automatic operation is started, the automatic operation that has been held is resumed after the tool is repositioned.
  - ❏ When the machine enters the tool retraction mode while automatic operation is stopped or held, the machine returns to the same mode after the tool is repositioned. If the cycle is restarted, automatic operation is resumed.



Signal

Tool retraction signal TRESC<G059#0>

[Classification] Input signal

[Function] Tool retraction mode is selected.

[Operation] When this signal is turned to 1, the control unit retracts the tool by a pre–programmed distance.

Tool retraction mode signal TRACT<F092#3>

[Classification] Output signal

[Function] This signal reports that tool retraction mode is set. When the control unit is reset while the signal is 1, the signal is turned to 0, and tool retraction mode is canceled.

[Output condition] The signal is turned to "1" when:
• Tool retraction mode is selected.
The signal is turned to "0" when:
• Tool retraction mode is not selected.

Tool return signal TRRTN<G059#1>

[Classification] Input signal

[Function] In tool retraction mode, a tool that has been retracted manually along an axis is returned to the retract position along the same axis.

[Operation] When this signal is turned to "1", the control unit traces back the path of the manually moved tool to automatically return the tool to the retract position.

Tool return completion signal TRSPS<F092#5>

[Classification] Output signal

[Function] This signal reports that the tool is in the retract position in tool retraction mode. When this signal is "0", re–positioning cannot be performed by pressing the cycle start button.

[Output condition] The signal is set to "1" when:
• Retraction has been completed.
• The tool has been returned to the retract position.

The signal is set to "0" when:
• The tool is not in the retract position in tool retraction mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G059							TRRTN	TRESC

	#7	#6	#5	#4	#3	#2	#1	#0
F092			TRSPS		TRACT			

Warning

WARNING

The retraction axes and retraction distances specified with G10.6 need to be changed in appropriate blocks depending on the figure to be machined. An incorrectly specified retraction distance may damage a workpiece, the machine, or the tool. So, be very careful when specifying a retraction distance.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III. 4.10	Tool Retraction and Return
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III. 4.10	Tool Retraction and Return

5.9 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 G62 (automatic corner override), G63 (tapping mode), or G64 (cutting mode), is commanded.

• Tapping Mode (G63)

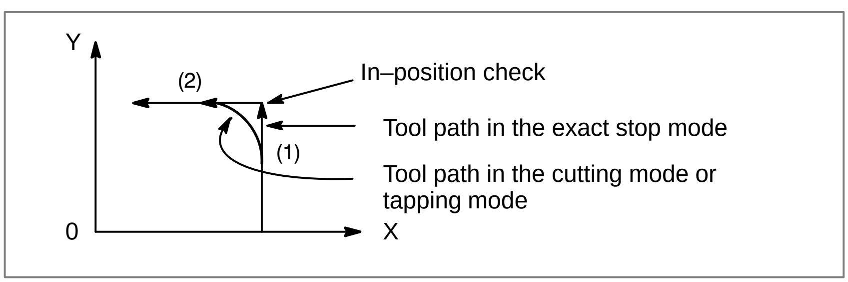
When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. This G63 is valid till G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

• Cutting Mode (G64)

When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid till G61 (exact stop mode), G62 (automatic corner override), or G63 (tapping mode) is commanded.
However, in G64 mode, feed rate is decelerated to zero and in–position check is performed in the following case;
1) Positioning mode (G00, G60)
2) Block with exact stop check (G09)
3) Next block is a block without movement command

*1 The term in–position indicates that the servo motor reaches in a range of positions specified by a parameter. See Section 7.2.6.1 and 7.2.6.2 for details.

(Example) Tool paths from block (1) to block (2)



Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.4.1	Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)

5.10 BALANCE CUT (2–PATH CONTROL FOR T SYSTEM)

General

When a thin workpiece is to be machined as shown in fig. 5.10, 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.

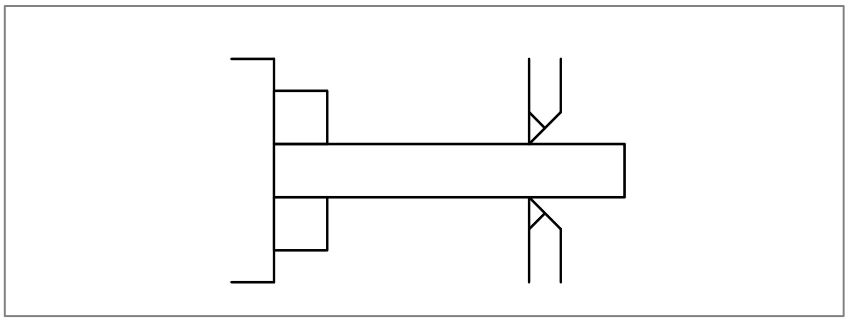


Fig. 5.10 Balance cut

Alarm and message

No.	Message	Contents
163	COMMAND G68/G69 INDEPENDENTLY (2–system control)	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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II. 20.4	Balance Cut (G68, G69)

5.11 DNC OPERATION

General

By starting automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read 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/puncher interface, and remote buffer in advance.

Signal

DNC operation select signal DNCI<G043#5>

[Classification] Input signal

[Function] Selects the DNC operation mode (RMT).
To select the DNC operation mode (RMT), it is necessary to select the memory operation mode (MEM) and set the DNC operation select signal to “1”.

[Operation] When the DNC operation select signal becomes “1”, the control unit operates as follows:

– If the memory mode (MEM) has not been selected, the signal is ignored, and nothing happens.
– If the memory operation mode (MEM) has been selected, the DNC operation mode (RMT) is selected, and DNC operation becomes possible. In this case, the DNC operation selection confirm signal MRMT becomes “1”.

DNC operation selection confirm signal MRMT<F003#4>

[Classification] Output signal

[Function] Indicates that the DNC operation mode (RMT) has been selected.

[Output condition] The DNC operation selection confirm signal becomes “1” when:
– The DNC operation mode (RMT) is selected.

The DNC operation selection confirm signal becomes “0” when:
– The DNC operation mode (RMT) is not selected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G043			DNCI					

	#7	#6	#5	#4	#3	#2	#1	#0
F003				MRMT				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0100			ND3					

Setting entry is accepted.

[Data type] Bit

ND3 In DNC operation, a program is:

0 : Read block by block. (A "DC3" code is output for each block.)

1 : Read continuously until the buffer becomes full. (A "DC3" code is output when the buffer becomes full.)

NOTE

In general, reading is performed more efficiently when ND3=1. This specification reduces the number of buffering interruptions caused by reading of a series of blocks specifying short movements. This reduces the cycle time.

Alarm and message

Number	Message	Description
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M99	M198 and M99 are executed in the schedule operation. Or M198 is executed in the DNC operation.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.6	SCHEDULING FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.4	SCHEDULING FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.5	SCHEDULING FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.4	SCHEDULING FUNCTION
	CONNECTION MANUAL	13.1	READER/PUNCHER INTERFACE
		13.2	REMOTE BUFFER

5.12 MANUAL INTERVENTION AND RETURN

General

If the tool movement along the axes is stopped by a feed hold during automatic operation, then restarted after manual intervention such as tool exchange, the tool moves back to the point of intervention before automatic operation is resumed.
This function is easy to operate because unlike the program restart function or the tool retract and restore function, it is unnecessary to operate switches on the operator's panel or MDI keys.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7001								MIN

[Data type] Bit

MIN The manual intervention and return function is:
0 : Disabled.
1 : Enabled.

Warning

WARNING
If you do not make manual intervention correctly according to the direction of machining and the shape of the workpiece, the machine and tool may be broken down. Use sufficient care.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.4.12	MANUAL INTERVENTION AND RETURN
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.4.8	MANUAL INTERVENTION AND RETURN
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III. 4.9	MANUAL INTERVENTION AND RETURN
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.4.8	MANUAL INTERVENTION AND RETURN

5.13 RETREAT AND RETRY FUNCTIONS (M SERIES)

General

The retreat and retry functions incorporate those functions that are needed to enable retreat and retry operations with a PMC and custom macros. Even if machining is interrupted by a reset or emergency stop, the tool can be returned from the interruption point (machining retreat function) to restart machining from the start block of the interrupted machining (machining retry function) easily.

The retreat and retry functions consist of the functions below.

(1) Management of machining cycles by means of sequence numbers

Machining cycle management is performed using the following sequence numbers:

N7000 to N7998: Machining start point
N7999: Clearing of data to perform machining retreat or retry operation
(Until N7999 is specified, data is not cleared to perform machining retreat or restart operation.)
N8000 to N8999: Machining cycle start point
N9000 to N9999: Machining cycle end point

(2) Saving of position information and modal information to custom macro variables at a machining start point and machining cycle start point

(3) Rigid tapping return function

(4) Restarting of machining at a machining start point or machining cycle start point

Format

Create a machining program in the format described below.

```
O0001  ;
(For an ordinary machining cycle)
N7000 · · · · · · · ·     - - - - -   (1)    - - - - -  Machining start point
   · · · · · · · · · · ·
N8000 · · · · · · · ·     - - - - -   (2)    -┐         Machining cycle
N9000 · · · · · · · ·     - - - - -   (3)    -┘
N8010 · · · · · · · ·                        -┐         Machining cycle
N9010 · · · · · · · ·                        -┘
   · · · · · · · · · · ·
N7999 · · · · · · · ·     - - - - -   (4)    - - - - -  Clears machining data
N7100
(For a drilling canned cycle)
N7010 · · · · · · ·  ·    - - - - -          - - - - -  Machining start point

N8010 · · · · · · · ·                        - - - - -  Machining cycle
N8020 · · · · · · · ·                        - - - - -  Machining cycle
   · · · · · · · · · · ·
N7020 · · · · · · · ·                        - - - - -  Machining start point
   · · · · · · · · · · ·
M30
```

(1) After specifying positioning at a machining start point, specify a sequence number from 7000 to 7998 in a block where various preparatory functions (M, S, and T) for machining cycles are specified. The start point of a block where a sequence number from 7000 to 7998 is specified is regarded as a machining start point. The absolute coordinates of the point are stored together with the program number and sequence number in macro variables. The M code specified in the block is stored as a machining type M code in a macro variable.

(2) In a block for starting actual machining (machining cycle) such as cutting and drilling, specify a sequence number from 8000 to 8999. The start point of a block where a sequence number from 8000 to 8999 is specified is regarded as a machining cycle start point. The absolute coordinates of the point are stored together with the sequence number in macro variables. The S/F codes and G codes of group 5 (G94/G95) of the block are also stored in macro variables.

When a sequence number from 8000 to 8999 is specified, the macro variable used for the hole bottom reach flag (described later) is cleared.

When a drilling canned cycle is used, the position stored based on a sequence number from 8000 to 8999 is not the hole position but the position where the drilling canned cycle is specified.

(3) Specify a sequence number from 9000 to 9999 in a block for ending the machining cycle. When a sequence number from 9000 to 9999 is specified, the specification of a cycle end point is assumed; the macro variable used for the hole bottom reach flag is set.

The set flag is cleared when a sequence number from 8000 to 8999 is specified.

When a drilling canned cycle is used, the use of a sequence number from 9000 to 9999 cannot be specified. So, when the drilling canned cycle is completed, the macro variable used for the hole bottom reach flag is directly set.

(4) When the sequence number 7999 is specified, the data stored in the macro variables is cleared. This is to indicate the end of one machining operation, and to prevent the workpiece from being damaged even if a restart command is inadvertently specified to return the tool to the previously stored position.

A restart command is ignored even if specified when the data stored in the macro variables has been cleared.

Explanations

- **Retreat function**

Each machine tool builder is to create a retreat function program, which is started from the PMC by using a workpiece number search capability or program number search capability. A machining start point or machining cycle start point is stored in a macro variable, and therefore can be used as required. When retreat operation varies from one machining cycle to another, specify an M function for each machining cycle. An M function is stored as a machining type M code in a macro variable. So, retreat operation can be specified for each machining cycle by referencing each macro variable in the retreat program.

- **Retry function**

When the restart of machining is specified from the PMC, the retry function moves the execution pointer of the machining program to one of the following:

A. Last machining start point executed

B. Last machining cycle start point executed
(If machining operation such as cutting and drilling is interrupted, the pointer is moved to this point when the machining is not completed.)

C. Machining cycle start point following the last machining cycle start point executed
(If machining operation such as cutting and drilling is interrupted, the pointer is moved to this point when the machining is completed.)

Then, when the cycle start button is set to on, machining is restarted where the cursor is placed. This function is implemented using the program restart function. For information about restrictions, see section 5.7.

When a miscellaneous function for the restart of machining is to be specified after the execution pointer of the machining program is moved to a desired restart block with the retry function, display the restart screen, and specify required commands in the MDI mode.

When machining is to be restarted at the machining cycle start point following the last machining cycle start point executed, P/S alarm No. 5066 is issued if a machining start point is detected before the next cycle start point is found.

- **Rigid tapping return**

If rigid tapping operation is interrupted by a reset or emergency stop, a movement can be made on the tapping axis to the initial point or point R in synchronism with the spindle according to the rigid tapping command information in the machining program. For this purpose, execute the command below in the retreat program.

The rigid tapping command is a one–shot code.

```
G30 P99 M29 S_rpm ;
```

(Example)

Machining program

```
. . . . . . . . . . . .
N7000   M29  S1000  ;
N8000   G84  X20.   Y20.   R–10.   Z–30.   F500  ;
N8010   X50.   Y50.   ;
N8020   X100.   Y100.   ;
        G80  ;
. . . . . . . . .
```

Retreat program

```
. . . . . . . . . . . .
IF      [#mmm  NE  29    ]   GOTO  1000  ;
G30  P99  M29  S1000  ;
G00  Z–10.   ;
G00  X#xxx  Y#yyy  Z#zzz  ;
GOTO  2000  ;
. . . . . . . . .
```

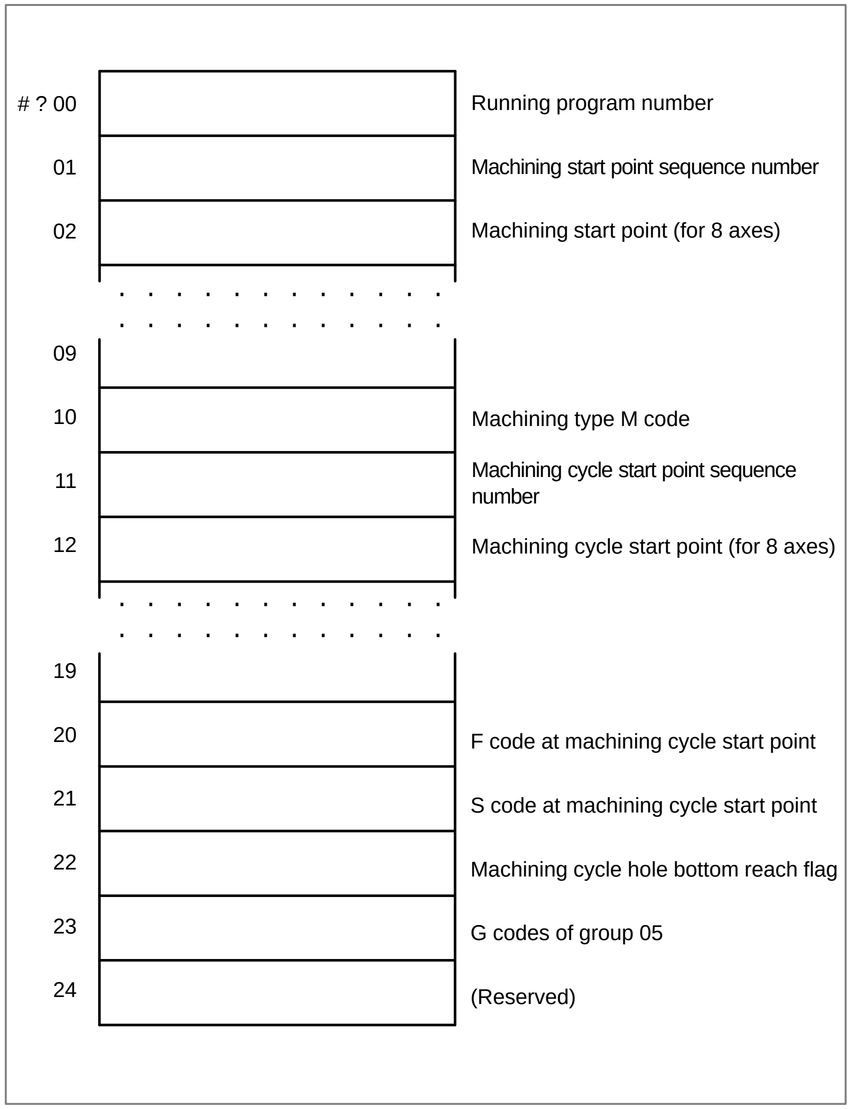
NOTE

1 Rigid tapping cannot be restarted from an intermediate hole. Be sure to restart rigid tapping at the rigid tapping start block (M29).

2 If a value other than 0 is specified in parameter No. 5210, specify the value in place of M29 in the program above.

- **Macro variables**

Information required for the machining retreat and retry functions is stored in macro variables. The start number of those variables is to be set in parameter No. 7351. Twenty–five successive variables starting with the variable specified in the parameter are used.

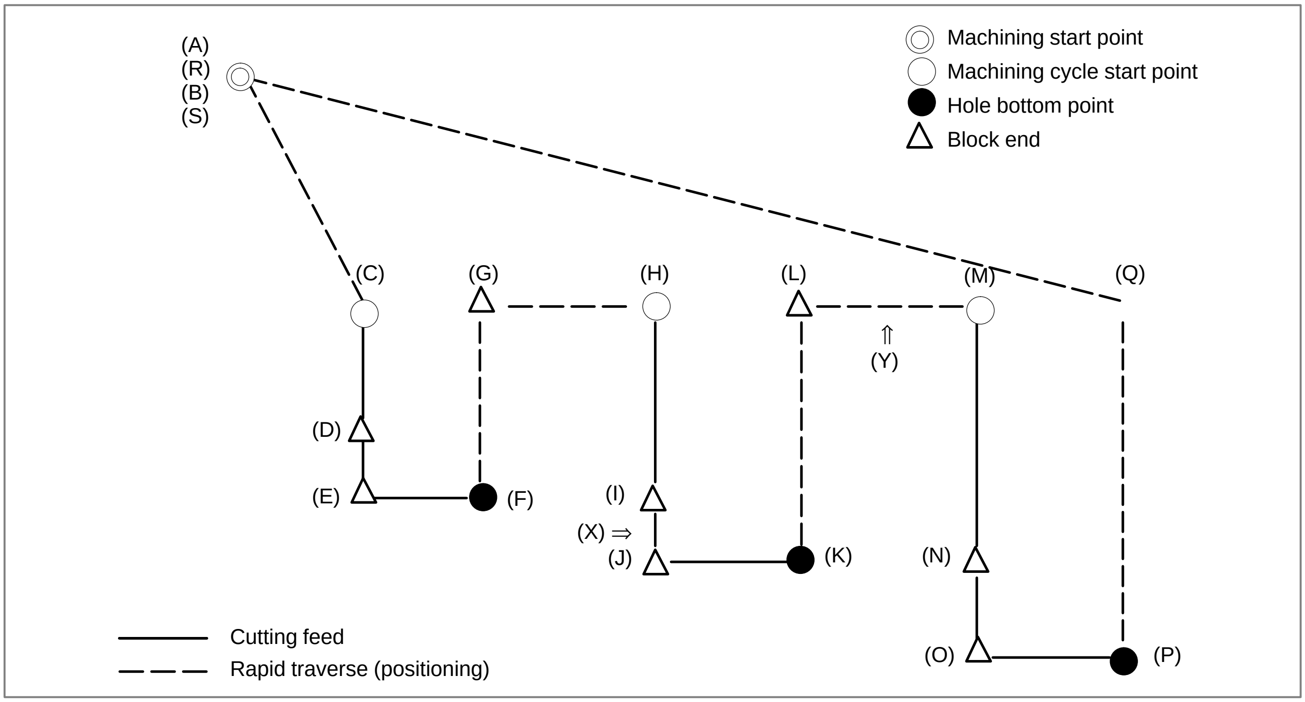


- **Retreat and retry sequence**

The retreat and retry sequences are explained using the sample machining program below.

```
O1000   ;
        . . . . . . . . . . . . . . . . . . . . . .
        G00   X100.   Y100.   Z100.   ;              (A)
N7010   M101   T10   S100   ;                        (B)
        G00   X0.   Y0.   Z0.   ;                    (C)
N8010   G01  Z– 20.   F100  ;                        (D)
        Z– 40.   ;                                   (E)
        Y20.   ;                                     (F)
N9010   G00  Z0.   ;                                 (G)
        X20.   ;                                     (H)
N8020   G01  Z– 40.   F200  ;                        (I)
        Z– 60.   ;                                   (J)
```

```
        Y40. ;                        (K)
N9020   G00 Z0. ;                     (L)
        X40. ;                        (M)
N8030   G01 Z– 80. F300 ;             (N)
        Z– 100. ;                     (O)
        Y60. ;                        (P)
N9030   G00 Z0. ;                     (Q)
        X100. Y100. Z100. ;           (R)
N7020   M102 T11 S200 ;               (S)
        . . . . . . . . . . . . . . . . . . . . . .   (T)
```



- **Machining retreat program**

If a machining cycle being executed is interrupted by a reset or emergency stop at (X), the retreat program is called and executed with a workpiece number search (refer to section 15.3) or program number search (refer to section 15.2) capability. In the retreat program, specify a return operation by referencing stored macro variables as required.

The start point of block (B) of N7010 is stored as the machining start point in a macro variable, and the start point of block (I) of N8020 is stored as the machining cycle start point in a macro variable.

M101 in block (B) of N7010 is stored as the machining type M code in a macro variable. By using this macro variable, a machining cycle can be determined so that the return cycle operation matching the machining cycle can be performed. For rigid tapping, a G code for rigid tapping return specification is available.

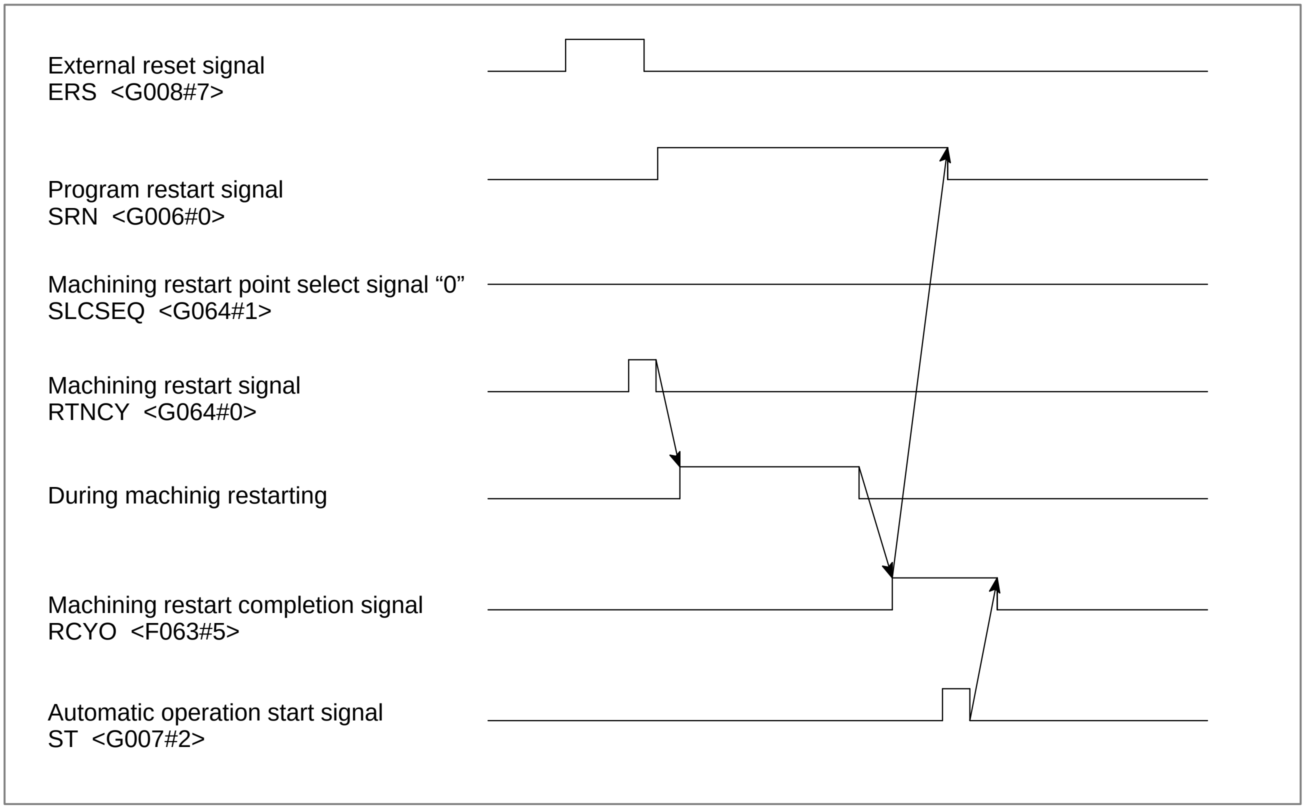
- **Restart of machining from the machining start point**

If a machining cycle being executed is interrupted by a reset or emergency stop at (X), machining can be restarted at machining start point (B) after starting the machining retreat program. In this case, use the following sequence:

1. Return the tool to machining start point (B) with the retreat program.
2. Set the External reset signal (ERS) to “1”.
3. Select the restart of machining at the machining start point with the machining restart point select signal (SLCSEQ).
4. Set the program restart signal (SRN) to “1”.
5. Set the machining restart signal (RTNCY) to “1”.

Thus, search operation is performed by virtually executing the programs from the start of program number 1000 to machining start sequence number 7010 stored in macro variables. Upon completion of search operation, the machining retry completion lamp is turned on.

Then, by pressing the cycle start button, machining is restarted from block (B).



- **Retry of machining from a machining cycle start point A**

If a machining cycle being executed is interrupted by a reset or emergency stop at (X), machining can be restarted at machining cycle start point (H) after starting the machining retreat program. In this case, use the following sequence:

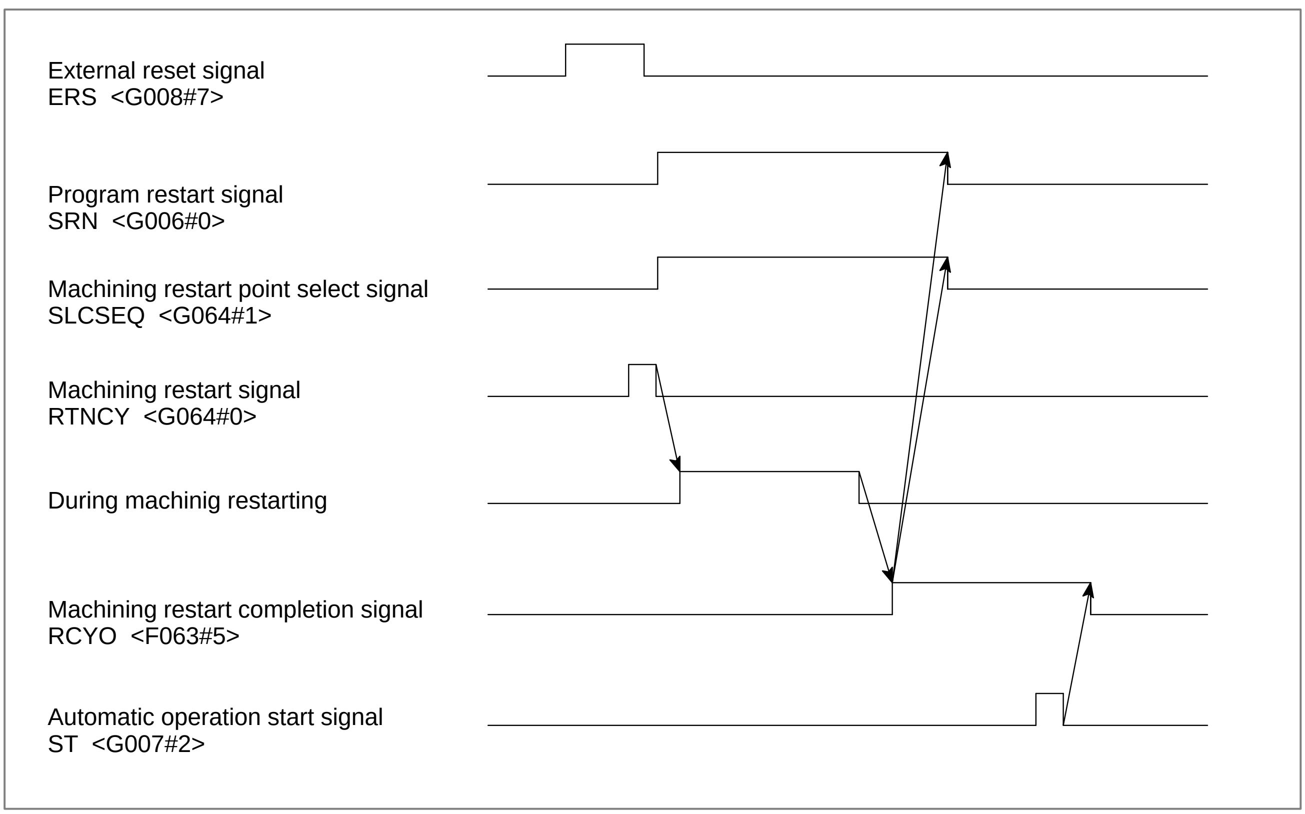
1. Return the tool to machining cycle start point (H) with the retreat program.
2. Set the external reset signal (ERS) to “1”.
3. Select the restart of machining at the machining cycle start point with the machining restart point select signal (SLCSEQ).
4. Set the program restart signal (SRN) to “1”.
5. Set the machining restart signal (RTNCY) to “1”.

Thus, search operation is performed by virtually executing the programs from the start of program number 1000 to machining cycle start sequence number 8020 stored in macro variables. Upon completion of search operation, the machining restart completion lamp is turned on.

Then, by pressing the cycle start button, machining is restarted from block (I).

- **Restart of machining from a machining cycle start point B**

If a machining cycle being executed is interrupted by a reset or emergency stop at (Y), machining can be restarted at the machining cycle start point after starting the machining retreat program. In this case, the restart sequence is the same as “Retry of machining from a machining cycle restart point A,” except that the restart sequence number is 8030, and machining is restarted at block (N) because the hole bottom reach flag is set in a macro variable.



Signal

Machining resume start signal RTNCY <G064#0>

[Classification] Input signal

[Function] Starts the processing required to resume machining.

[Operation] When the signal status changes from 1 to 0 in the reset state, the processing required to resume machining can be started.

Machining resume point selection signal SLCSEQ <G064#1>

[Classification] Input signal

[Function] Allows either the machining start point or machining cycle start point to be selected as the sequence number used to resume machining.

[Operation] When the signal is set to "1", the control unit causes machining to be resumed:
– At the machining cycle start point

When the signal is set to "0", the control unit causes the machining to be resumed:
– At the machining start point

Machining resume end signal RCYO <F063#5>

[Classification] Output signal

[Function] Indicates that the processing required to resume machining has been completed.

[Operation] The signal is turns to "1" when:
– The processing required to resume machining has been completed.

The signal is turns to "0" when:
– A reset or cycle start occurs.

Machining start point signal RTNMVS <F066#3>

[Classification] Output signal

[Function] Indicates that a machining start point sequence number is specified.

[Operation] The signal is turns to "1" when:
– A machining start point sequence number (N7000 to N7998) is specified.

The signal is turns to "0" when:
– A reset or emergency stop occurs, or a machining cycle start point sequence number (N8000 to N8999) is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G064							SLCSEQ	RTNCY
G063			RCYO					

	#7	#6	#5	#4	#3	#2	#1	#0
F066					RTNMSV			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5202								ORI

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

[Data type] Bit

ORI When rigid tapping is started:

0 : Spindle orientation is not performed.

1 : Spindle orientation is performed.

NOTE
This parameter can be used only for a serial spindle.

5382	Overshoot in rigid tapping return

[Data type] 2–word

[Unit of data] Input increment

[Valid data range] 0 to 99999999

For rigid tapping return (in the machining return or restart function), the tap axis can be extracted from the rigid tapping start position further to the position determined by adding a value specified in this parameter.

7351	Macro variable start number

[Data type] Word

[Valid data range] 500 to 975

This parameter specifies the first variable number where data for the machining return or restart function is stored. If a value out of the valid data range is specified, 500 is assumed.

NOTE
1 If 0 is specified, the machining return or restart data will not be set in a macro variable.
2 If a value from 507 to 531 is specified, 500 is assumed. If 532 or greater is specified, the custom macro variable expansion option becomes necessary.

Alarm and message

Number	Message	Description
5066	RESTART ILLEGAL SEQUENCE NUMBER	During program restart using the retreat/retry function, a sequence number between 7000 and 7999 was read while performing search for the next sequence number.

Note

NOTE
The following requirements must be satisfied to restart machining:
1. MEM mode is set.
2. The reset state is set.
3. Background editing is not performed.
4. Data required for machining cycles is stored in macro variables.
5. The notes for program restart operation must be observed.
(See III–5.7)

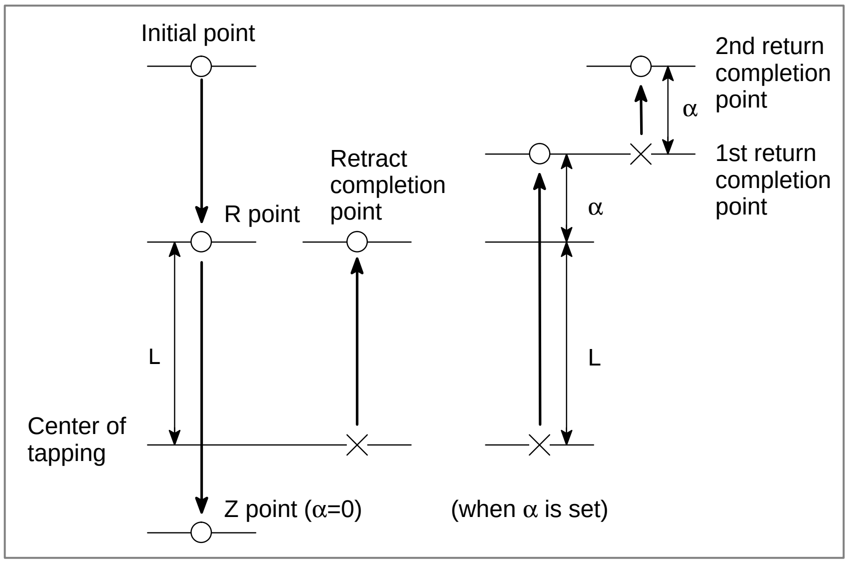
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.20.9	Retreat and retry function
		III.4.13	Retreat and retry function

5.14 RETRACTION FOR RIGID TAPPING (M SERIES)

General

When rigid tapping is stopped, either as a result of an emergency stop or a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap retraction signal is input, only drawn out of the rigid tapping cycle is executed, based on the stored information. The tap is pulled toward the R point. When a retract value α is set in parameter No. 5382, the pulling distance can be increased by α.



Basic procedure

(1) Start

Reset the CNC, then select MDI mode. Setting rigid tapping retraction start signal RTNT to “1” starts rigid tapping retraction.

(2) Completion

Upon the completion of rigid tapping retraction, rigid tapping retraction completion signal RTPT is set to “1”, with which the CNC automatically enters the reset state. Setting rigid tapping retract start signal RTNT to “0” sets rigid tapping retraction completion signal RTPT to “0”.

(3) Stop

During rigid tapping retraction, setting rigid tapping retraction start signal RTNT to “0” stops rigid tapping retraction, placing the CNC in the reset state. To resume rigid tapping retraction, set rigid tapping retraction start signal RTNT to “1”. Rigid tapping retraction can also be stopped by means of a reset or feed hold.

(4) Resume

Once rigid tapping retraction has been stopped, it can be resumed by performing the same operation as that used for starting rigid tapping retraction. If rigid tapping retraction has been completed, however, the start operation does not restart rigid tapping retraction. If retract value α is set in parameter No. 5382, however, the start operation performs rigid tapping retraction using α only.

Start and completion time chart

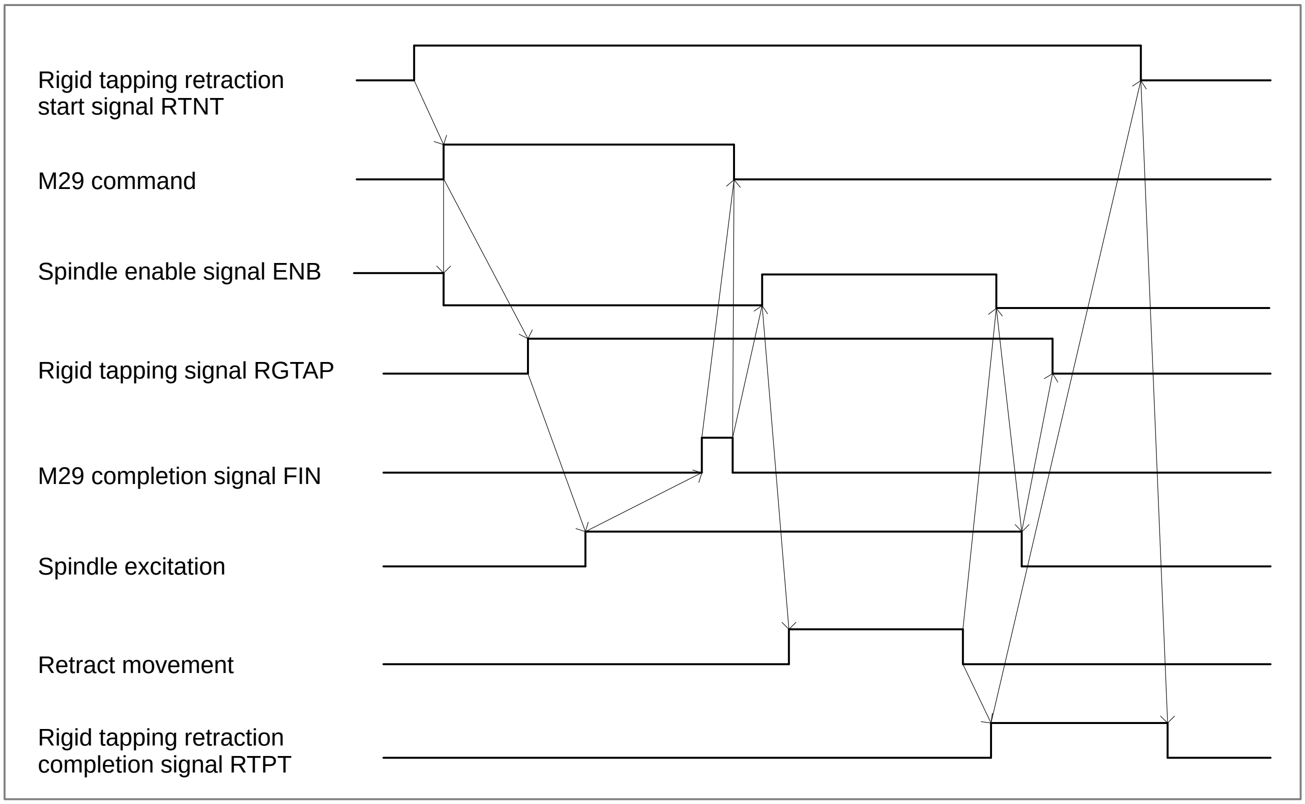
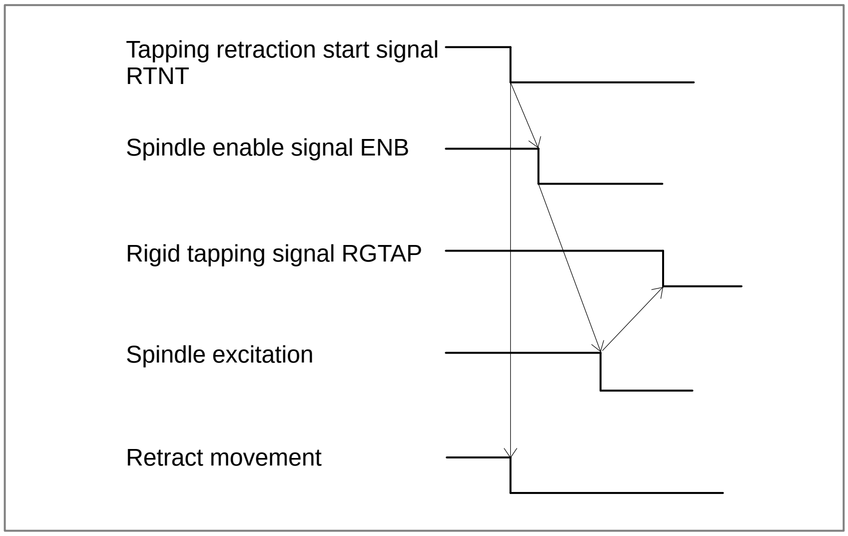


Fig. 5.14 Start and completion time chart

In the reset state, setting rigid tapping retraction start signal RTNT to “1” in MDI mode causes the rigid tapping M command to be output. For rigid tapping retraction, specify neither gear switching nor orientation. Spindle function strobe signal SF is also output if no S command has been specified after power–on.

Upon the completion of rigid tapping retraction, spindle enable signal ENB is set to “0”, in the same way as at the end of ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. Once rigid tapping retraction has been completed, rigid tapping retraction completion signal RTPT is set to “1” and the CNC enters the reset state.

Time chart for stopping rigid tapping retraction



When tapping retract is stopped, spindle enable signal ENB is set to “0”, in the same way as for ordinary rigid tapping. Therefore, perform the sequence for canceling rigid tapping. The CNC also automatically enters the reset state when tapping retract is stopped.

Signal

Rigid tapping retraction start signal RTNT <G062#6>

[Classification] Input signal

[Function] Starts rigid tapping retraction.

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

- Starts rigid tapping retraction.

Rigid tapping retraction completion signal RTPT <F066#1>

[Classification] Output signal

[Function] Notifies the completion of rigid tapping retraction.

[Output condition] This signal is set to “1” in the following case:

- Rigid tapping retraction has been completed.

This signal is set to “0” in the following case:

- Rigid tapping retraction start signal has been set to “0”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G062		RTNT						

	#7	#6	#5	#4	#3	#2	#1	#0
F066							RTPT	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5200				DOV				

[Data type] Bit

DOV For tool extraction during rigid tapping, override is:

0 : Disabled.
1 : Enabled.

5381	Override for rigid tapping retraction

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 200

Sets an override value to be applied to rigid tapping retraction. No override is applied if 0 is set.

NOTE
This parameter is enabled only when the parameter used to enable tool extraction (DOV: bit 4 of No. 5200) is set to 1.

5382	Retract value α for rigid tapping return

[Data type] 2–word

[Unit of data] Input increments

[Valid data range] 0 to 99999999

Sets an extra retract value for rigid tapping retraction. The tool will be pulled beyond the R point by α. If rigid tapping retraction has already been completed, the tool is pulled by α only.

Caution

CAUTION

1. If rigid tapping is stopped as a result of an emergency stop, the position on the tapping axis (Z–axis) is maintained but the spindle position is lost. In such a case, therefore, the positional relationship between the spindle and tapping axis is not guaranteed when operation is resumed.
2. Rigid tapping retraction is performed based on the tapping axis (Z–axis) commands accumulated for tapping. If rigid tapping is stopped as a result of an emergency stop, therefore, rigid tapping retraction may fail to draw the tapping tool completely out of the workpiece. In such a case, set retract value α (parameter No. 5382).
3. During rigid tapping retraction, switching the mode to manual mode stops rigid tapping retraction.
4. For rigid tapping retraction, the CNC internally activates a return program. Rigid tapping retraction may, therefore, cause some G codes or M/F/S codes to be overwritten (G80/G84/G74, G94/G95, G30).

Note

NOTE

1. Setting rigid tapping retraction start signal RTNT to "1" starts rigid tapping retraction only when the CNC is placed in both the reset state and MDI mode.
2. The machining data for rigid tapping retraction is maintained until a rigid tapping command is subsequently specified, even while the power is turned off. Rigid tapping retraction can, therefore, be specified even if the power has been turned off after rigid tapping.
3. Rigid tapping retraction is not performed if the input increments (inches or mm) selected when tapping return is specified differ from those selected when the machining data for tapping retraction was stored.
4. An override can be applied to rigid tapping retraction, if it is enabled with the corresponding parameter.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.13.2	Rigid tapping
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.13.8	Rigid tapping
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.13.2	Rigid tapping
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.13.7	Rigid tapping
	Connection manual (this manual)	9.11	Rigid tapping

6 INTERPOLATION FUNCTION

6.1 POSITIONING

General

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

The tool path is determined by selecting one of the following with parameter LRP (No. 1401#1):

- Linear interpolation type positioning
  The tool path for positioning is determined in the same manner as linear interpolation (G01). The tool moves at an appropriate speed so that positioning can be performed in the shortest time without the speed exceeding the rapid traverse rate for each axis.
- Non–linear interpolation type positioning
  Positioning is performed with each axis independently at the rapid traverse rate. Generally, the tool path is not a straight line.

The rapid traverse rate in the G00 command is set to the parameter No.1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401							LRP	

[Data type] Bit

LRP Positioning (G00)

0 : Positioning is performed with non–linear type positioning so that the tool moves along each axis independently at rapid traverse.
1 : Positioning is performed with linear interpolation so that the tool moves in a straight line.

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotaion axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

Note

NOTE
The rapid traverse rate cannot be specified in the address F.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.1	POSITIONING (G00)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.1	POSITIONING (G00)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.1	POSITIONING (G00)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.1	POSITIONING (G00)

6.2 LINEAR INTERPOLATION

General

Tools can move along a line

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01$\alpha\underline{\alpha}\beta\underline{\beta}\gamma\underline{\gamma}\zeta\underline{\zeta}$ $F\underline{f}$;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

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 axisβ (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the αandβ cartesian coordinate system is commanded by F(mm/min).
β–axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above fromula, then the β –axis feedrate unit is changed to deg/min.

A calculation example is as follows.

(Example)

G91 G01 X20.0B40.0 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

$$\frac{40}{0.14907} \doteq 268.3 \text{ deg/min}$$

In simultaneous 3 axes control, the feed rate is calculated the same way as in 2 axes control.

Parameter

1411	
	Cutting feedrate when the power is turned on

Setting entry is acceptable.

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 32767	6 – 32767
Inch machine	0.01 inch/min	6 – 32767	6 – 32767

When the machine requires little change in cutting feedrate during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate in the NC command data.

The feedrate set in this parameter is effective between the CNC being cleared, upon a power–on or a reset, and a feedrate being specified with a program command (F command). Once a feedrate has been specified with an F command, that feedrate becomes effective.

1422	Maximium cutting feedrate for all axes

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.
A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead. (M series)

1430	
	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotaion axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.
A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

NOTE
1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1422 is effective.
2 If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								G01

[Data type] Bit

G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

Alarm and message

No.	Message	Description
011	NO FEEDRATE COMMANDED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.

Reference item

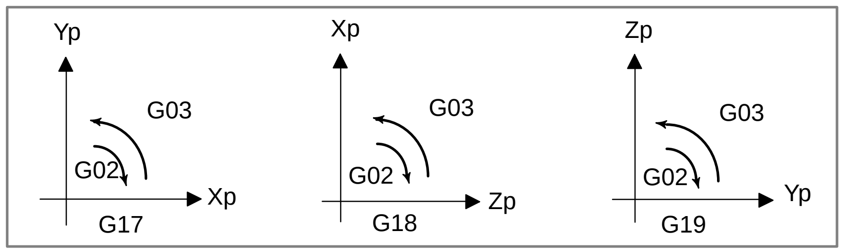
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.3	LINEAR INTERPOLATION (G01)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.2	LINEAR INTERPOLATION (G01)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.3	LINEAR INTERPOLATION (G01)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.2	LINEAR INTERPOLATION (G01)

6.3 CIRCULAR INTERPOLATION

General

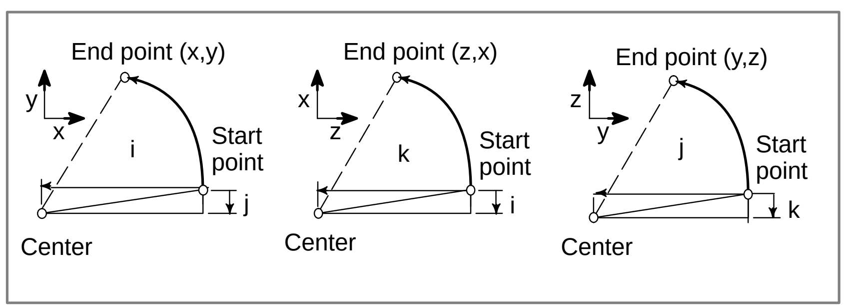
The command below can move a tool along a circular arc in the defined plane.

“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.



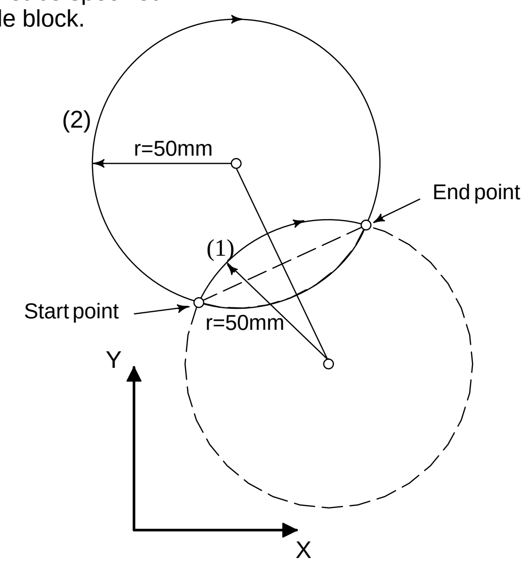
I0,J0, and K0 can be omitted. When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02 Ii; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.020) occurs.

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K. In this case, one arc is less than 180°, and the other is more than 180° are considered.

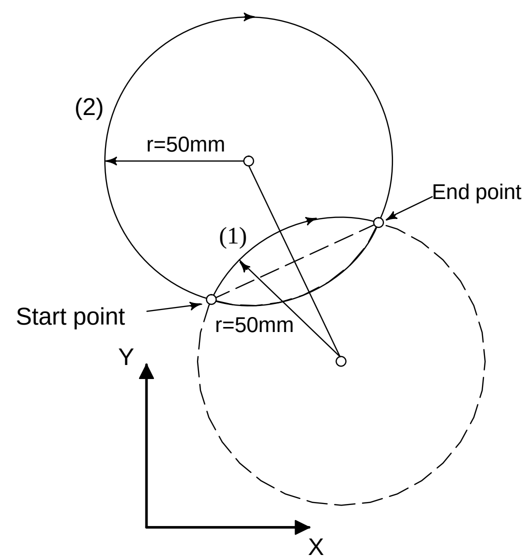
For T series, an arc with a sector angle of 180° or wider cannot be specified (P/S alarm No. 023).
For M series, specify an arc more than 180° with a negative radius value commanded.

If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0°is programmed.
G02Rr ; (The tool does not move.)

(Example) (T series)
For arc (1) (less than 180°)
G02 W60.0 U10.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
An arc with a sector angle of 180° or wider cannot be specified within a single block.



(Example) (M series)
For arc (1) (less than 180°)
G91 G02 X60.0 Y20.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
G91 G02 X60.0 Y20.0 R–50.0 F300.0 ;



When the option for specifying arc radius R with nine digits is selected for the T series, the valid radius range for circular interpolation is expanded as follows:

		Input increments	
		Metric input	**Inch input**
Increment system	IS–B	0.001 to 999999.999 mm	0.0001 to 99999.9999 inch
	IS–C	0.0001 to 99999.9999 mm	0.00001 to 9999.99999 inch

NOTE

1 Specifying an arc center with addresses I, K, and J
When the distance from the arc start point to the arc center is specified with addresses I, K, and J, a P/S alarm (No. 5059) is issued if:

$$\text{Maximum value which can be specified} < \sqrt{I^2 + K^2}$$

Example:
When IS–B and metric input are selected, issuing the following command (radius specification) will result in the issue of a P/S alarm (No. 5059):

```
G50 X0 Z0;
G18 G02 X11.250 Z10. I–800000.000 K900000.000 F5.0;
```

$$\because \sqrt{I^2 + K^2} = \sqrt{(-800000.000)^2 + 900000.000^2}$$
$$= 1204159.458$$
$$> 999999.999$$

2 Tool nose radius compensation
In tool nose radius compensation mode, a P/S alarm (No. 5059) is issued if the distance from the tool nose radius center to the arc center exceeds the maximum value which can be specified.

The feedrate in circular interpolation is equal to the feedrate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feedrate is measured along the arc after the cutter compensation (M series) or tool nose radius compensation (T series) is applied.

Parameter

1022	Setting of each axis in the basic coordinate system

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

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
						G19	G18	

[Data type] Bit

G18 and G19 Plane selected when power is turned on or when the control is cleared

G19	G18	G17, G18 or G19 mode
0	0	G17 mode (plane XY)
0	1	G18 mode (plane ZX)
1	0	G19 mode (plane YZ)

3410	Tolerance of arc radius

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

When a circular interpolation command (G02, G03) is executed, the tolerance for the radius between the start point and the end point is set. If the difference of radii between the start point and the end point exceeds the tolerance set here, a P/S alarm No. 20 is informed.

> **NOTE**
> When the set value is 0, the difference of radii is not checked.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Cutting feedrate was not commanded or the feedrate was inadequate. Modify the program.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
023	ILLEGAL RADIUS COMMAND (T series)	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03 (M series)	F0 (rapid traverse) was instructed by F1 –digit command in circular interpolation. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
P/S 5059	RADIUS IS OUT OF RANGE	For circular interpolation, the radius specified with addresses I and K exceeds the allowable range.

Note

NOTE

1 For T series, the U, V and W axes (parallel with the basic axis) can be used with G–code system B and C.
2 If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
3 If an axis not comprising the specified plane is commanded, an alarm is displayed.
For example, when G code system B or C is used, if U axis with X axis is specified as a parallel axis to X axis when plane XY is specified, an P/S alarm (No.028)is displayed.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.4	CIRCULAR INTERPOLATION (G02,G03)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.4	CIRCULAR INTERPOLATION (G02,G03)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.3	CIRCULAR INTERPOLATION (G02,G03)

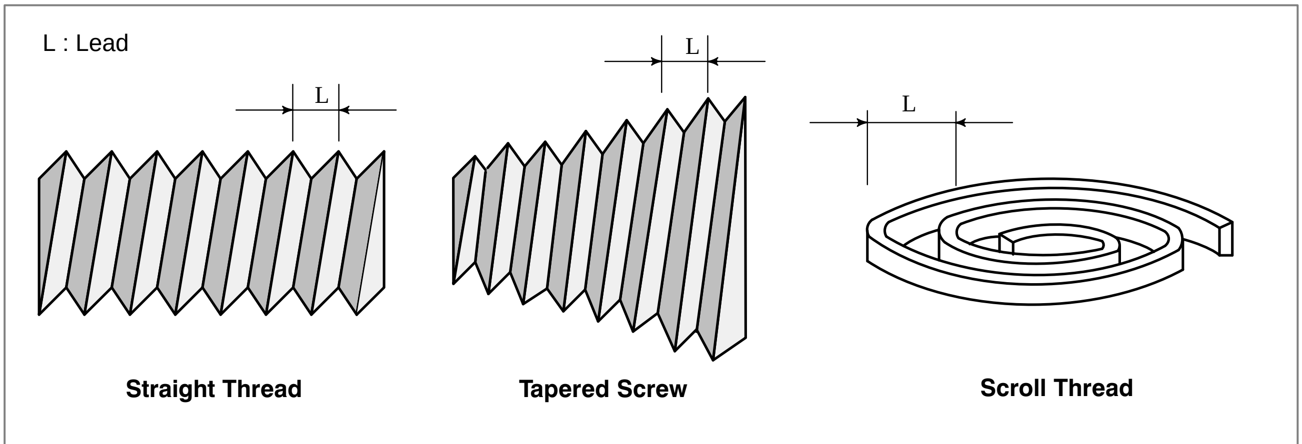
6.4 THREAD CUTTING

6.4.1 Thread Cutting

General

Tool movement can be synchronized with spindle rotation when cutting threads.

The spindle speed is continuously read through the position coder attached to the spindle. Then, it is converted to a cutting feedrate (feed per minute) to feed the tool.



In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

Signal

Thread cutting signal THRD<F002#3>

[Function] This signal indicates that thread cutting is in progress.

[Output condition] This signal turns to “1” in the following cases:
• Thread cutting mode in progress
• Thread cutting cycle for turning

This signal turns to “0” in the following case.
• Neither thread cutting mode nor thread cutting are in progress.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002					THRD			

Parameter

Enabling/disabling dry run operation during threading

	#7	#6	#5	#4	#3	#2	#1	#0
1401			TDR					

[Data type] Bit

TDR Dry run during threading or tapping (tapping cycle G74 or G84, rigid tapping)

0 : Enabled
1 : Disabled

Setting the thread cutting

	#7	#6	#5	#4	#3	#2	#1	#0
3405					G36			

[Data type] Bit

G36 For a G code used with the automatic tool compensation function:

0 : G36/G37 is used
1 : G37.1/G37.2 is used

Checking the spindle speed arrival signal before starting threading

	#7	#6	#5	#4	#3	#2	#1	#0
3708							SAT	SAR
								SAR

[Data type] Bit

SAR: The spindle speed arrival signal is:

0 : Not checked
1 : Checked

SAT: Check of the spindle speed arrival signal at the start of executing the thread cutting block

0 : The signal is checked only when SAR, #0 of parameter 3708, is set.
1 : The signal is always checked irrespective of whether SAR is set.

CAUTION
When thread cutting blocks are consecutive, the spindle speed arrival signal is not checked for the second and subsequent thread cutting blocks.

Setting the time constant for the threading cycle

1626	Time constant of exponetial acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the FL feedrate for the thread cutting cycle

1627	FL rate of exponential acceleration /deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

Setting the chamfering distance for the thread cutting cycle

5130	Chamfering distance in the thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in the thread cutting cycles G76 and G92.

Setting the minimum depth of cut for the multiple repetitive canned cycle G76

5140	Minimium depth of cut in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

This parameter sets the minimum depth of cut in the multiple repetitive canned cycle G76.

Setting the finishing allowance for the multiple repetitive canned cycle G76

5141	Finishing allowance in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the finishing allowance in the multiple repetitive canned cycle G76.

Setting the repetition count of finishing for the multiple repetitive canned cycle G76

5142	Repetition count of final finishing in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in the multiple repetitive canned cycle G76.

Setting the tool angle for the multiple repetitive canned cycle G76

5143	Tool nose angle in the multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 (When FS15 TAPE FORMAT is used)
0, 29, 30, 55, 60, 80 (When FS15 TAPE FORMAT is not used)

This parameter sets the tool nose angle in the multiple repetitive canned cycle G76.

Warning

WARNING

During threading, stopping feed without stopping the spindle is dangerous because the cutting depth will abruptly increase. Feed hold is, therefore, disabled during threading. If attempted during threading, feed stops in the same way as single block stop upon the completion of the first non–threading block after the termination of threading mode. The feed hold lamp (SPL lamp), however, lights immediately after the feed fold button (on the machine operator's panel) is pressed. The lamp goes off when feed stops (the CNC enters the single block stop state).

Caution

CAUTION

1 Feedrate override is ignored during thread cutting, 100% being assumed.
2 During threading, spindle override is ignored, 100% being assumed.
3 When the first non–threading block is executed after threading mode has been finished, and the feed hold button is pressed again (or the feed hold button has been held down), the execution of the non–threading block is stopped immediately.
4 When thread cutting is executed in the single block status, the tool stops after execution of the first block not specifying thread cutting.
5 When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1–turn signal even if the present block is a thread cutting block.
6 When a dry run operation is performed the dry run rate becomes the longitudinal axis feedrate.
7 For T series, the thread cutting retract function is supported only for the threading cycle.

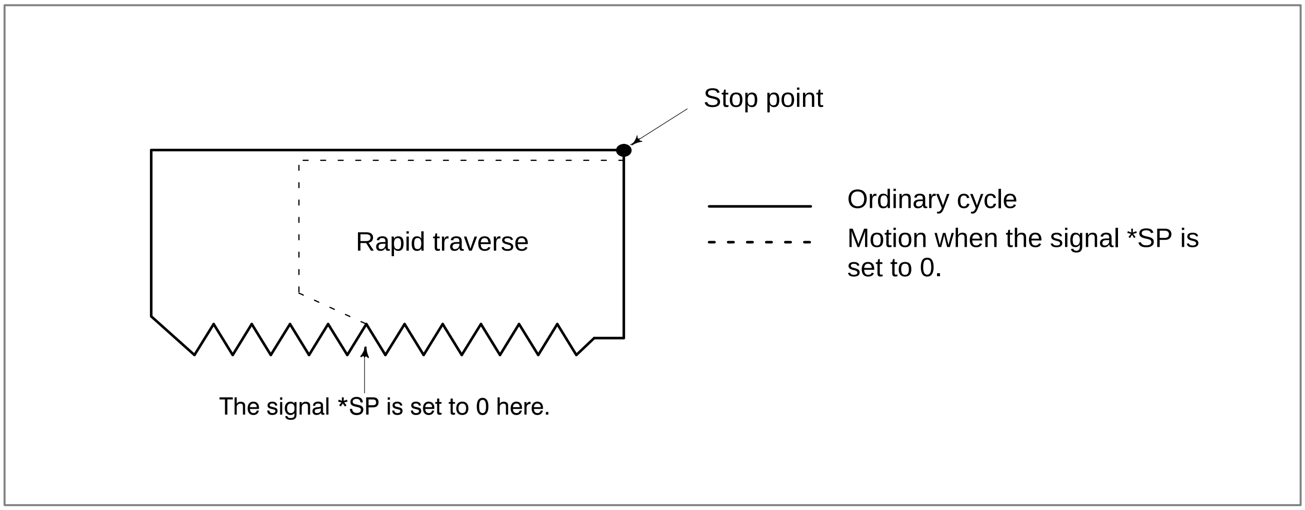
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.15	CONSTANT LEAD THREAD CUTTING
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.8	CONSTANT LEAD THREAD CUTTING
		II.4.9	VARIABLE LEAD THREAD CUTTING
		II.4.10	CONTINUOUS THREAD CUT-TING
		II.4.11	MULTIPLE THREAD CUTTING
		II.4.12	CIRCULAR THREAD CUTTING
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.7	CONSTANT LEAD THREAD CUTTING
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.6	CONSTANT LEAD THREAD CUTTING
		II.4.7	VARIABLE LEAD THREAD CUTTING
		II.4.8	CONTINUOUS THREAD CUT-TING
		II.4.9	MULTIPLE THREAD CUTTING

6.4.2 Thread Cutting Cycle Retract (T series)

General

When the automatic operation stop signal *SP <G008#5> is set to 0 during threading in a threading cycle, the tool immediately retracts while performing chamfering, then returns to the start point of the current cycle, first along the X–axis, then along the Z–axis.



Parameter

- **Setting to enable the override function during thread cutting cycle retraction**

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV							

[Data type] Bit

RTV Override while the tool is retracting in threading

0 : Override is effective.
1 : Override is not effective.

- **Setting a chamfering distance in thread cutting cycle retraction**

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1 pitch

[Valid data range] 0 to 127

This parameter sets the chamfering in thread cutting cycles G76 and G92.

Caution

CAUTION
While the tool is retracting, automatic operation stop signal *SP <G008#5> is ignored.

Note

NOTE
The chamfering distance for retraction is determined by the setting of parameter No. 5130.

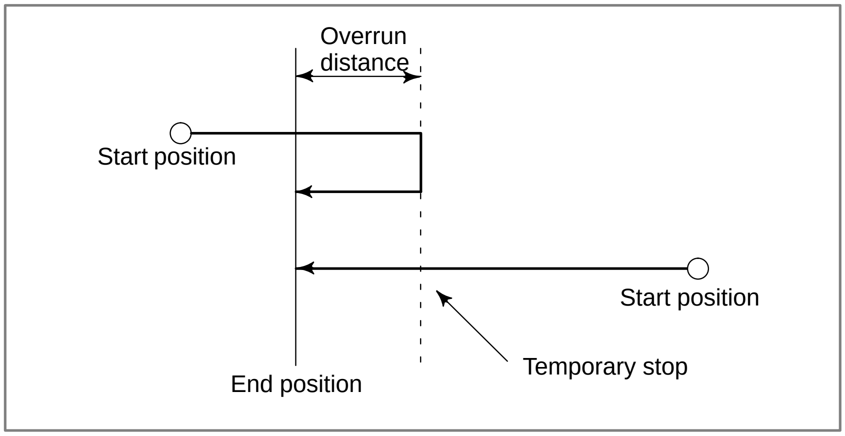
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.13.1.2	Thread Cutting Cycle
		II.13.2.7	Multiple Thread Cutting Cycle
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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 (No. 5440). 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
5431								MDL

[Data type] Bit

MDL Specifies whether the G code for single direction positioning (G60) is included in one–shot G codes (00 group) or modal G codes (01 group)

0: One–shot G codes (00 group)
1: Modal G codes (01 group)

5440	Positioning direction and overrun distance in uni–directional positioning for each axis

[Data type] Word axis

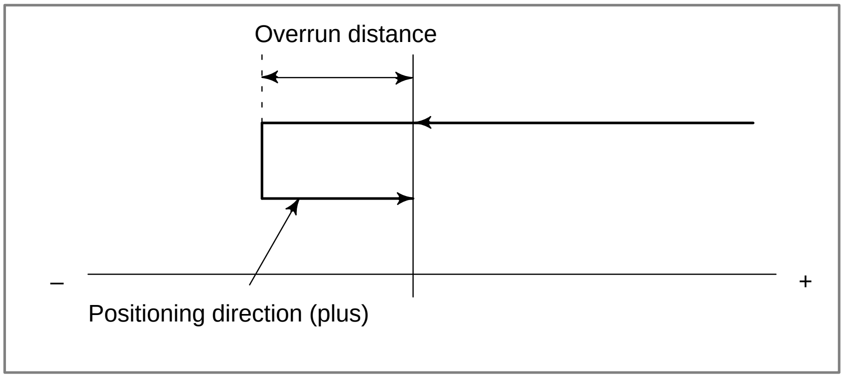
[Unit of data]

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

[Valid data range] –16383 to +16383

This parameter sets the positioning direction and overrun distance in uni–directional positioning (G60) for each axis. The positioning direction is specified using a setting data sign, and the overrun distance using a value set here.

Approach > 0: The positioning direction is positive (+).
Approach < 0: The positioning direction is negative (−).
Approach = 0: Uni–directional positioning is not performed.



Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.2	Single direction positioning
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.2	Single direction positioning

6.6 HELICAL INTERPOLATION

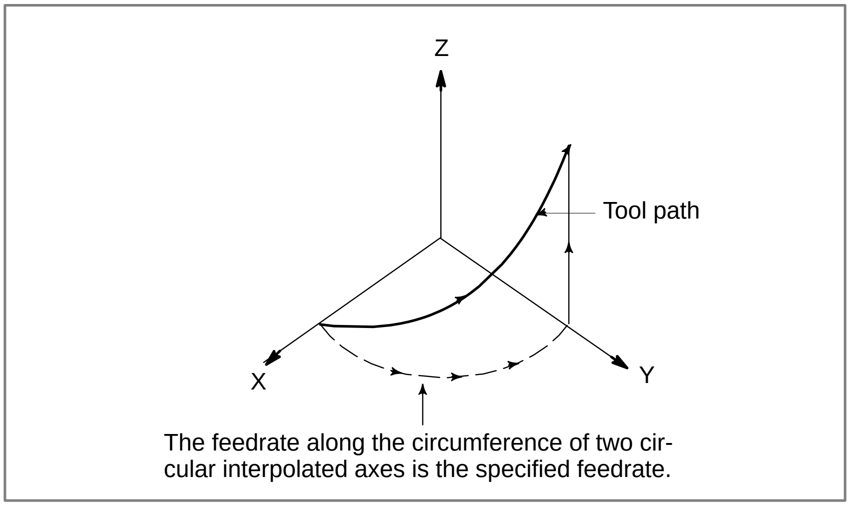
General

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
1404								HFC

HFC The feedrate for helical interpolation is:

0 : Clamped so that the feedrates along an arc and linear axis do not exceed the maximum cutting feedrate specified by parameter.

1 : Clamped so that the composite feedrate along an arc and linear axis does not exceed the maximum cutting feedrate specified by parameter.

When HFC is 1, and two linear axes exist, the combined feedrate for the four axes (two axes (arc) + two axes (straight line)) is clamped so that it does not exceed the maximum cutting feedrate.

<Parameters used for clamping>

When HFC is 0

No. 1430: Maximum cutting feedrate for each axis

Since the cutting feedrate for the arc is clamped to the above parameter value, the feedrate along the linear axis is clamped to the smaller parameter value.

Example: No. 1430 X 1000
Y 1200
Z 1400

G17 G03 X0. Y100. R100. Z1000. F5000;

The feedrate along the linear axis is clamped to 1000.

No. 1422: Maximum cutting feedrate (common to all axes)

If parameter No. 1430 is set to 0, the feedrate is clamped to the value set in this parameter.

When HFC is 1

No. 1422: Maximum cutting feedrate (common to all axes)

The cutting feedrate is clamped to the value set in this parameter. The value set with parameter No. 1430 is ignored.

Alarm and message

If more than two axes are specified together with the two axes for circular interpolation in a block specifying a helical interpolation operation, P/S alarm No. 232 is issued.

No.	Message	Description
0232	TOO MANY HELICAL AXIS COMMANDS	Three or more axes are specified as helical axes.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.5	Helical Interpolation
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.4	Helical Interpolation
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.5	Helical Interpolation

6.7 INVOLUTE INTERPOLATION (M SERIES)

General

With the involute interpolation function, an involute curve can be machined. Cutter compensation C is also possible. The use of involute interpolation eliminates the need to use short lines or arcs to approximate an involute curve. Pulse distribution is no longer interrupted by the high–speed operation of small blocks. As a result, smooth, high–speed operation is possible. In addition, part programs can be created more easily, and the required paper tape can be shortened.

Parameter

5610	Limit of initial permissible error during involute interpolation

[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 allowable limit of deviation between an involute curve passing through a start point and an involute curve passing through an end point for an involute interpolation command.

Alarm and message

No.	Message	Description
241	END POINT, I, J, K, AND R ARE MISSING	The end point of an involute curve, I, J, or K is not specified.
242	SPECIFICATION ERROR (INVOLUTE)	An illegal value is specified for involute interpolation. (1)The specified start point or end point is located inside the base circle. (2)Zero is specified for I, J, K, or R. (3)The start point or end point is located more than 100 turns from the beginning of the involute curve.
243	END POINT NOT ON INVOLUTE CURVE	The end point is not on the involute curve that passes through the start point, and is beyond the range specified with in parameter No. 5610.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.4.10	Involute Interpolation

6.8 POLAR COORDINATE INTERPOLATION

General

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This function is useful for grinding a cam shaft.

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 6.8). Polar coordinate interpolation is performed on this plane.

- **Polar coordinate interpolation plane**

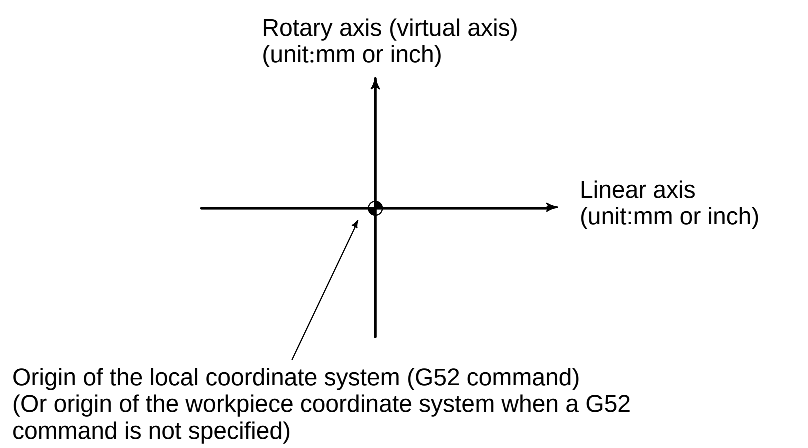


Fig. 6.8 Polar coordinate interpolation plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

NOTE
To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

5460	Axis (linear axis) specification for polar coordinate interpolation
5461	Axis (rotary axis) specification for polar coordinate interpolarion

[Data type] Byte

[Valid data range] 1, 2, 3, ... control axes count

These parameters set control axis numbers of linear and rotary axes to execute polar interpolation.

5462	Maximum cutting feedrate during polar coordinate interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0, 6 – 240000	0, 6 – 100000
Inch machine	0.1 inch/min	0, 6 – 96000	0, 6 – 48000
Rotation axis	1 deg/min	0, 6 – 240000	0, 6 – 100000

This parameter sets the upper limit of the cutting feedrate that is effective during polar coordinate interpolation. If a feedrate greater than the maximum feedrate is specified during polar coordinate interpolation, it is clamped to the feedrate specified by the parameter. When the setting is 0, the feedrate during polar coordinate interpolation is clamped to the maximum cutting feedrate usually specified with parameter 1422.

Alarm and message

No.	Message	Description
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.4.8	Polar Coordinate Interpolation
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.4.5	Polar Coordinate Interpolation
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.4.4	Polar Coordinate Interpolation

6.9 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.

Use parameter No. 1022 to specify whether the rotation axis is the X–, Y–, or Z–axis, or an axis parallel to one of these axes.
Only one rotation axis can be set for cylindrical interpolation.

Parameter

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

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

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

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

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

To determine the following planes used for circular interpolation, cutter compensation C (for the M series), tool nose radius compensation (for the T series), etc., each control axis is set to one of the basic three axes X, Y, and Z, or an axis parallel to the X, Y, or Z axis.

G17: Plane Xp–Yp

G18: Plane Zp–Xp

G19: Plane Yp–Zp

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

Alarm and message

Number	Message	Description
175	ILLEGAL G107 COMMAND	Conditions when performing cylindrical interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of “G07.1 rotation–axis name radius of cylinder.”
176	IMPROPER G–CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning, such as G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.7	Cylindrical Interpolation
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.6	Cylindrical Interpolation
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.4.6	Cylindrical Interpolation
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.5	Cylindrical Interpolation

6.10 POLYGONAL TURNING (T SERIES)

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

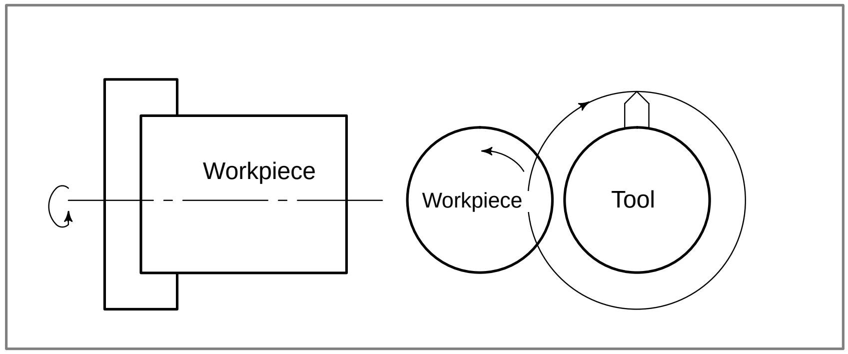


Fig. 6.10 (a) Polygonal turning

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.

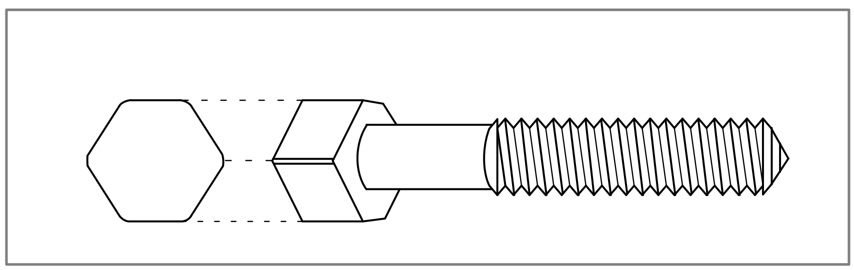


Fig. 6.10 (b) Hexagon bolt

This function controls the workpiece (spindle) and tool (rotation tool axis) so that the relationship between the spindle speed and tool speed is maintained at a constant ratio specified in a command given to the CNC.

(For the principle of polygonal turning, refer to Section 20.1, Part II of the “Operator’s Manual (For Lathe).”)

Either of the following can be selected as the tool rotation axis:

· CNC controlled axis (servo axis)

· Second spindle (with two serial spindles connected)

In the following descriptions, the term polygonal turning refers to a turning operation in which a servo axis is used as the tool rotation axis (See Section 6.10.1.).

The term polygonal turning with two spindle refers to a turning operation in which the second spindle is used as the tool rotation axis (See Section 6.10.2.).

6.10.1 Polygonal Turning

General

One of the axes (servo axes) controlled by the CNC is assigned as a tool rotation axis. Either serial spindle or analog spindle can be used as a workpiece axis (spindle).

Polygonal turning using a servo axis is detailed in the operator's manual (for lathe).

This section focuses on supplementary information and examples for the connection.

• Spindle connection

A position coder must be mounted on the spindle. However, polygonal turning requires no additional changes to the spindle connection (See Section 9.3.).

Polygonal turning uses the position coder feedback signal to control the positional relationship (cutting position) between the spindle and tool rotation axis, and the ratio of speed.

• Tool rotation axis (servo axis) connection

Parameter No. 7610 specifies the controlled axis (servo axis) to be used as the tool rotation axis.

The same parameter setting as for ordinary servo axes applies to the servo axis connection for polygonal turning except for some parameters.

When the machine is not in the polygonal turning mode, the servo axis specified as the rotation tool axis functions as a feed axis. So, the servo axis can be:

· Used as a subspindle under PMC axis control

· Positioned by a move command from a machining program.

However, be careful about the angle to rotate through and feedrate. Read the operator's manual (for lathe) and the following examples.

• Examples of parameter setting

· The following descriptions exemplify typical parameter setting for polygonal turning using a serial pulse coder (with a million pulse capability).

→ The parameter setting described here is not a must for polygonal turning.

→ Specify typical values for parameters unless otherwise stated.

● Tool rotation axis setting

This example uses the CNC's fourth axis (connected as the Y–axis) as a rotation tool axis for polygonal turning.

Parameter No. 7610 = 4
(controlled axis number for the tool rotation axis)

The following description assumes that the axis type parameter is set to the fourth axis.

● Servo parameter setting

Set the servo parameters as listed below:

CMR = 1
DMR = 36/1000

(With the above setting, the reference counter capacity is 36000.)

Parameter No. 1820 = 2 (CMR)
Parameter No. 1821 = 36000 (reference counter capacity)
Parameter No. 2084 = 36 (DMR numerator)
Parameter No. 2085 = 1000 (DMR denominator)

For the other servo parameters, specify typical values.

- Parameter setting for polygonal turning

The least command increment, detection unit, the angle to rotate through per rotation for the polygon axis are as follows:

$$\text{Least command increment} = \frac{L \times CMR}{Q \times DMR}$$

$$\text{Detection unit} = \frac{\text{least command increment}}{DMR} = \frac{L}{Q \times DMR}$$

Angle to rotate through per tool axis rotation

$$= \frac{360}{\text{least command increment}}$$

where

L: Tool axis rotation angle per motor rotation (degrees), (360 × speed increment ratio)

When the servo motor is connected directly to the rotation tool, for example, L = 360. When the tool speed is doubled, L = 720.

Q: Number of pulses per pulse coder rotation (For a serial pulse coder, Q = 1000000.)

The least command increment specified here is specific to the polygon axis. It is determined regardless of what is specified in parameter No. 1004 (ISA/ISC). However, both ISA and ISC must be set to 0 for IS–B setting.

If the servo motor is connected directly to the rotation tool:

$$\text{Least command increment} = \frac{360 \times 1}{1000000 \times \frac{36}{1000}} = 0.01 \text{ (degrees)}$$

Detection unit = 0.01 (degrees)

$$\text{Angle to rotate through per tool axis rotation} = \frac{360}{0.01} = 36000$$

(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. 7620 = 36000 (angle to rotate through per tool axis rotation)
No. 7621 = 2000 (upper limit to tool rotation axis speed)

- Feedrate parameter setting

Because the least command increment is 0.01 degrees, the input unit for the feedrate is 10 degrees/min.

To obtain a rapid traverse speed of 2000 rpm, for example, specify as follows:

No. 1420 = 72000 (= 2000 $\times \frac{360}{10}$)

Also specify other feedrates in 10 degrees/min units.

- Commands from the NC program

When the machine is not performing polygonal turning, the machining program can issue move commands to the polygon axis.

Such commands can be issued in the same way as for ordinary axes. However, be careful about the angle to rotate through and feedrate.

Assuming the polygon axis is the Y–axis, the polygon axis rotates through 0.03 degrees by the following command:

V3;

Likewise, the polygon axis rotates through 10.00 degrees by the following command:

V1.0;

The feedrate unit is also increased by tenfold.

The current position of the polygon in the machine coordinate system is normalized according to the value specified by parameter No. 7620.

Typical values range from 0.000 to 35.999.

Signal

Polygon synchronization under way signal PSYN <F063#7>

[Classification] Output signal

[Function] Informs the PMC that the machine is in the polygon turning mode.

[Output condition] The polygon synchronization signal is set to logical “1” by the polygon turning mode command (G51.2) and stays at “1” during the polygonal turning mode.

The signal is reset to logical “0” by the polygon turning mode reset command (G50.2) or a reset. It stays at logical “0” when the machine is not in the polygonal turning mode.

CAUTION
This signal uses the same address for both polygonal turning (using the servo axis) and polygonal turning with two spindles.

· Other signals (related to the tool rotation axis)

→ Some signals related to the CNC controlled axis used as the tool rotation axis may be made ineffective depending on whether the machine is in the polygonal turning mode.

For these signals, read the note in operator’s manual (for lathe).

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F063	PSYN							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7600	PLZ							

[Data type] Bit

PLZ Synchronous axis using G28 command

0: Returns to the reference position in the same sequence as the manual reference position return.

1: Returns to the reference position by positioning at a rapid traverse. The synchronous axis returns to the reference position in the same sequence as the manual reference position return when no return–to–reference position is performed after the power is turned on.

7610	Control axis number of tool rotation axis for polygon turning

[Data type] Byte

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

This parameter sets the control axis number of a rotation tool axis used for polygon turning.

7620	Movement of tool rotation axis per revolution

[Data type] Two–word

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

[Valid data range] 1 to 9999999

This parameter sets the movement of a tool rotation axis per revolution.

7621	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)

[Data type] Word

[Unit of data] rpm

[Valid data range] For polygonal turning using servo motors:

$$\frac{0 \text{ to } 1.2 \times 10^{8}}{\text{set value of the parameter No. 7620}}$$

This parameter sets the upper–limit rotation speed of a tool rotation axis. The rotation speed of the tool rotation axis is clamped by the set upper–limit rotation speed during polygon turning. The spindle and tool rotation axis go out of synchronization when the rotation speed is clamped (P/S alarm No. 5018).

Alarm and message

Number	Message	Description
217	DUPLICATE G251 (COMMANDS)	G51.2 (or G251) is further commanded in the polygonal turning mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G51.2 (or the G251) block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G51.2 (or G251) and G50.2 (or G250) are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHR–MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHR–MODE	Polygon machining synchronous operation and Cs contouring control or balance cutting are executed at a time. Modify the program.

Caution

CAUTION

1 Before issuing a G51.2, rotate the spindle. If it is not rotating when the G51.2 is issued, the program stops to wait for a one–rotation signal from the position coder on the spindle. This does not apply to a dry run.

2 A reset releases the polygonal turning mode.

3 Machine a workpiece at the same spindle speed until finish machining for the workpiece.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.19.1	POLYGONAL TURNING
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.19.1	POLYGONAL TURNING

6.10.2 Polygonal Turning with Two Spindles

General

In the polygonal turning with two spindles, the first spindle is used as a workpiece rotation axis (master axis). The second spindle is used as a tool rotation axis (polygon synchronization axis). Spindle rotation control is applied to both spindles with a constant ratio.

The polygonal turning with two spindles can use different spindle speeds for the same workpiece, because it performs automatic phase compensation when a polygon synchronization mode command is issued or the S command is changed during polygon synchronization mode. With this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.

Moreover, polygonal turning works with the first and second spindles on each tool post in a two–path lathe application. However, polygonal turning does not work with spindles on different tool posts.

- **Command format**

The CNC command format for polygonal turning with two spindles is described below. For the CNC command format for polygonal turning (see Section 6.10.1), refer to the operator's manual (for lathe). The two formats are almost identical. The differences are in that the polygonal turning with two spindles can specify a phase command (R) and re–issue commands.

- **Mode command and command value change**

◆ G51.2 P_Q_R_;

This command starts the polygon synchronization mode or changes the values specified for the polygon synchronization mode.

P: Master axis (first spindle) rotation ratio

Range of command value: Integer 1 to 999

(The direction in which the master axis rotates depends on the commands (such as M03 and M04) issued under ordinary spindle control.)

Q: Polygon synchronization axis (second spindle) rotation ratio

Range of command value: Integers 1 to 999 and –1 to –999

(The direction in which the polygon synchronization axis rotates depends on the algebraic sign of the Q value, except when bit 1 (GDRC) of parameter No. 7603 = 1, in which case the polygon synchronization axis rotates in the same direction as the first spindle. In this case, Q cannot take a negative value.)

R: Relative phase difference between the master and polygon synchronization axes

The range of command value and the increment system are the same as for the rotation axis. However, the angle to rotate through is in 360/4096 degrees units.

(R is omissible. If it is not specified at all, the phase difference is assumed to be 0. If bit 5 (PCOF) of parameter No. 7602 = 1 to disable phase control, the R command is ignored, but no alarm condition is assumed.)

The G51.2 command is modal. Once specified, the P, Q, and R values stay unchanged until another G51.2 is issued to change them or polygon synchronized mode is released.

The S command issued to the first spindle during polygon synchronization mode specifies that the second spindle be used as a polygon synchronization axis and rotates at a speed of S × Q/P with a phase difference of R.

• Release command

◆ G50.2

This command releases the polygon synchronization mode. This mode is released also when:

(1) Reset
(That mode is not released by bit 0 (RPLM) of parameter No. 7603.)

(2) Power is turning off.

(3) An alarm condition occurs in the spindle control unit, and the serial spindle control unit stops in an emergency on the PMC signals *ESPA<G0071#1> and *ESPB<G0075#1>.

(4) P/S alarm 218, 219, or 221 occurs

• Cautions for using commands

G51.2 and G50.2 must be issued separately from other commands.

In a G51.2 issued to enter the polygon synchronization mode, R is omissible, but P and Q are required.

After a G51.2 is issued to enter the polygon synchronization mode, changing modal values of P, Q, and R requires another G51.2. In this case, R can be specified separately from P and Q. However, P and Q must be specified together even if only one of them is changed.

• Spindle operation during the spindle–spindle polygon synchronization mode

When a G51.2 is issued to start the spindle–spindle polygon synchronization mode, the speed of the spindle (polygon synchronization axis) is changed to Q/P times the speed of the first spindle (master axis) to achieve a speed ratio of P:Q, and phase adjustment is performed.

(If no S command is issued to the first spindle after a G51.2, the previous S command remains effective.)

If the spindle is in an acceleration, deceleration, or phase adjustment state, synchronization at a rotation ratio of P:Q is not guaranteed. So, it is necessary to control SAR<G0029#4> by checking the speed arrival signal PSAR<F0063#2> for polygonal turning with two spindles or to allow sufficient time in the program.

The method to specify the spindle speed during the polygon synchronization mode is the same as for ordinary modes. However, each time an S command is issued to the first spindle during polygon synchronization mode, or a rotation ratio P:Q or phase value R command is re–specified during the polygon synchronization mode, phase adjustment is performed after speed control. In addition, speed commands (such as under multi-spindle control or spindle output control for the second spindle by the PMC) for the second spindle become ineffective.

If the specified polygon synchronization axis speed (S × Q/P for the first spindle at S rpm) exceeds the clamp speed specified in parameter No. 7621, the polygon synchronization axis speed is clamped, and P/S alarm No. 5018 is issued.

Each time the spindle speed command for the first spindle changes or P and Q are re–specified in a G51.2, the clamp speed is checked to determine whether to issue P/S alarm No. 5018.
Note that a reset can clear the alarm with the speed clamped.
(A rotation speed ratio of P:Q cannot be maintained with the speed clamped. Bit 2 (QCL) of DGN No. 471 indicates whether the speed is clamped.)

- **PMC sequence**

Although this function is based on the G–code system, it is necessary to add or change PMC ladder sequences because control on the part of the spindle is also required (See signals in Section 6.10.2).

- **Example of polygonal turning with two spindles**

This example of polygonal turning with two spindles produces a square using single–edged cutting tools (for roughing and finishing).

. M○□; T○○ΔΔ ;	Step 1.	Mount a roughing tool on the polygon synchronization axis (second spindle as tool rotation axis).
G00 X100. Z20. M03 S1000 ; . .	Step 2.	Rotate the workpiece (with the first spindle as the master axis at 1000 rpm)
G51. 2 P1 Q2 ;	Step 3.	Start rotating the tool. Energize the second spindle in response to the polygon synchronization under way signal using the PMC ladder. After accelerating the second spindle as the polygon synchronization axis to 2000 rpm, perform phase adjustment (Execute R0 to omit an R value.). By checking PSYC<F0063#7>, the PMC ladder can detect when the polygon synchronization mode is entered. During the polygon synchronization mode, the PMC ladder controls SAR<G0029#4> based on the speed arrival signal PSAR<F0063#2> during the polygon synchronization mode.
G01 X80. F10. ; . .	Step 4.	Starts cutting along the X–axis after SAR<G0029#4> becomes logical 1 in signal control at step 3.
G04 P4000 ; .	Step 5.	Polygonal turning (roughing 1)
G00 X100. ; .	Step 6.	Retract the tool along the X–axis.
G51. 2 R180 ; .	Step 7.	Change the phase by 180 degrees.
[Repeat steps 4, 5, and 6.] .	Step 8.	Polygonal turning (roughing 2)
G50. 2 ; MΔ□ ; TΔΔ □□ ; .	Step 9.	Release the polygon synchronization mode. Change to a finishing tool.

G51. 2 P1 Q2 ; S2000; . .	Step 10.	Change the spindle speed for finishing (master axis at 2000 rpm and polygon synchronization axis at 4000 rpm with a phase difference of 0).
[Repeat steps 4, 5, and 6.] .	Step 11.	Polygonal turning (finishing 1)
[Repeat step 7.] .	Step 12.	Change the phase by 180 degrees.
[Repeat steps 4, 5, and 6.] . .	Step 13.	Polygonal turning (finishing 2)
G50. 2 ; . . .	Step 14.	Release the polygon synchronization mode. The polygon synchronization axis (second spindle as tool rotation axis) stops. The first spindle rotates at a speed specified by an S command.
M05; . .	Step 15.	The first spindle stops (end).

● Diagnosis display (DGN)

For polygonal turning with two spindles, the following information is displayed on the diagnosis display screen.

Polygonal turning with two spindles Indication of information about the polygon synchronization mode

DGN	#7	#6	#5	#4	#3	#2	#1	#0
470	SC0	LGE		SCF			PST	SPL

SPL Spindle–spindle polygon synchronization under way

PST Spindle–spindle polygon synchronization mode being activated

#2 Spindle–spindle polygon synchronization mode released

#3 Spindle speed being changed during spindle–spindle polygon synchronization mode

SCF Spindle speed changed during spindle–spindle polygon synchronization mode

#5 → Not used

LGE The loop gain is different between the spindles during spindle–spindle polygon synchronization mode.

SC0 Actual speed command is 0 during spindle–spindle polygon synchronization mode.

CAUTION

1 DGN indicates the loop gain because this function requires that both spindles be controlled with the same loop gain. However, no alarm is issued even if the loop gain is different between the spindles.
(For the serial spindle control unit, the parameters used are changed according to the state of the CTH1 and CTH2 signals.)

2 SC0 is not a value specified by the program. It is set to 1 under any of the following conditions:
   1. When the S command value is adjusted according to the signals related to spindle control, SSTP<G0029#6> and SOV0– SOV7<G0030> and the signal related to multi–spindle control <G0027>, the result is 0.
   2. The S command value is smaller than the spindle control resolution (the result of multiplying the S command value by a value of 4095/(maximum spindle speed) is less than 1).
      The S command value is specified by SIND control <G0032, G0033>, and it is 0.

   If SC0 = 1, the spindle speed becomes 0 and bit 0 of DGN No. 471 becomes 1. In this case, the polygon synchronization rotation ratio is impractical, but P/S alarm No. 5018 does not occurs, because it is regarded as the result of the command.

NOTE

1 The normal state during spindle–spindle polygonal turning is: SPL = 1, SCF = 1, #1 = 0, #2 = 0, and #3 = 0

2 If only PST becomes 1, but no change occurs, and the program stops in a block containing a G51.2 command, the speed of an spindle does not reach the targeted polygon synchronization speed, for example, because bit 7 (PST) of parameter No. 7603 = 0 keeps the spindle from being energized.

3 When the speed is changed during polygon synchronization mode, LGE is set to 1 if the spindle synchronization control loop gain used by the serial spindle control unit is different between the first and second spindles.

Polygonal turning with two spindles Indication of causes for P/S alarms 5018 and 218

DGN	#7	#6	#5	#4	#3	#2	#1	#0
471	NPQ	PQE		NSP	SUO	QCL	PCL	

#0 to #3 → Causes for P/S alarm No. 5018
P/S alarm No. 5018 is cleared by a reset, but the indication of its causes remains until the causes are cleared or the polygon synchronization mode is released.

#4 to #7 → Causes for P/S alarm No. 218
When P/S alarm No. 218 occurs, the polygon synchronization mode is released, but the indication of its causes remains until the alarm is cleared by a reset.

#0 The specified speed is too low during spindle–spindle polygon synchronization mode. (The unit of speed calculated internally becomes 0.)

PCL The first spindle (polygon synchronization master axis) is clamped.

QCL The second spindle (polygon synchronization axis) is clamped.

SUO The specified speed is too high during the spindle–spindle polygon synchronization mode. (It is clamped to the upper limit calculated internally.)

NSP A spindle necessary for control is not connected. (For example, there is not a serial spindle or the second spindle.)

#5 When bit 1 (QDRC) of parameter No. 7603 = 1, a negative value is specified at Q.

PQE In a G51.2, either P or Q has a value out of the specifiable range.
Or, P and Q are not specified as a pair.

NPQ In a G51.2, R is specified when P and Q have not been specified at all, or none of P, Q, and R has been specified.

CAUTION

#0 becomes 1 also when the specified spindle speed is 0 (DGN 470#7 = 1). In this case, however, P/S alarm No. 5018 is not issued (because the command is 0). When DGN 470#7 = 0 and DGN 471#0 = 1, P/S alarm No. 5018 occurs. Normally this does not occur with speed at which the spindle can rotate.

NOTE

1 PCL indicates that the master axis has received a command with a speed that is higher than the value specified by the maximum first spindle speed parameters (No. 3741 to 3744) and is clamped to that speed. PCL will not become 1 as long as the first spindle is connected correctly.
2 QCL becomes 1, when the second spindle (polygon synchronization axis) receives a command with a polygon synchronization speed that is higher than the value specified in parameter No. 7621 and is clamped at that speed.
3 SUO occurs, if a result of (speed specified for the first spindle)/(value specified at P) is higher than 59998. In other words, the first spindle must rotate at a speed lower than 59998 rpm assuming P = 1.

Indication of values specified during the spindle–spindle polygon synchronization mode

DGN 474	Rotation ratio for the master axis during the spindle–spindle polygon synchronization mode (P command value)

This indication is the current rotation ratio (P command value) of the master axis (first spindle) during the spindle–spindle polygon synchronization mode.

DGN 475	Rotation ratio for the polygon synchronization axis during the spindle–spindle polygon synchronization mode (Q command value)

This indication is the current rotation ratio (Q command value) of the polygon synchronization axis (second spindle) during the spindle–spindle polygon synchronization mode.

DGN 476	Phase difference between the two spindles under spindle–spindle polygon synchronization control (R command value)

This indication is the current phase value (R command value) specified during the spindle–spindle polygon synchronization mode. (The unit of measurement is the least increment system for the rotation axis of the machine.)

However, if bit 5 (RDGN) of parameter No. 7603 = 1, the indication is the amount of shifting specified for the serial spindle (number of pulses after conversion is performed assuming 360 degrees = 4096 pulses).

Indication of the actual speed of each spindle during the spindle–spindle polygon synchronization mode

DGN 477	Actual master axis speed (rpm) during the spindle–spindle polygon synchronization mode

This indication is the actual speed of the master axis (first spindle) during the spindle–spindle polygon synchronization mode.

DGN 478	Actual polygon synchronization axis speed (rpm) during the spindle–spindle polygon synchronization mode

This indication is the actual speed of the polygon synchronization axis (second spindle) during the spindle–spindle polygon synchronization mode.

NOTE

The indications of DGN No. 477 and 478 vary because of no sampling being performed. Consider these DGN values only guidelines.

Signal

Polygon synchronization under way signal PSYN<F063#7>

[Classification] Output signal

[Function] Informs the PMC that the system is in the polygon synchronization mode.

[Output condition] The polygon synchronization mode command (G51.2) sets this signal to logical “1”. It stays at “1” as long as the system is in the polygon synchronization mode. It is turned to “0” when the polygon synchronization mode is cleared (G50.2 command or a reset). It stays at “0” when the system is not in the polygon synchronization mode.

> **NOTE**
> The same address is used for this signal in both polygonal turning (using the servo axis) and the polygonal turning with two spindles.

Polygon spindle stop signal *PLSST<G038#0>

[Classification] Input signal

[Function] This function is enabled when bit 7 (PST) of parameter No. 7603 = 1. This signal is used to stop the spindle during the polygonal turning mode with two spindles.

“0” = polygon spindle stop
“1” = polygon spindle operable

During the polygonal turning mode with two spindles, the spindles are controlled with a positional loop set up. When issuing a spindle stop command (like M05) to deenergize the spindle, it is necessary to specify S = 0 using *SSTP<G00296#6>. Otherwise the stop command remains effective even after the spindle is deenergized. This error accumulates and causes a dangerous behavior of the spindle when it is energized again. This signal is intended to inhibit distribution of the S command to the spindle if it cannot be set to 0 while the spindle is deenergized. In such a case, the signal should be used in step with the energizing state of the polygon spindle.

Spindle polygonal speed arrival signal PSAR<F063#2>

[Classification] Output signal

[Function] Informs the PMC that the spindle has reached its constant–speed for polygon synchronization during polygonal turning with two spindles.

[Output condition] During polygonal turning mode with two spindles, whether the constant–speed is reached for polygon synchronization is output as shown below:

"0" = not reached (during phase change or acceleration/deceleration under way)

"1" = reached

During the polygon control mode, this signal becomes logical "1" when the speed of each spindle reaches the acceptable level specified in parameter No. 7631 and remains there for a period specified in parameter No. 7632.

If the speed of either spindle goes off the acceptable level, or a change is made to the S command, the signal returns to logical "0" and begins monitoring the above condition.

When this signal is "0", the specified speed ratio and phase are not guaranteed for polygonal turning. If the signal is confirmed before actual turning is started, however, the operation is more efficient than when a dwell command (like G04) is used to allow wait time.

Master axis not arrival signal PSE1<F063#0> Polygon synchronization axis not arrival signal PSE2<F063#1>

[Classification] Output signal

[Function] Informs the PMC whether the actual speed of each spindle has reached the specified speed during polygonal turning mode with two spindles.

[Output condition] During polygonal turning mode with two spindles, whether each spindle has reached the polygon synchronization speed is output as shown below:

"0" = reached

"1" = not reached (during phase change or acceleration/deceleration under way)

During the polygon control mode, this signal becomes logical "1" when the speed of master axis (first spindle) and polygon synchronization axis (second spindle) does not reach the acceptable level specified in parameter No. 7631.

- **PMC sequence**

When a G51.2 is issued to put the system in the polygon synchronization mode, the polygon synchronization under way signal PSYN<F063#7> becomes on.

Set up a PMC sequence for the polygon synchronization mode by monitoring this signal with a PMC ladder.

There are two control methods, (A) and (B), to control energizing of the spindle. First select (A) or (B) and creates a PMC sequence according to the selected method.

Method A

(A) Energize the first and second spindles automatically during the G51.2 mode.
Basically, do not discontinue energizing during this mode.

In this case, keep bit 7 (PST) of parameter No. 7603 = 0.

In the PMC sequence, detect when the polygon synchronization under way signal PSYN<F063#7> changes from 0 to 1, then energize the first and second spindles.

The NC stops at the G51.2 command block which puts the system in the polygon synchronization mode, and remains there until the spindle reaches the polygon synchronization speed.

Also, keep the spindle energized, for example, by preventing it from receiving a spindle stop command (like M05) for ordinary spindle control while PSYN<F063#7> = 1. Basically, deenergize the spindle when the polygon synchronization under way signal PSYN<F063#7> changes from 1 to 0.

Method B

(B) Control the energizing of the spindle using M codes even during the G51.2 mode. Alternatively, deenergize the spindle even during the G51.2

In this case, keep bit 7 (PST) of parameter No. 7603 = 1.

This parameter setting enables use of the spindle stop signal *PLSST<G038#0> during the polygon synchronization mode. It also makes the G51.2 command block stop waiting for the spindle to reach the polygon synchronization speed.

In the PMC sequence, while the polygon synchronization under way signal PSYN<F063#7> = 1, set *PLSST<G038#0> to 1 after confirming both first spindle and second spindles are energized.

If either spindle has been deenergized, reset *PLSST<G038#0> to 0.

When the polygon synchronization under way signal PSYN <F063#7> changes from 1 to 0, basically keep *PLSST <G038#0> at 0. (This is intended to keep *PLSST <G038#0> from becoming 1 in a deenergized state when the polygon synchronization mode is entered again.)

When *PLSST <G038#0> changes from 0 to 1 during the polygon synchronization mode, the spindle is accelerated from a stop state to the specified speed and placed under phase control, even if the S command has not been changed.

Sequence common to methods (A) and (B)

Regardless of whether the method you use is (A) or (B), set up the PMC sequence as follows:

- Do not use the SFR/SRV signal to switch the rotation direction of the first spindle. Instead, fix the energizing method of the spindle at SFR and change the polarity of the command. (To change the polarity of the command, issue M03/M04 with bit 7 (TCW) of parameter No. 3706 = 1, or control SGN<G033#5> with SSIN<G033#6> = 1.)

Also fix the energizing method of the second spindle at SFR.

- To check whether the spindle has reached its constant–speed, control SAR<G029#4> using PSAR<F063#2>.

  PSAR<F063#2> can be used to check whether both spindles have reached the conditions specified in parameter Nos. 7631 and 7632 after completion of phase control. Checking the speed arrival signal for each spindle (SARA<F045#3> and SARB<F049#3>) cannot guarantee proper cutting start conditions, because phase control may occur afterward.

  The specification (parameter No. 3740) of time allowed before the spindle constant–speed reached signal is checked remains valid until after execution of the G51.2 command.

  If you are not using SAR<G029#4> as the cutting feed start condition, start cutting after allowing time using the program (like G04) for both spindles to reach their constant speeds, when the polygon synchronization mode is entered and each time the S command is changed during the polygon synchronization mode.

- It is impossible to rotate the second spindle separately from the first spindle and to use the spindle orientation function (ORCMA<G070#6>, ORCMB<G074#6>) during polygon synchronization mode with two spindles. Basically, do not perform gear change, tool change, or workpiece change during the polygon synchronization mode. Have the PMC reject such commands and output a message prompting to release the two–spindle polygon synchronization mode, as required.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G038								*PLSST

	#7	#6	#5	#4	#3	#2	#1	#0
F063	PSYN					PSAR	PSE2	PSE1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7602			COF	HST	HSL	HDR	SNG	MNG

[Data type] Bit

MNG The rotational direction of the master axis (first spindle) in the spindle–spindle polygon turning mode is:

0: Not reversed.
1: Reversed.

SNG The rotational direction of the polygon synchronization axis (second spindle) in the spindle–spindle polygon turning mode is:

0: Not reversed.
1: Reversed.

HDR When phase control is exercised in spindle–spindle polygon turning mode (COF = 0), the phase shift direction is:

0: Not reversed for phase synchronization.
1: Reversed for phase synchronization.

> **NOTE**
> Use MNG, SNG, and HDR when the specified rotational direction of the master axis or polygon synchronization axis, or the specified phase shift direction is to be reversed in spindle–spindle polygon turning mode.

HSL When phase control is exercised in spindle–spindle polygon turning mode (COF = 0), this parameter selects the spindle that is subject to a phase shift operation for phase synchronization:

0: The polygon synchronization axis (second spindle) is selected.
1: The master axis (first spindle) is selected.

HST When phase control is applied in spindle–spindle polygon turning mode (COF = 0), and spindle–spindle polygon turning mode is specified:

0: Spindle–spindle polygon turning mode is entered with the current spindle speed maintained.
1: Spindle–spindle polygon turning mode is entered after the spindle is stopped.

> **NOTE**
> This parameter can be used, for example, when single–rotation signal detection cannot be guaranteed at an arbitrary feedrate because a separate detector is installed to detect the spindle single–rotation signal, as when a built–in spindle is used. (When bit 7 of parameter No. 4016 for the serial spindle is set to 1, together with this parameter, a single–rotation signal detection position in spindle–spindle polygon turning mode is guaranteed.)

COF In spindle–spindle polygon turning mode, phase control is:

0: Used.
1: Not used.

> **CAUTION**
> When the use of phase control is not selected, the steady state is reached in a shorter time because phase synchronization control is not applied. Once steady rotation is achieved, however, polygonal turning must be completed without changing the steady state. (If the rotation is stopped, or the rotational speed altered, polygonal turning is disabled because of the inevitable phase shift.)
> Setting this parameter to 1 does not issue an alarm on the R command (phase command) in the same block as the G51.2. It is only ignored.

	#7	#6	#5	#4	#3	#2	#1	#0
7603	PST		RDG				QDR	RPL

[Data type] Bit

RPL Upon reset, spindle–spindle polygon turning mode is:

0 : Released.
1 : Not released.

QDR The rotational direction of the polygon synchronization axis:

0 : Depends on the sign (+/−) of a specified value for Q.
1 : Depends on the rotational direction of the first spindle. (If − is specified for Q, P/S alarm No. 218 is issued.)

RDG On the diagnosis screen No. 476, for spindle–spindle polygon phase command value (R), displays:

0 : The specified value (in the increment system for the rotation axis).
1 : The actual number of shift pulses.

NOTE
A phase command is specified in address R, in units of degrees. For control, the actual shift amount is converted to a number of pulses according to the conversion formula: 360 degrees = 4096 pulses. This parameter switches the display of a specified value to that of a converted value.

PST The polygon spindle stop signal *PLSST (bit 0 of G038) is:

0 : Not used.
1 : Used.

7621	Maximum allowable speed for the tool rotation axis (polygon synchronization axis)

[Data type] Word

[Unit of data] rpm

[Valid data range] For polygon turning with two spindles:
Set a value between 0 and 32767, but which does not exceed the maximum allowable speed, as determined by the performance of the second spindle and other mechanical factors.

This parameter sets the maximum allowable speed of the tool rotation axis (polygon synchronization axis).
If the speed of the tool rotation axis (polygon synchronization axis) exceeds the specified maximum allowable speed during polygon turning, the speed is clamped at the maximum allowable speed. When the speed is clamped at a maximum allowable speed, however, synchronization between the spindle and tool rotation axis (polygon synchronization axis) is lost. And, when the speed is clamped, P/S alarm No. 5018 is issued.

7631	Allowable spindle speed deviation level in spindle–spindle polygon turning

[Data type] Byte

[Unit of data] rpm

[Valid data range] 0 to 255

[Standard setting value] 1 to 10

This parameter sets the allowable level of deviation between the actual speed and specified speed of each spindle in spindle–spindle polygon turning. The value set with this parameter is used for both the master axis and polygon synchronization axis.

.

7632	Steady state confirmation time duration in spindle polygon turning

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 32767

This parameter sets the duration required to confirm that both spindles have reached their specified speeds in spindle–spindle polygon turning. If the state where the speed of each spindle is within the range set with parameter No. 7631, and has lasted at least for the duration specified with parameter No. 7632, the spindle polygon speed arrival signal PSAR (bit 2 of F0063) is set to 1.

Alarm and message

Number	Message	Description
218	NOT FOUND P/Q COMMAND IN G251	The G51.2 block does not contain P or Q, or a specified value is invalid. The causes of this alarm are detailed in DGN No. 471. (See below.) DGN No. 471#7 NPQ → When P and Q are not specified at all, R is specified. Alternatively, none of P, Q, and R has been specified. DGN No. 471#6 PQE → P or Q is out of the valid data range. Alternatively, P and Q are not specified as a pair. DGN No. 471#5 → A negative value is specified at Q when bit 1 (QDRC) of parameter No. 7603 = 1. DGN No. 471#4 NSP → There is no spindle necessary for control. (The spindle is not a serial spindle, or there is not the second spindle, etc.)
219	COMMAND G250/G251 INDEPENDENTLY	G51.2/G251 and G50.1/G250 are specified together with other commands in the same block. Correct the program.
221	ILLEGAL COMMAND IN SYNCHRO	An attempt was made to perform polygon synchronization operation together with Cs contouring control or balance cutting. Alternatively the program issued a command for spindle–spindle polygon synchronization mode, when the spindle is under spindle synchronization control, Cs contouring control, spindle positioning control, or rigid tapping control, etc. Correct the program.
5018	POLYGON SPINDLE SPEED ERROR	The specified rotation ratio cannot be maintained during the G51.2 mode, because the speed of the spindle or polygon synchronization axis exceeds the clamping value or is too low. The causes of this alarm are detailed in DGN No. 471. (See below.) DGN No. 471#3 SUO → The specified speed is too high. DGN No. 471#2 QCL → The polygon synchronization axis (second spindle) is clamped. DGN No. 471#1 PCL → The master axis (first spindle) is clamped. DGN No. 471#0 → The specified speed is too low.

(Remark)

· In a properly connected machine, P/S alarm No. 5018 does not basically occur for other than a reason that the polygon synchronization axis is clamped. (See descriptions of DGN for details.)

· To the contrary to P/S alarm No. 221, P/S alarm No. 194 occurs if another NC control spindle function is specified during the two–spindle polygon synchronization mode.

Caution

CAUTION

1 The maximum spindle speed for each gear stage (No. 3741 to 3744) must be specified correctly according to the model of the machine. In addition, ordinary spindle connections must have been terminated.

2 This function uses the one–rotation signal for the spindle as a reference point for phase adjustment.
When a built–in sensor is used, and there are gears between the spindle and spindle motor, it is necessary to install a detector on the spindle separately to take a one–rotation signal from the spindle. If the detector does not guarantee detection of a correct position from arbitrary speed, set bit 4 (PHST) of parameter No. 7602 and bit 7 of serial spindle parameter No. 4016 to 1. This setting reduces the spindle speed automatically down to 0 for spindle position detection each time the spindle–spindle polygonal turning mode is entered, thus guaranteeing a correct phase relationship during spindle–spindle polygon synchronization mode.

3 This function uses the spindle synchronization function for serial spindles. (However, it does not require the spindle synchronization option for the CNC.) You may need to specify the relevant serial spindle parameters (such as Nos. 4032 to 4035).
Specify the same serial spindle loop gain for both spindles. If the same serial spindle loop gain is not used for both spindles, polygonal turning may not be accurate.
If an attempt is made to perform spindle–spindle polygonal turning, DGN 470#6 LGE becomes 1. (No alarm is issued.)

4 Before using the polygon synchronization mode, place both first and second spindles in the spindle control mode. The polygon synchronization mode cannot be used if they are already in other modes (Cs contouring control mode or spindle orientation mode, etc.).

5 During the polygon synchronization mode, the speed of the second spindle cannot be controlled independently of the speed of the first spindle.
During the polygon synchronization mode, the spindle orientation function (ORCMA<G070#6>, ORCMB<G074#6>) cannot be used for either the first spindle or second spindle. Therefore, gear, tool or workpiece change is basically unusable during the polygon synchronization mode.

CAUTION

6 During polygon synchronization mode, speed change and phase adjustment are performed each time the spindle speed is changed. Therefore, this mode cannot be used together with a function that causes continuous spindle speed change (such as G96 constant surface speed control)

7 During the polygon synchronization mode, the rotation ratio between the master axis and polygon synchronization axis is controlled with priority. Therefore, the difference between the master axis speed and S command value may become larger than during ordinary spindle control. (The master axis speed may be up to 2 rpm lower than specified.)

Note

NOTE

1 During the polygon synchronization mode, phase control is performed in the least command increment of 36/4096 = 0.08789...(degrees) in reference to the one–rotation signal for each spindle. Actually, a command value out of a range from 0 to 359.999... is meaningless because the relative phase is controlled within one rotation of each spindle. However, this function does not limit the R command value and the CNC converts the R command value to a value below 360 degrees.

2 The G51.2 command during conversational function is equivalent to the G50.2 command. (The system does not enter the polygon synchronization mode. If it is already in the polygon synchronization mode, release it using a G51.2 command.)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.19.1	POLYGONAL TURNING
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.19.1	POLYGONAL TURNING

6.11 NORMAL DIRECTION CONTROL (M SERIES)

General

When a tool with a rotation axis (C–axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C–axis is always perpendicular to the tool path (Fig. 6.11).

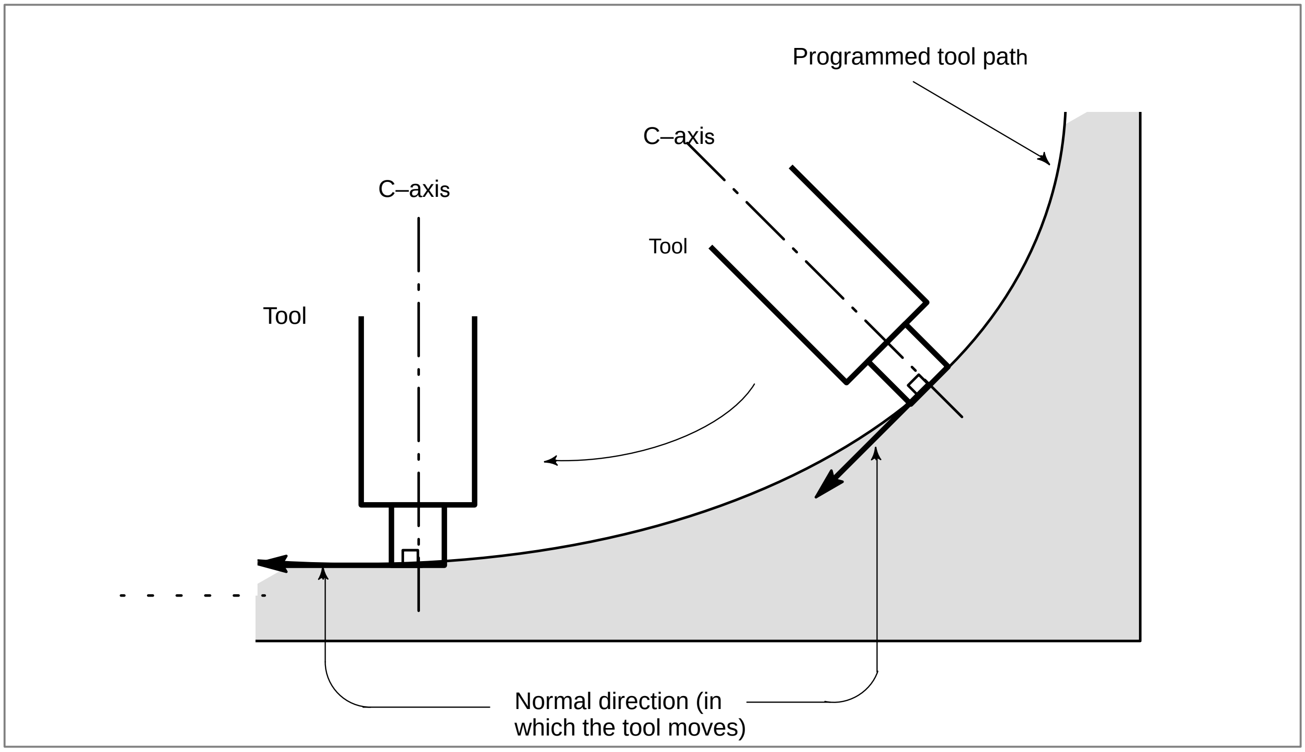


Fig. 6.11 Sample Movement of the tool

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X–and Y–axes in rapid traverse (G00) mode, the rapid traverse rate is applied.

If the federate of the C axis exceeds the maximum cutting feedrate of the C axis specified to parameter No. 1422, the federate of each of the other axes is clamped to keep the federate of the C axis below the maximum cutting feedrate of the C axis.

Parameter

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

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

[Data type] Bit axis

[Valid data range] ROTx, ROSx Setting linear or rotation axis

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

NOTE

The rotation axis must be set to the normal direction control axis.

5480	Number of the axis for controlling the normal direction

[Data type] Byte

[Valid data range] 1 to the maximum control axis number

This parameter sets the control axis number of the axis for controlling the normal direction.

5481	Rotation feedrate of normal direction control axis

[Data type] Word

[Unit of data] 1 deg/min

[Valid data range] 1 to 15000

This parameter sets the feedrate of a normal direction control axis that is inserted at the start point of a block during normal direction control.

5482	Limit value that ignores the rotation insertion of normal direction control axis

[Data type] Two–word

[Unit of data]

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

[Valid data range] 1 to 99999999

The rotation block of a normal direction control axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting value. The ignored rotation angle is added to the next rotation insertion angle. The block insertion is then judged.

NOTE
1 No rotation block is inserted when 360 or more degrees are set.
2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation is 180 or more degrees.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

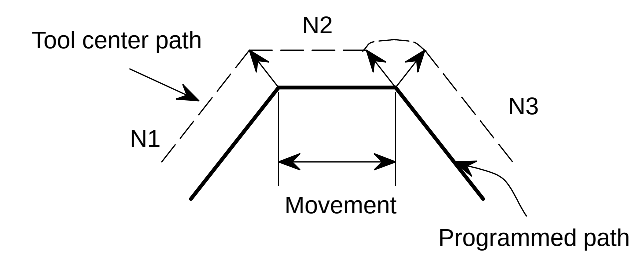
[Data type] Two–word

[Unit of data]

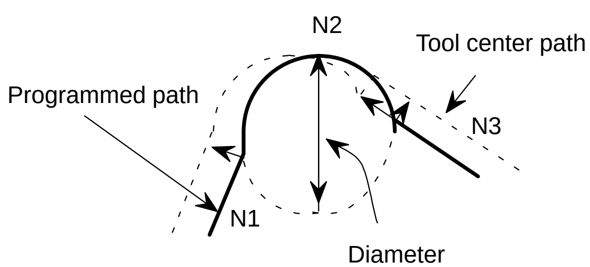
Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

This parameter sets the limit value of movement at the normal direction angle of a preceding block.



For straight line
Block N2 is machined with the tool being normal to block N1 when the movement of N2 in the figure on the left does not exceed the set value.



For arc
Arc N2 is machined with the tool being normal to block N1 when the arc diameter of N2 in the figure on the left does not exceed the setting value. A normal direction axis is not controlled to move in the normal direction according to the arc movement.

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

> **NOTE**
> To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead.

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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.14.11	Normal Direction Control
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.14.9	Normal Direction Control

6.12 EXPONENTIAL INTERPOLATION (M SERIES)

General

Exponential interpolation exponentially changes the rotation of a workpiece with respect to movement on the rotary axis. Furthermore, exponential interpolation performs linear interpolation with respect to another axis. This enables tapered groove machining with a constant helix angle (constant helix taper machining). This function is best suited for grooving and grinding tools such as end mills.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5630								
								SPN

[Data type] Bit type

SPN The amount of linear axis division (span value) in exponential interpolation is:

0 : Specified with parameter No. 5643.
1 : Specified using address K in a block containing G02.3/G03.3. When address K is not specified, the value set with parameter No. 5643 is used.

5641	
	Linear axis number subject to exponential interpolation

[Data type] Byte type

[Valid data range] 1 to number of controlled axes

This parameter sets the ordinal number, among the controlled axes, for the linear axis to which exponential interpolation is applied.

5642	
	Rotation axis number subject exponential interpolation

[Data type] Byte type

[Valid data range] 1 to number of controlled axes

This parameter sets the ordinal number, among the controlled axes, for the rotation axis to which exponential interpolation is applied.

5643	Amount of linear axis division (span value) in exponential interpolation

[Data type] 2–word type

[Valid data range]

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 amount of linear axis division in exponential interpolation when bit 0 (SPN) of parameter No. 5630 is set to 0.

Alarm and message

No.	Message	Contents
5060	ILLEGAL PARAMETER IN G02.3/G03.3	Parameter setting is illegal. No. 5641 (setting of the linear axis) is not specified. No. 5641 specifies an axis other than a linear axis. No. 5642 (setting of the rotation axis) is not specified. No. 5642 specifies an axis other than a rotation axis. The CNC cannot control the linear or rotation axis (the value of No. 1010 is exceeded).
5061	ILLEGAL FORMAT IN G02.3/G03.3	The command for exponential interpolation (G02.3/G03.3) contains a format error. Address I, J, or K is not specified. Addresses I, J, and K are 0.
5062	ILLEGAL COMMAND IN G02.3/G03.3	The command for exponential interpolation (G02.3/G03.3) contains an illegal value. The specified value is not suitable for exponential interpolation (for example, a negative value is subject to ln).

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.11	Exponential interpolation

6.13 SMOOTH INTERPOLATION (M SERIES)

General

Either of two types of machining can be selected, depending on the program command.

- For those portions where the accuracy of the figure is critical, such as at corners, machining is performed exactly as specified by the program command.
- For those portions having a large radius of curvature where a smooth figure must becreated, points along the machining path are interpolated with a smooth curve, calculated from the polygonal lines specified with the program command (smooth interpolation).

Smooth interpolation can be specified when CDSP (bit 5 of parameter No. 8485) is set to 1 in high–speed contour control mode (between G05 P10000 and G05 P0). Smooth interpolation performed in high–speed contour control mode is described below. For details of high–speed contour control, see subsec 7.1.14.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
8485			CDS					

[Data type] Bit

CDS 0 : Disables smooth interpolation in HPCC mode.
1 : Enables smooth interpolation in HPCC mode.

To apply smooth interpolation, be sure to set this parameter to 1.

8486	Maximum travel distance of a block where smooth interpolation is applied

[Data type] Two–word

[Unit of data] Least input increment (depending on the set reference axis)

[Valid data range] 0 to 99999999

This parameter specifies a block length used as a reference to decide whether to apply smooth interpolation. If the line specified in a block is longer than the value set in the parameter, smooth interpolation will not be applied to that block. This parameter can be used, for example, to specify the maximum line length of a folded line to which a metal die workpiece is approximated with some tolerance.

Alarm and message

Number	Message	Description
5085	SMOOTH IPL ERROR 1	A block for specifying smooth interpolation contains a syntax error.

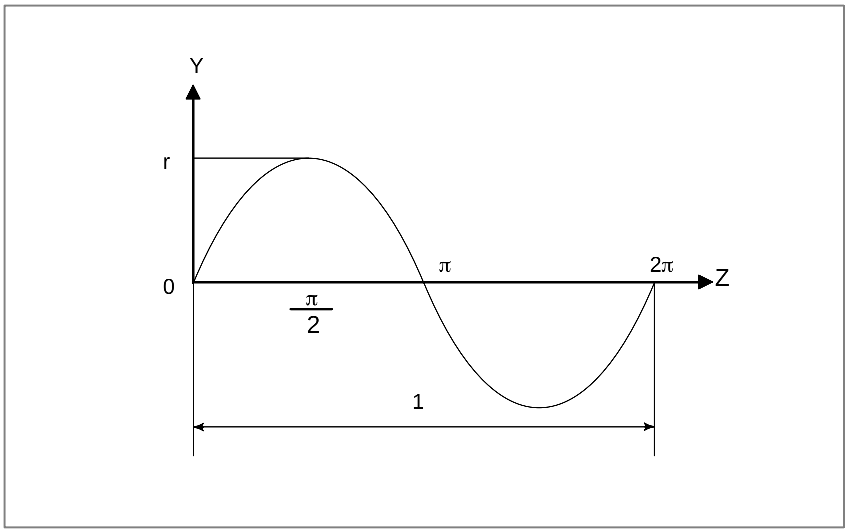
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.12	Smooth interpolation

6.14 HYPONENTIAL AXIS INTERPOLATION

General

In helical interpolation, when pulses are distributed with one of the circular interpolation axes set to a hypothetical axis, sine interpolation is enable.
When one of the circular interpolation axes is set to a hypothetical axis, pulse distribution causes the speed of movement along the remaining axis to change sinusoidally. If the major axis for threading (the axis along which the machine travels the longest distance) is set to a hypothetical axis, threading with a fractional lead is enabled. The axis to be set as the hypothetical axis is specified with G07.



Reference item

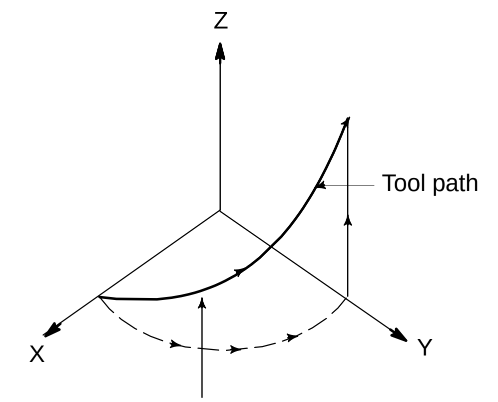
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.14	Hyponential axis interpolation
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.7	Hyponential axis interpolation

6.15 HELICAL INTERPOLATION B (M SERIES)

General

Helical interpolation B moves the tool helically. This interpolation can be executed by specifying the circular interpolation command together with up to four additional axes in simple high–precision contour control mode.

Basically, the command can be specified by adding two movement axes to a standard helical interpolation command. Address F should be followed by a tangential velocity, determined by also taking movement along the linear axes into consideration.



The feedrate equals the tangential velocity determined by also taking movement along the linear axes into consideration.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.6	Helical interpolation B

6.16 SPIRAL INTERPOLATION, CONICAL INTERPOLATION (M SERIES)

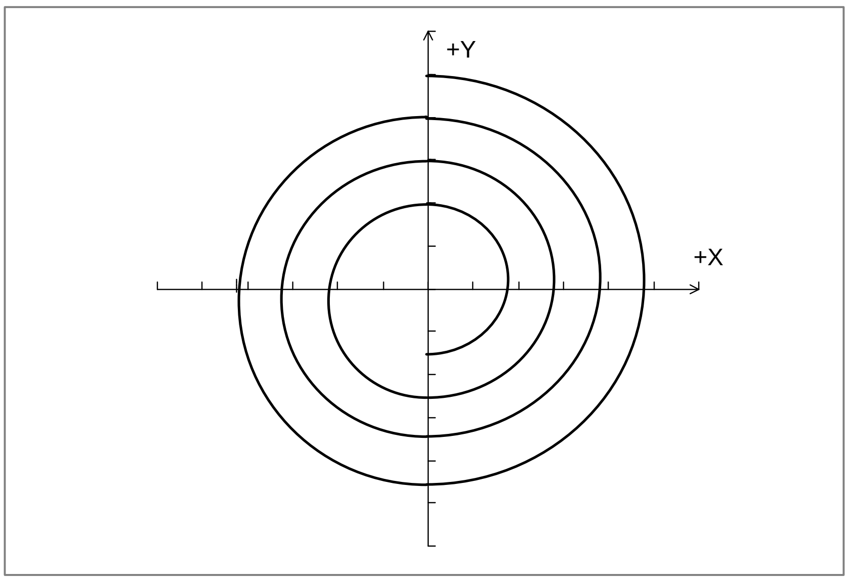
General

Spiral interpolation is enabled by specifying the circular interpolation command together with a desired number of revolutions or a desired increment (decrement) for the radius per revolution.

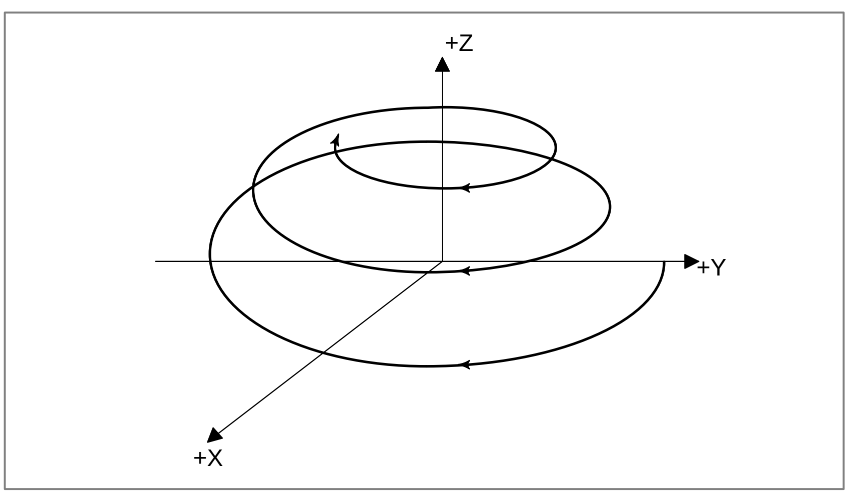
Conical interpolation is enabled by specifying the spiral interpolation command together with one or two additional axes of movement, as well as a desired increment (decrement) for the position along the additional axes per spiral revolution.

Spiral interpolation and conical interpolation do not support bell–shaped acceleration/deceleration after interpolation for cutting feed.

- **Spiral interpolation**



- **Conical interpolation**



Parameter

3471	Allowable difference between the specified end point and that calculated from the increment (or decrement) and number of revolutions, for spiral or conical interpolation

[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

Sets the maximum allowable value for the difference (absolute value) between the specified end point and that calculated from the increment (or decrement) and number of revolutions, for spiral or conical interpolation.

3472	Minimum radius with which the actual feedrate is maintained for spiral or conical interpolation

[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] 1000 to 99999999 (metric input)
10000 to 99999999 (inch input)

If this parameter is set to 0 or a value that falls outside the valid data range, the minimum value in the valid data range is assumed.

Spiral and conical interpolation usually maintain a constant feedrate. Near the center of the spiral, however, the radius is very small, such that applying a constant feedrate would cause the angular velocity to become very high. To prevent this, the angular velocity is regulated to a constant value once the radius of the spiral has decreased to the value specified with the parameter. As a result, the actual feedrate decreases.

Alarm and message

Number	Message	Description
5122	ILLEGAL COMMAND IN SPIRAL	An invalid command has been specified for spiral or conical interpolation. The most likely causes are as follows: 1) L = 0 specified 2) Q = 0 specified 3) R/, R/, C specified 4) Height increment of 0 specified 5) More than three height axes specified 6) Height increment specified together with two height axes 7) Conical interpolation specified when the helical interpolation option is not being used 8) Q < 0 specified when the radius difference > 0 9) Q > 0 specified when the radius difference < 0 10) Height increment specified without specifying a height axis
5123	OVER TOLERANCE OF END POINT	The difference between the specified end point and calculated end point exceeds the allowable range (parameter No. 3471).
5124	CAN NOT COMMAND SPIRAL	Spiral or conical interpolation has been specified in any of the following modes: 1) Scaling 2) Programmable mirror image 3) Polar coordinate interpolation Alternatively, in cutter compensation mode C, the center coincides with the start or end point.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.7	Spiral interpolation, conical interpolation

6.17 NURBS INTERPOLATION (M SERIES)

General

Many computer–aided design (CAD) systems used to design metal dies for automobiles and airplanes utilize non–uniform rational B–spline (NURBS) to express a sculptured surface or curve for the metal dies.

This function enables NURBS curve expression to be directly specified to the CNC. This eliminates the need for approximating the NURBS curve with minute line segments. This offers the following advantages:

1. No error due to approximation of a NURBS curve by small line segments
2. Short part program
3. No break between blocks when small blocks are executed at high speed
4. No need for high–speed transfer from the host computer to the CNC

When this function is used, a computer–aided machining (CAM) system creates a NURBS curve according to the NURBS expression output from the CAD system, after compensating for the length of the tool holder, tool diameter, and other tool elements. The NURBS curve is programmed in the NC format by using these three defining parameters: control point, weight, and knot.

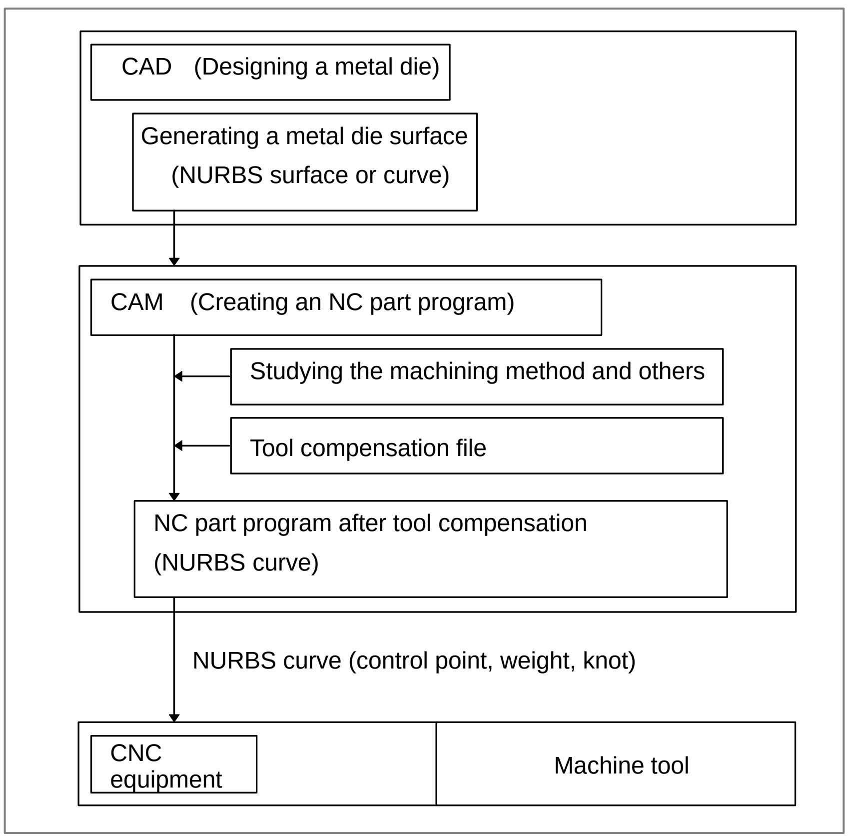


Fig. 6.17 NC part program for machining a metal die according to a NURBS curve

NURBS interpolation must be specified in high–precision contour control mode (between G05 P10000 and G05 P0). The CNC executes NURBS interpolation while smoothly accelerating or decelerating the movement so that the acceleration on each axis will not exceed the allowable maximum acceleration of the machine. In this way, the CNC automatically controls the speed in order to prevent excessive strain being imposed on the machine.

Alarm and message

Number	Message	Description
5115	SPL: ERROR	An illegal rank is specified.
		No knot is specified.
		An illegal knot is specified.
		Too many axes are specified.
		Other program error
5116	SPL: ERROR	A look–ahead block contains a program error.
		The knot does not increase at a constant rate.
		An inhibited mode is specified in NURBS interpolation mode.
5117	SPL: ERROR	The first NURBS control point is illegal.
5188	SPL: ERROR	An attempt was made to resume NURBS interpolation after manual intervention in manual absolute mode.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.13	NURBS interpolation

7 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL

7.1 FEEDRATE CONTROL

The feed functions control the feedrate of the tool. The following two feed functions are available:

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse rate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

Override can be applied to a rapid traverse rate or cutting feedrate using the override signal.

7.1.1 Rapid Traverse Rate

General

The positioning command (G00) positions the tool by rapid traverse.

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ ; Dimension word for the end point

In rapid traverse, the next block is executed after the specified rate becomes 0 and the servo motor reaches a certain range set by the parameter (No.1826) (in–position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse rate need be programmed.
The following overrides can be applied to a rapid traverse rate with the rapid traverse override signal:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
In addition, the use of the 1% rapid traverse override selection signal enables a rapid traverse override of between 0% and 100% to be applied in 1% steps.

Signal

Rapid traversing signal RPDO <F002#1>

[Function] This signal indicates that a move command is executed at rapid traverse.

[Output condition] “1” indicates that an axis starts moving after rapid traverse has been selected.
“0” indicates that an axis starts moving after a feedrate other than rapid traverse has been selected. This holds true for both automatic and manual operation modes.

NOTE

1 The rapid traverse in automatic operation includes all rapid traverses in canned cycle positioning, automatic reference point return, etc., as well as the move command G00. The manual rapid traverse also includes the rapid traverse in reference position return.

2 Once rapid traverse has been selected, this signal remains "1", including during a stop, until another feedrate has been selected and movement is started.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002							RPDO	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR		RFO			LRP	

[Data type] Bit

LRP Positioning (G00)

0 : Positioning is performed with non–linear type positioning so that the tool moves along each axis independently at rapid traverse.
1 : Positioning is performed with linear interpolation so that the tool moves in a straight line.

RFO When cutting feedrate override is 0% during rapid traverse,

0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

RDR Dry run for rapid traverse command

0 : Disabled
1 : Enabled

1420	Rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1424	Manual rapid traverse rate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 240000	30 – 100000
Inch machine	0.1 inch/min	30 – 96000	30 – 48000
Rotation axis	1 deg/min	30 – 240000	30 – 100000

Set the rate of manual rapid traverse for each axis when the rapid traverse override is 100% for each axis.

NOTE
If 0 is set, the rate set in parameter 1420 is assumed.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.2	Rapid traverse
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.5.2	Rapid traverse
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.2	Rapid traverse
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.5.2	Rapid traverse

7.1.2 Cutting Feedrate Clamp

General

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit. For M series, the upper limit can be set on the cutting feedrate for each axis with parameter No. 1430.

Parameter

1422	Maximum cutting feedrate for all axes

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

Specify the maximum cutting feedrate.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

> **NOTE**
> To specify the maximum cutting feedrate for each axis, use parameter No. 1430 instead. (M series)

1430	
	Maximum cutting feedrate for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000
Rotation axis	1 deg/min	6 – 240000	6 – 100000

Specify the maximum cutting feedrate for each axis.

A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

NOTE

1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1422 is effective.

2 If the setting for each axis is 0, the maximum feedrate specified in parameter No. 1422 is applied to all axes and the feedrate is clamped at the maximum feedrate.

Alarm and message

Number	Message	Description
5009	PARAMETER ZERO (DRY RUN)	The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is set to 0 in the HPCC model.
5011	PARAMETER ZERO(CUT MAX)	The maximum cutting feedrate (parameter No. 1422)is set to 0 in the HPCC mode.

NOTE

For HPCC mode, refer to 7.1.14 HIGH PRECISION CONTOUR CONTROL.

Warning

WARNING

CNC calculation may involve a feedrate error of +2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting Feed
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.5.3	Cutting Feed
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting Feed
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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.

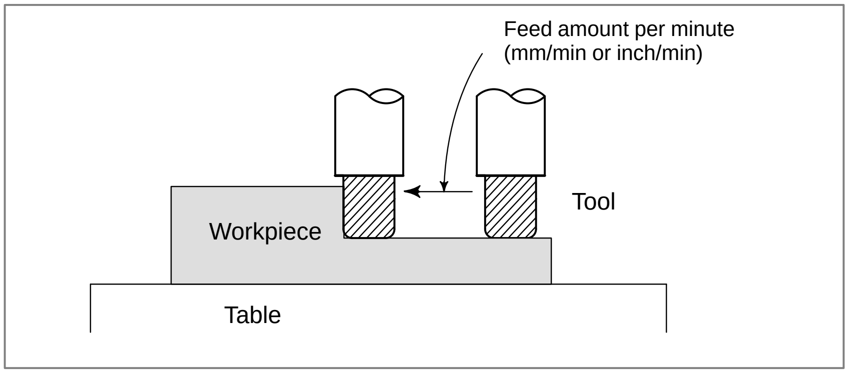


Fig. 7.1.3 Feed per minute

> **CAUTION**
> No override can be used for any commands such as for threading.

Format

```
For M series
    G94;   G code for feed per minute (Group 05)
    F_;    Feed rate (mm/min or inch/min)
```

```
For T series
    G98;   G code for feed per minute (Group 05)
    F_;    Feed rate (mm/min or inch/min)
```

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

NOTE
M series are not equipped with this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

	#7	#6	#5	#4	#3	#2	#1	#0
3401							FCD	

[Data type] Bit

FCD When an F command and a G command (G98, G99) for feed per minute or feed per rotation are specified in the same block, and the G command (G98, G99) is specified after the F command, the F command is:

0 : Assumed to be specified in the mode (G98 or G99) when the F command is specified

1 : Assumed to be specified in the mode of the G command (G98 or G99) of the same block

NOTE
1 When FCD = 1:
If the block containing a G command (G98, G99) does not include an F command, the last F command specified is assumed to be specified in the G command mode of the block.
Example 1: N1 G99 ;
N2 Faaaa G98 ;
- Faaaa is assumed to be specified in the G98 mode.
N3 Fbbbb ;
- Fbbbb is assumed to be specified in the G98 mode.
N4 G99 ;
- Fbbbb is assumed to be specified in the G99 mode.
2 In G code system B or C, G98 and G99 function are specified in G94 and G95.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.5.3	Cutting feed
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.5.3	Cutting feed

7.1.4 Feed Per Revolution/ Manual Feed Per Revolution

General

- **Feed per revolution**

After specifying G95 (G99 for T series) (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 (G99 for T series) is a modal code. Once a G95 is specified, it is valid until G94 (G98 for T series) (feed per minute) is specified.
An override of between 0 and 254% (in steps of 1%) can be applied to feed per rotation, using the feedrate override signals (*FV0 to *FV7). (See Subsection 7.1.7.2.)

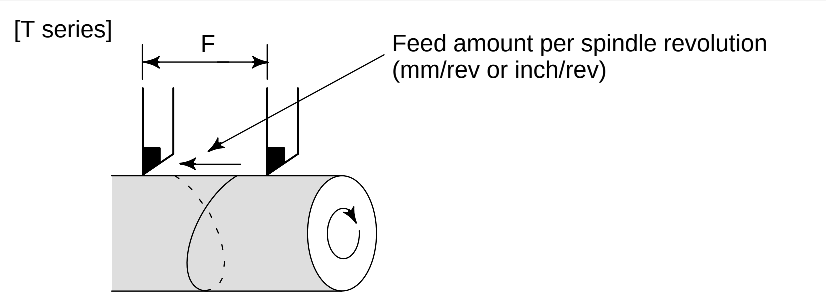


Fig. 7.1.4 Feed per revolution

- **Manual feed per revolution**

Jog feedrate can be specified by feed per revolution.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1402				JRV				NPC
								NPC

[Data type] Bit

JRV Jog feed
0 : Jog feed is performed at feed per minute.
1 : Jog feed is performed at feed per rotation.

> **NOTE**
> Specify a feedrate in parameter No. 1423.

NPC Feed per revolution command
0 : The feed per revolution command is ignored when the position coder is not installed.
1 : The feed per revolution command is accepted even when the position coder is not installed. (The CNC automatically converts the feed per revolution command to a feed–per–minute operation.)

1423	Feedrate in jog feed for each axis

[Data type] Word axis

When JRV, bit 4 of parameter No. 1402, is set to 1 (feed per revolution) in T series, specify a feedrate in jog feed (feed per revolution) with an override of 100% applied to the jog feedrate.

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	0.01 mm/rev	0 to 32767
Inch machine	0.001 inch/rev	
Rotation axis	0.01 deg/rev	

Caution

CAUTION
When the speed of the spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.5.3	Cutting feed
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.5.3	Cutting feed

7.1.5 F1-digit Feed (M series)

General

When a one–digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 1451 to 1459) is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing F1–digit feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{Fmax}{100X}$$

Fmax : feedrate upper limit for F1–F4 set by parameter 1460, or feedrate upper limit for F5–F9 set by parameter 1461
X : any value of 1–127 set by parameter 1450

The feedrate set or altered is kept even while the power is off. The current feedrate is displayed on the CRT screen.

Signal

F1-digit feed select signal F1D <G016#7>

[Classification] Input signal

[Function] Increases or decreases F1-digit speed set by the parameters No. 1451 to 1459 using the manual pulse generator.
Since the manual pulse generator may also be used for axis feeding, signal F1D (G016#7) designates which function may be used.

[Operation] When the signal is “1”, the F1–digit speed can be increased/decreased using the manual pulse generator.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G016	F1D							

Parameter

1450	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 1460 or 1461.

Fmax1: Upper limit of the feedrate for F1 to F4 (parameter 1460)

Fmax2: Upper limit of the feedrate for F5 to F9 (parameter 1461)

1451	Feedrate for F1 digit command F1
1452	Feedrate for F1 digit command F2
1453	Feedrate for F1 digit command F3
1454	Feedrate for F1 digit command F4
1455	Feedrate for F1 digit command F5
1456	Feedrate for F1 digit command F6
1457	Feedrate for F1 digit command F7
1458	Feedrate for F1 digit command F8
1459	Feedrate for F1 digit command F9

Input for setting is enabled.

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	0.1 mm/min	6 – 150000	6 – 120000
Inch machine	0.01 inch/min	6 – 60000	6 – 48000
Rotation axis	0.1 deg/min	6 – 150000	6 – 120000

Set Feedrates for F1-digit feed commands F1 to F9.

When an F1-digit feed command is executed, as the feedrate is changed by turning the manual pulse generator, these parameter values also change accordingly.

1460	Upper limit of feedrate for the F1–digit feed command (F1 to F4)

1461	Upper limit of feedrate for the F1-digit feed command (F5 to F9)

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the upper limit of feedrate for the F1-digit feed command.

As the feedrate increases by turning the manual pulse generator, the feedrate is clamped when it reaches the upper limit set. If an F1-digit feed command F1 to F4 is executed, the upper limit is that set in parameter 1460. If an F1-digit command F5 to F9 is executed, the upper limit is that set in parameter 1461.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting feed
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting feed

7.1.6 Feedrate Inverse Time Specification (M series)

General

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN).

- **Linear interpolation (G01)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Distance}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Distance: mm (metric input)
inch (inch input)

- **Circular interpolation (G02, G03)**

$$FRN = \frac{1}{\text{Time (min)}} = \frac{\text{Speed}}{\text{Circle radius}}$$

Speed: mm/min (metric input)
inch/min (inch input)
Circle radius:
mm (metric input)
inch (inch input)

CAUTION
In circular interpolation, the distance is not an actual distance of the block but the speed is calculated from the circle radius.

Alarm and message

Number	Message	Description
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed, F0 was specified or the feedrate calculated (less than 0.001 mm/min, for metric input or less than 0.00001 inch for inch input) becomes less than an allowable range.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting feed
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting feed

7.1.7 Override

7.1.7.1 Rapid traverse override

General

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).
Also, 1% rapid traverse override select signal allows rapid traverse override every 1% in the range of 0 to 100%.

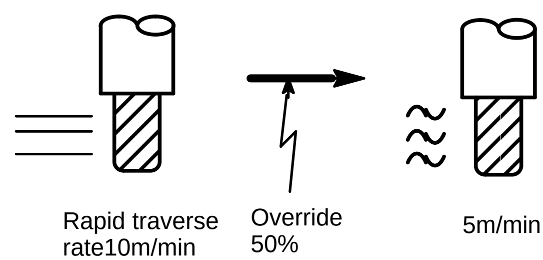


Fig.7.1.7.1 Rapid traverse override

- **Feedrate**

Actual feedrate is obtained by multiplying the rapid traverse rate preset by parameter no.1420 by the override value determined by this signal, whether in automatic or manual operation (including manual reference position return).

- **F0 rate**

For F0 value, an absolute value is set by parameter no.1421 within a range of 0 to rapid traverse rate (for each axis).

- **1% step rapid traverse override selection signal**

1% step rapid traverse override selection signal HROV determines whether rapid traverse override specified with rapid traverse override signals ROV1 and ROV2 is used or 1% step rapid traverse override is used.

When signal HROV is 0, override is applied to the rapid traverse rate using signals ROV1 and ROV2.

When signal HROV is 1, ROV1 and ROV2 are ignored, 1% step rapid traverse override signals *HROV0 to *HROV6 being used to override the rapid traverse rate.

- **PMC axis control**

These 1% step rapid traverse override signals are also effective to the rapid traverse rate for the PMC axis. When rapid traverse override is applied to the PMC axis (using signals ROV1E and ROV2E) with the setting of the OVE bit (bit 2 of parameter No. 8001) independently of the CNC, the 1% step rapid traverse override signals are ineffective.

Signal

Rapid traverse override signal ROV1,ROV2 <G014#0, #1>

[Classification] Input signal

[Function] These signals override the rapid traverse rate

[Operation] These code signals correspond to the rates as follows:

Rapid traverse override		Override value
ROV2	ROV1	
0	0	100 %
0	1	50 %
1	0	25 %
1	1	Fo %

Fo: Set in parameter No. 1421

1% step rapid traverse override selection signal HROV <G096 #7>

[Classification] Input signal

[Function] Selects whether the rapid traverse override signals or the 1% step rapid traverse override signals are enabled.

[Operation] When HROV is 1, signals *HROV0 to *HROV6 are effective and rapid traverse override with signals ROV1 and ROV2 is ignored.

When HROV is 0, signals *HROV0 to *HROV6 are ineffective, and rapid traverse override with signals ROV1 and ROV2 is effective.

1% step rapid traverse override signals *HROV0 to *HROV6 <G096 #0 to #6>

[Classification] Input signal

[Function] Applies override to the rapid traverse rate in the range of 0% to 100% in steps of 1%.

[Operation] These seven signals give a binary code indicating an override applied to the rapid traverse rate.

· When a binary code corresponding to an override value of 101% to 127% is specified, the override applied is clamped at 100%.

· Signals *HROV0 to *HROV6 are inverted signals.
To set an override value of 1%, set signals *HROV0 to *HROV6 to 1111110, which corresponds to a binary code of 0000001.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G014							ROV2	ROV1
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0

Parameter

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	30 – 15000	6 – 12000
Inch machine	0.1 inch/min	30 – 6000	6 – 4800
Rotation axis	1 deg/min	30 – 15000	6 – 12000

Set the F0 rate of the rapid traverse override for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8001						OVE		

[Data type] Bit

OVE Dry run and override signals during axis control by the PMC

0 : Use the same signals as CNC
(1) Feedrate override signal *FV0 to *FV7
(2) Override cancel signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT

1 : Use dedicated axis control signals by the PMC.
(1) Feedrate override signal *FV0E to *FV7E
(2) Override cancel signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.5.3	Rapid traverse override
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.5.3	Rapid traverse override
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.5.3	Rapid traverse override
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.5.3	Rapid traverse override

7.1.7.2 Feedrate override

General

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial.This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

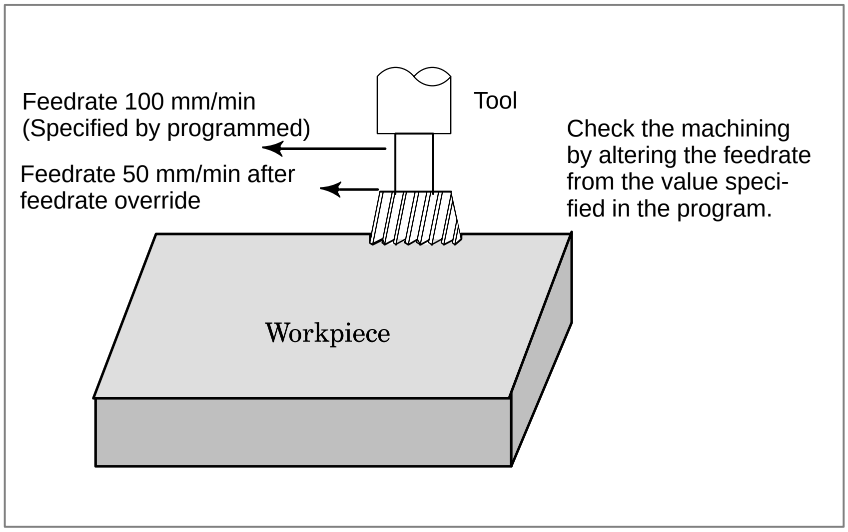


Fig. 7.1.7.2 Feedrate override

Signal

Feedrate Override signal *FV0 to *FV7 <G012>

[Classification] Input signal

[Function] These signals override the cutting feedrate. Eight binary code signals correspond to override values as follows:

$$\text{Override value} = \sum_{i=0}^{7} (2^{i} \times Vi) \ \%$$

Vi=0 when *FVi is “1” and
Vi=1 when *FVi is “0”

These signals have the following weight.

*FV0 : 1%	*FV1 : 2%
*FV2 : 4%	*FV3 : 8%
*FV4 : 16%	*FV5 : 32%
*FV6 : 64%	*FV7 : 128 %

When all signals are “0”, they are regarded as overriding 0% in the same way as when all signals are “1” .
Thus, the override is selectable in steps over a range of 0 to 254%.

[Operation] Actual feedrate is obtained by multiplying the 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
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1401				RFO				

[Data type] Bit

RFO When cutting feedrate override is 0% during rapid traverse,

0 : The machine tool does not stop moving.
1 : The machine tool stops moving.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.5.3	Cutting feed
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.3	Cutting feed
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.5.3	Cutting feed

7.1.7.3 Second feedrate override

General

These signals override the cutting feedrate after the cutting feedrate has been overridden by first override *FV0 to *FV7.

Signal

Second feedrate override signal *AFV0 to *AFV7 <G013>

These eight binary code signals correspond to the override values as follows.

$$\text{Override value} = \sum_{i=0}^{7} (2^i \times Vi) \%$$

Vi=0 when *AFVi is “1” and
Vi=1 when *AFVi is “0”

These signals have the following weight.

*AFV0 : 1%	*AFV1 : 2%
*AFV2 : 4%	*AFV3 : 8%
*AFV4 : 16%	*AFV5 : 32%
*AFV6 : 64%	*AFV7 : 128 %

If all signals are “0” or “1”, the override is regarded as 0%. The override is selectable in steps over a range of 0 to 254%.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G013	*AFV7	*AFV6	*AFV5	*AFV4	*AFV3	*AFV2	*AFV1	*AFV0

7.1.7.4 Override cancel

General

The override cancel signal fixes the feedrate override to 100%.

Signal

Override cancel signal OVC <G006#4>

[Classification] Input signal

[Function] Feedrate override is fixed to 100%.

[Operation] When the signal is “1”, the CNC operates as follows:

- Feedrate override is fixed to 100% irrespective of feedrate override signal.
- Rapid traverse override and spindle speed override are not affected.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G006				OVC				

7.1.8 Automatic Corner Override (M series)

General

- **Inner corner automatic override**

When G62 is specified, and the tool path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.
There are four types of inner corners (Fig. 7.1.8).
2,≦θ≦θp≦178, in Fig. 7.1.8
θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

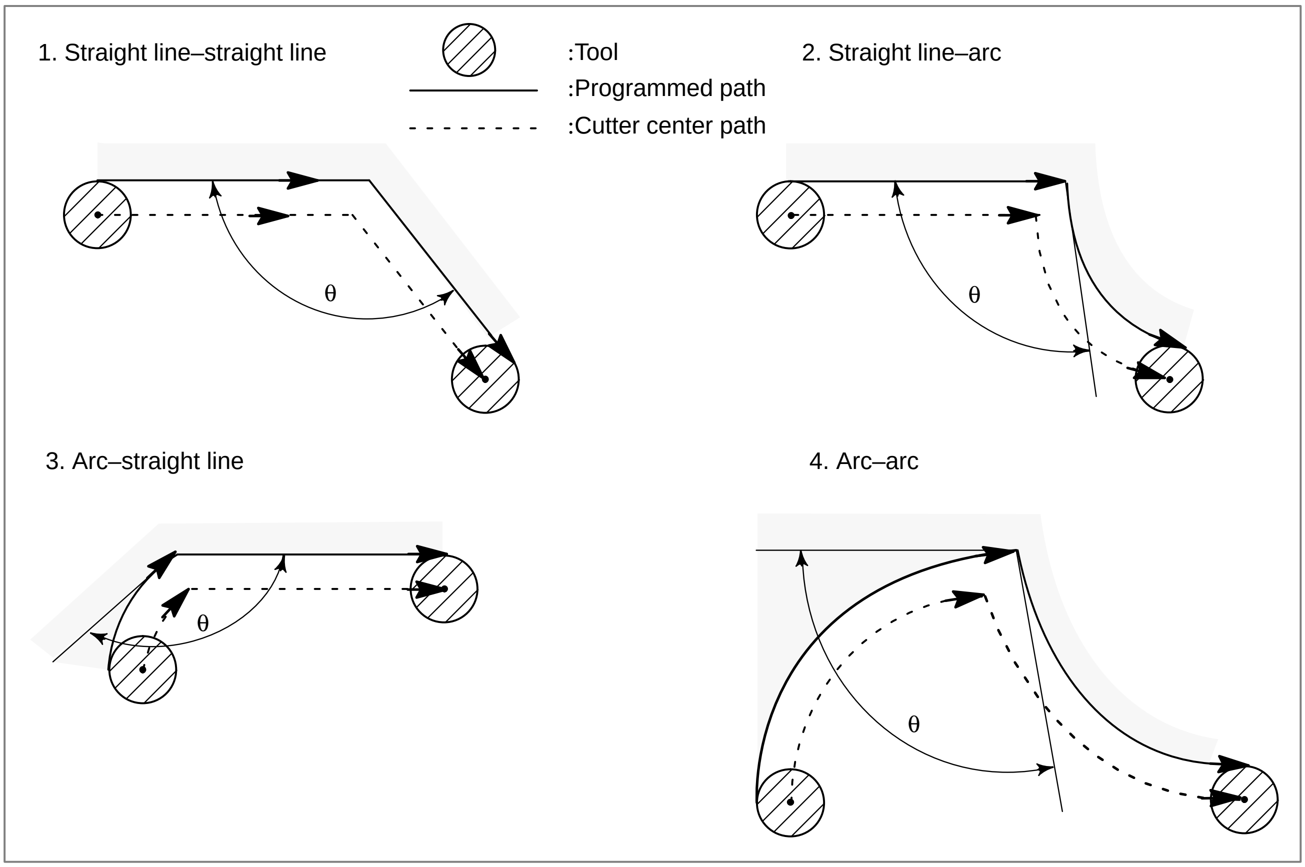


Fig. 7.1.8 Inner corner

> **WARNING**
> When the block before a corner is a start–up block, or the block after a corner includes G41 or G42, the feedrate is not overridden. The feedrate override function is disabled when the offset value is 0.

- **Override value**

An override value is set with parameter No. 1712. An override value is valid even for dry run and F1–digit feed specification.
In the feed per minute mode, the actual feedrate is as follows:

F × (inner corner automatic override) × (feedrate override)

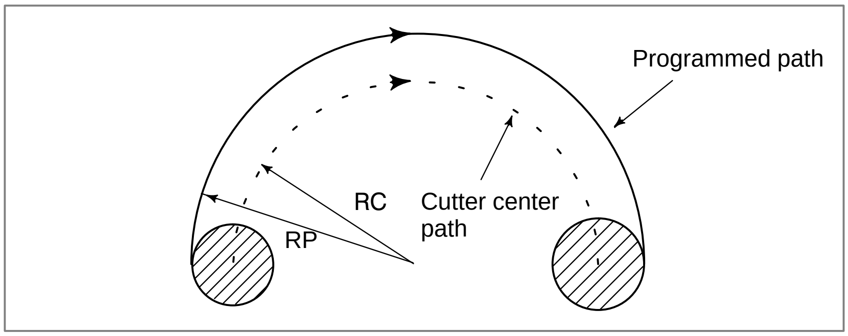
- **Internal circular cutting feedrate change**

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below. This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F \times \frac{Rc}{Rp}$$

Rc : Cutter center path radius
Rp : Programmed radius

It is also valid for the dry run and the F1–digit feed command.



Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp ≐ 0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp ≦ MDR, the feedrate of the tool is (F×MDR).

> **CAUTION**
> When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times \text{(inner corner override)} \times \text{(feedrate override)}$$

Parameter

1710	Minimum deceleration ratio (MDR) of the inner circular cutting rate in automatic corner override

[Data type] Byte

[Unit of data] %

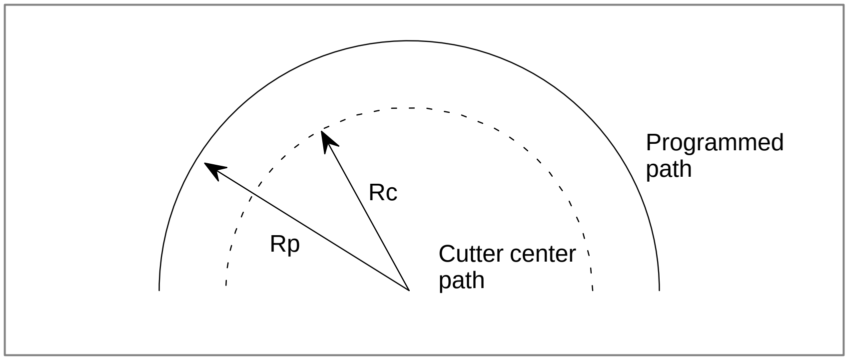
[Valid data range] 1 to 100

Set the minimum deceleration ratio (MDR) in changing the inner circular cutting feedrate by automatic corner override.

In circular cutting with an inward offset, the actual feedrate for a specified feedrate (F) becomes as follows:

$$F \times \frac{Rc}{Rp} \qquad \left(\begin{array}{l} Rc\text{: Radius of the path of the cutter's center} \\ Rp\text{: Programmed radius} \end{array} \right)$$

As the actual feedrate becomes the value obtained from the above equation, the specified rate F can be achieved on the program path.



If Rc is too samall in comparison with Rp so that $\frac{Rc}{Rp} \doteq 0$, the cutter will stop. To prevent this, the minimum deceleration ratio (MDR) is set.

When $\frac{Rc}{Rp} \doteq 0$,

the actural rate becomes as follows:

$F \times (MDR)$

1711	Angle (θp) to recognize the inner corner in automatic override

[Data type] Byte

[Unit of data] Degree

[Valid data range] 1 to 179 (standard value = 91)

Set the angle to recognize the inner corner when automatic corner override is performed for the inner corner

1712	Amount of automatic override for an inner corner

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100 (standard value = 50)

Set inner corner automatic override value when automatic corner override is performed.

1713	Distance Le from the starting point in inner corner automatic override

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

Set distance Le from the starting point in an inner comer for automatic corner override.

1714	Distance Ls up to the ending point in inner corner automatic override

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Input in mm	1	0.1	0.01	mm
Input in inches	0.1	0.01	0.001	inch

[Valid data range] 0 to 3999

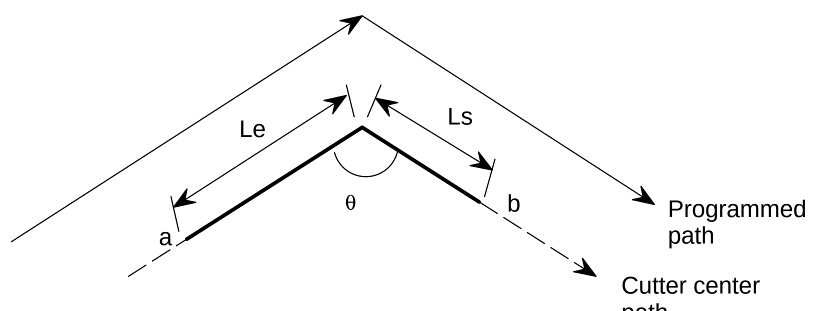
Set distance Ls up to the end point in an inner corner for automatic corner override.

If $\theta \leqq \theta$p, the inside of a comer is recognized. (θ is set in parameter 1711.)

When an inner corner is recognized, the feedrate is overridden in the range of Le in the block immediately before the intersection of the corner and Ls in the next block following the intersection.

Ls and Le are each a straight line connecting the intersection of the corner and a given point on the path of the cutter's center.

Ls and Le are set in parameters 1713 and 1714.



An override is applied from point a to b.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.5.4.2	Automatic Override for Inner Corners
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.5.4.2	Automatic Override for Inner Corners

7.1.9 External Deceleration

General

These signals decelerate the feedrate of the control axes down to the speed which has been set by parameter No. 1426 and 1427.

Signal

External deceleration signal *+ED1 to *+ED8<G118> *–ED1 to *–ED8<G120>

[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.

* + ED ×

1 … The 1st axis is decelerated.

2 … The 2nd axis is decelerated.

3 … The 3rd 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.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
G120	*–ED8	*–ED7	*–ED6	*–ED5	*–ED4	*–ED3	*–ED2	*–ED1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1005			EDMx	EDPx				

[Data type] Bit axis

EDPx External deceleration signal in the positive direction for each axis

0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

EDMx External deceleration signal in the negative direction for each axis

0 : Valid only for rapid traverse
1 : Valid for rapid traverse and cutting feed

1426	External deceleration rate of cutting feed

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the external deceleration rate of cutting feed.

1427	External deceleration rate of rapid traverse for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

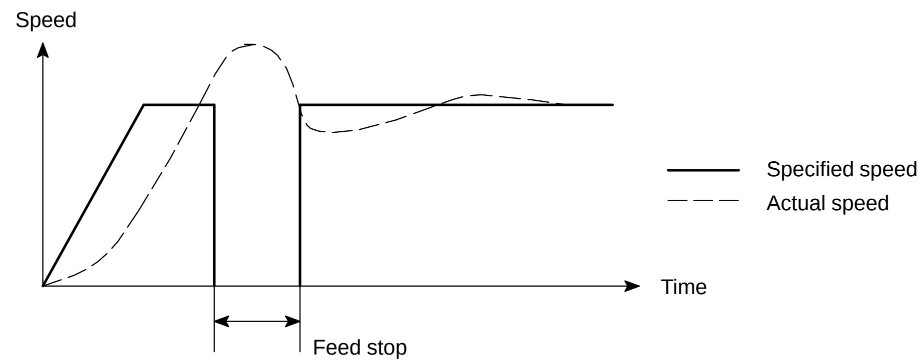
Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the external deceleration rate of rapid traverse for each axis.

7.1.10 Feed Stop Function

General

During axis motion, the feed stop function checks a position deviation amount at all times. When the amount exceeds the “feed stop position deviation amount” set by the parameter (No. 1832), the function suspends pulse distribution and acceleration/deceleration control during such a period of time, and terminates the move command for the positioning control circuit. Thus the function can minimize an overshoot that may occur with a large servo motor in rapid traverse acceleration operation.



Parameter

1832	Feed stop positioning deviation for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the feed stop positioning deviation for each axis.

If the positioning deviation exceeds the feed stop positioning deviation during movement, pulse distribution and acceleration/deceleration control are stopped temporarily. When the positioning deviation drops to the feed stop positioning deviation or below, pulse distribution and acceleration/deceleration control are resumed.

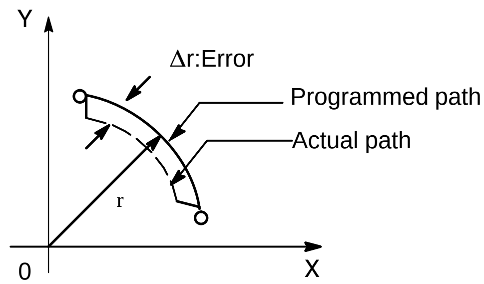
The feed stop function is used to reduce overshoot in acceleration/deceleration mainly by large servo motor.

Generally, set the middle value between the positioning deviation limit during movement and the positioning deviation at rapid traverse as the feed stop positioning deviation.

7.1.11 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:



$$\Delta r = \frac{1}{2}(T_1^2+T_2^2)\frac{v^2}{r} \quad \ldots\ldots \text{(Expression 1)}$$

Δr: Maximum radial error (mm)
v: Feedrate (mm/s)
r: Arc radius (mm)
T_1: Time constant for exponential acceleration/deceleration of cutting feed (s)
T_2: Time constant of the servo motor (s)

When actual machining is performed, radius r of the arc to be machined and permissible error Δr are given. Then, maximum allowable feedrate v (mm/min) is determined from the above expression.

The function for clamping the feedrate by the arc radius automatically clamps the feedrate of arc cutting to the value set in a parameter. This function is effective when the specified feedrate may cause the radial error for an arc with a programmed radius to exceed the permissible degree of error.

When the permissible error Δr is determined, the maximum permissible speed V for the arc radius R is obtained from expression 2.

$$\Delta r = \frac{1}{2}(T_1^2+T_2^2)\frac{V^2}{R} \quad \ldots\ldots \text{(Expression 2)}$$

For the arc radius r, the maximum permissible speed v to set the permissible error to Δr is obtained from expression 1. From expressions 1 and 2, the following expression is obtained:

$$\frac{1}{2}(T_1^2+T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2+T_2^2)\frac{V^2}{R}$$

$$\therefore \quad v = \sqrt{\frac{r}{R}}\,V \quad \ldots\ldots \text{(Expression 3)}$$

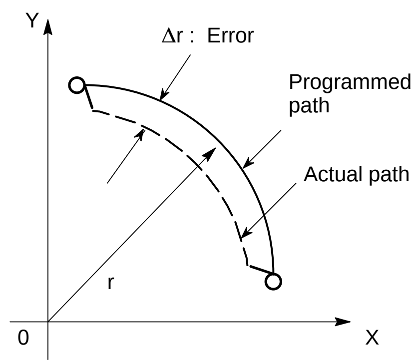
When a given arc radius R and the maximum permissible speed V for that arc radius are set as parameters, the maximum permissible speed v for an arc with a programmed radius r can be obtained from expression 3. Then, if a specified feedrate exceeds the speed v, the feedrate is automatically clamped to the speed v.

The maximum permissible speed v obtained from expression 3 decreases with the specified arc radius. To prevent the maximum permissible speed from decreasing excessively, the lower limit imposed on the maximum permissible speed v can be set in parameter 1732.

Provided the specified feedrate does not exceed the maximum permissible speed v obtained from expression 3, arc cutting is performed at the specified feedrate.

- **Cutting feed linear acceleration/deceleration**

When the cutting feed linear acceleration/deceleration function is used, an approximate error in arc cutting can be obtained from expression 4.



Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Arc radius (mm)
T_1 : Time constant for cutting feed linear acceleration/deceleration (s)
T_2 : Servo motor time constant (s)

$$\Delta r = \left(\frac{1}{24}T_1{}^2 + \frac{1}{2}T_2{}^2\right)\frac{V^2}{r} \quad \cdots\cdots\cdots \text{(Expression 4)}$$

As can be seen from expression 4, expression 3 also holds for linear acceleration/deceleration after interpolation. Therefore, feedrate clamping by the arc radius is enabled.

- **Actual error**

Expressions 1, 2, and 4 logically denote an approximate error in the CNC. They do not denote an actual error in machining.

Let the actual error in machining be Δr all. Then, it is expressed as follows:

$$\Delta r\ \mathit{all} = \underbrace{\Delta r\ \mathit{NC}}_{\text{Error caused by the acceleration/deceleration time constant } (T_1) \text{ and servo motor time constant } (T_2)} + \underbrace{\Delta r\ \mathit{machine}}_{\text{Error caused by the machine}} \quad \cdots\cdots\cdots \text{(Expression 5)}$$

This function controls only the first term on the right side of expression 5. In other words, this function suppresses only the error caused by the acceleration/deceleration time constant (T_1) and servo motor time constant (T_2) to a certain level. This function does not control the error caused by the machine.

Expressions 1, 2, and 4 are approximate expressions. This means that, as the arc radius becomes smaller, the approximate precision lowers. Therefore, even when the feedrate is clamped to the maximum permissible speed v obtained from expression 3, the permissible error may be exceeded.

Parameter

1730	Maximum feedrate for arc radius R

[Data type] Word

[Unit of data]

[Valid data range]

.Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 – 15000	0 – 12000
Inch machine	0.1 inch/min	8 – 6000	0 – 4800

Set a maximum feedrate for the arc radius set in parameter No. 1731.

1731	Arc radius value corresponding to a maximum feedrate

[Data type] Two–word

[Unit of data]

Unit	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter machine)	0.01	0.001	0.0001	mm
Linear axis (inch machine)	0.001	0.0001	0.00001	inch

[Valid data range] 1000 to 99999999

Set the arc radius corresponding to the maximum feedrate set in parameter No. 1730.

1732	Minimum value (RV min) for arc radius–based feedrate clamp

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 15000	0 – 12000
Inch machine	0.1 inch/min	0 – 6000	0 – 4800

The arc radius–based feedrate clamping function reduces the maximum feedrate as the arc radius decreases. When the specified maximum feedrate is not greater than RV min (minimum value for arc radius–based feedrate clamping), RV min is used as the maximum feedrate.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.19.2	Feedrate clamp by circle radius
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.19.1	Feedrate clamp by circle radius

7.1.12
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 CSD bit (bit 4 of parameter No. 1602).

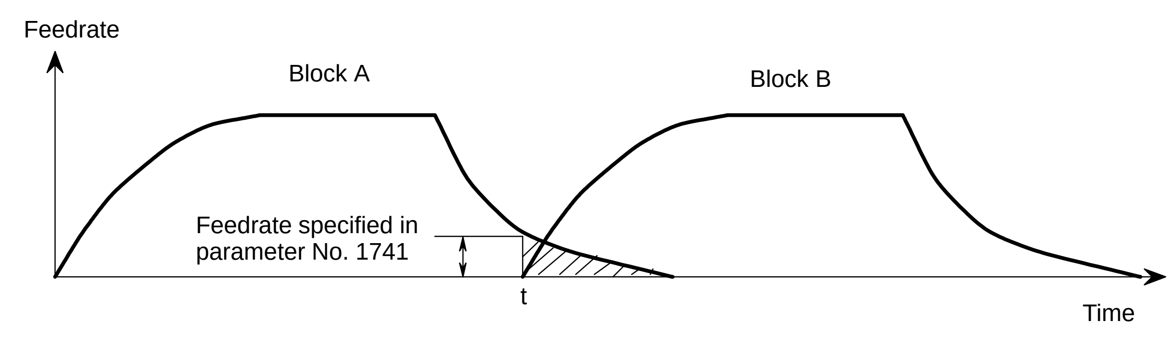
Feedrate control according to corner angle

- **Overview**

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates along the first and second axes on that plane are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

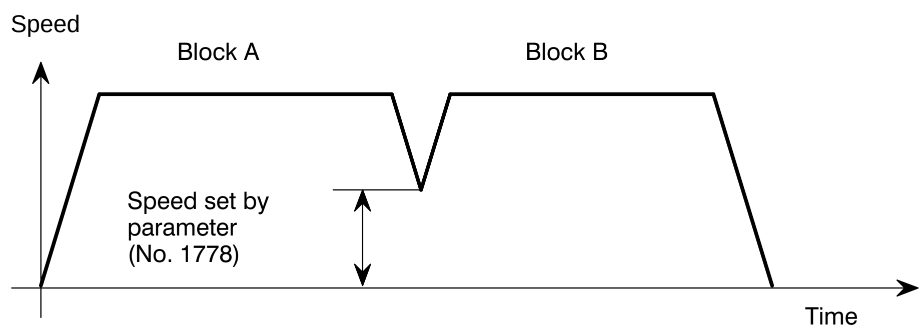
The figure shows the relationship between feedrate and time when a corner angle is smaller than the angle specified in the parameter.

At time t, some accumulated pulses remain, as indicated by the shaded part. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in the parameter.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

If the angle made by blocks A and B is smaller than that specified in parameter No. 1740 (for the selected plane), and if the feedrates programmed for blocks A and B are higher than the value set in parameter No. 1778, the feedrate is reduced to the value specified in the parameter in block A. In block B, the feedrate is increased to the programmed feedrate. The rate of acceleration depends on the parameter for linear acceleration/deceleration before interpolation for cutting feed.



● Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner deceleration function)

0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1740	Critical angle subtended by two blocks for automatic corner deceleration

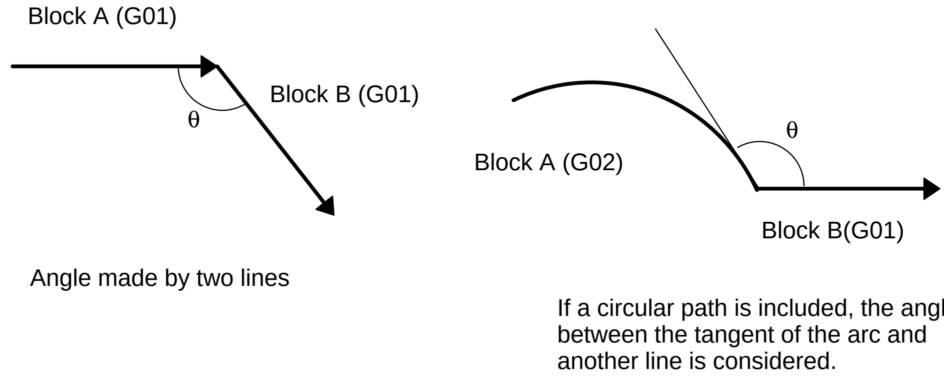
[Data type] Two–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 180000

Set a critical angle to be subtended by two blocks for corner deceleration when the angle–based automatic corner deceleration function is used.

The angle subtended by two blocks is defined as θ in the examples shown below.



1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

1778	Minimum speed for the automatic corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 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.

Caution

CAUTION

1 The angle of the machining tool path is compared with that specified in parameter No. 1740 only for the selected plane. The actual feedrate and that specified in parameter No. 1741 are compared only for the first and second axes of the selected plane. Even if simultaneous movement is performed along three or more axes, the feedrates of only the first and second axes are compared for the selected plane.

2 The roundness of a corner is determined by the angle and feedrate specified in parameter Nos. 1740 and 1741, respectively. If a sharp corner is always required, set a feedrate of zero and an angle of 180000 (180 degrees).

3 If a G09 (exact stop) command is executed, an exact stop is performed, irrespective of the angle and feedrate specified in parameter Nos.1740 and 1741.

4 This function is disabled in single block and dry run mode.

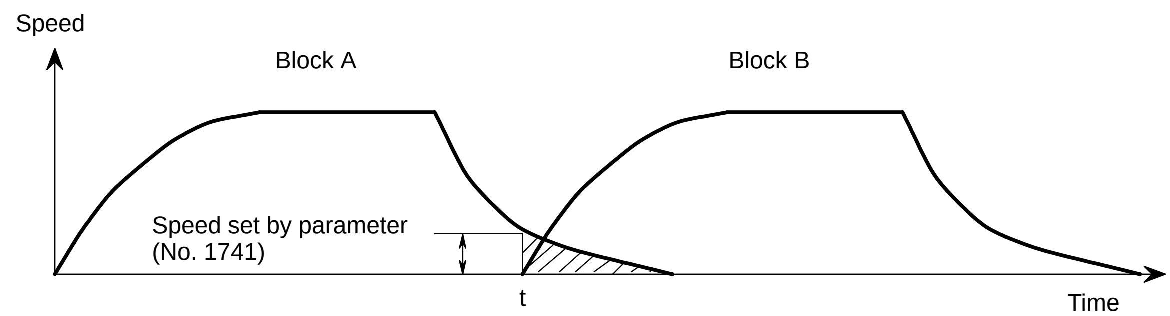
Feedrate control according to the feedrate difference for each axis

• Overview

If the difference between the programmed feedrates at the end of block A and at the beginning of block B for each axis exceeds the value specified in parameter No. 1781, and if the feedrates for all axes are lower than that specified in parameter No. 1741, the system executes block B, assuming that no pulses are accumulated.

The figure shows the relationship between the feedrate and time when the feedrate difference for each axis exceeds the value specified in parameter No. 1781.

At time t, some accumulated pulses remain, as indicated by the shaded section. The system, however, starts the next block because the feedrate of the automatic acceleration/deceleration circuit is lower than that specified in parameter No. 1741.



· **When linear acceleration/deceleration before interpolation for cutting feed is enabled**

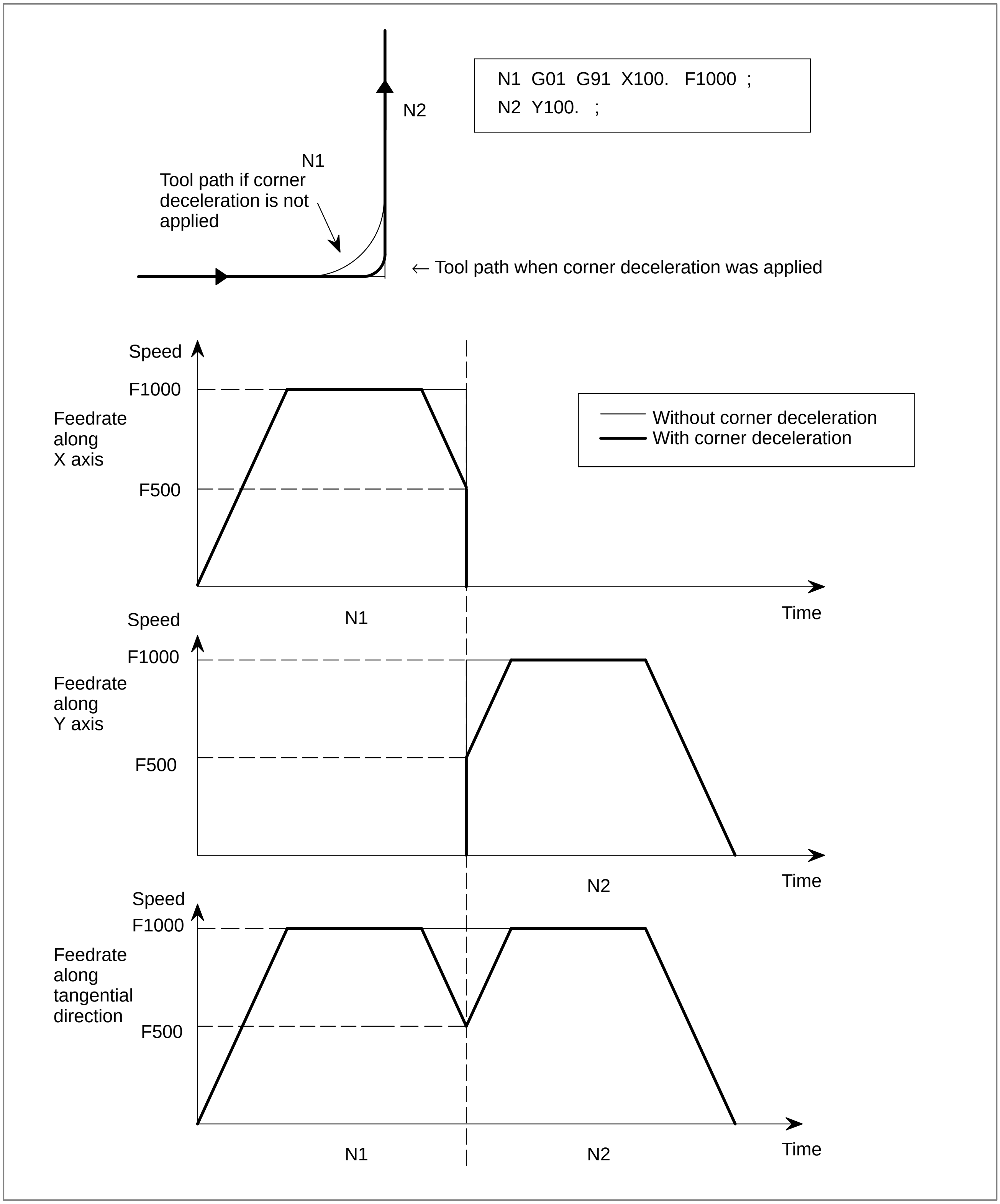
If the difference between the feedrates of blocks A and B for each axis exceeds the value specified in parameter No. 1780, the feedrate at the corner is calculated from the difference for each axis, as shown below. The feedrate is reduced to the calculated value in block A.

The feedrate change for each axis (Vc[X], Vc[Y], ...), caused by the movement at programmed feedrate F, is compared with Vmax specified in parameter No. 1780. If an feedrate change exceeding Vmax is detected, the target feedrate after deceleration Fc is calculated, using maximum comparison value Rmax.

$$R = \frac{Vc}{Vmax}$$

$$Fc = \frac{F}{Rmax}$$

If, for example, the direction of movement is changed from the X–axis to the Y–axis, that is through 90 degrees, and if the programmed feedrate is 1000 mm/min and the permissible feedrate difference specified in parameter No. 1780 is 500 mm/min, the deceleration shown below is performed:



Different permissible feedrate differences can be specified for different axes. If a value is specified in parameter No. 1783, the permissible feedrate difference for each axis becomes valid. Deceleration at a corner is calculated for the axis for which the permissible feedrate difference is exceeded with the highest ratio of actual feedrate difference to permissible feedrate difference.

• Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601		ACD						

[Data type] Bit

ACD Function for automatically reducing the feedrate at corners (automatic corner deceleration function)

0 : The function is not used.
1 : The function is used.

	#7	#6	#5	#4	#3	#2	#1	#0
1602				CSD				

[Data type] Bit

CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

1741	Feedrate for assuming the termination of automatic corner deceleration (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set the feedrate for assuming the termination of deceleration in automatic corner deceleration.

1780	Allowable speed difference for the speed difference–based automatic corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the speed difference for the speed difference–based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

1781	Allowable speed difference for the speed difference–based automatic corner deceleration function (for acceleration/deceleration after interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set a speed difference for the speed difference–based automatic corner deceleration function when acceleration/deceleration after interpolation is used.

1783	Allowable feedrate difference in automatic corner deceleration based on the feedrate difference (for linear acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

This parameter sets the feedrate difference for each axis in automatic corner deceleration based on the feedrate difference when acceleration/deceleration before interpolation is used. When this parameter is set, the value set in parameter 1780 becomes invalid.

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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.5.4.3	Automatic corner deceleration

7.1.13 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

For details on the above functions, see the descriptions of the functions.

Signal

advanced preview control mode signal G08MD <F066#0>

[Classification] Output signal

[Function] Informs that is in the advanced preview control mode.

[Output condition] The signal is "1" in the following case:
·In the advanced preview control mode
The signal is "0" in the following case:
·It is not the advanced preview control mode

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F066								G08MD

Parameter

1431	Maximum cutting feedrate for all axes in the advanced preview control mode

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 240000	0 – 100000
Inch machine	0.1 inch/min	0 – 96000	0 – 48000
Rotation axis	1 deg/min	0 – 240000	0 – 100000

Specify the maximum cutting feedrate for all axes in the advanced preview control mode.

A feedrate in the tangential direction is clamped in cutting feed so that it does not exceed the feedrate specified in this parameter.

WARNING
In a mode other than the advanced preview mode, the maximum cutting feedrate specified in parameter No. 1422 or No. 1430 is applied and the feedrate is clamped at the maximum feedrate.

NOTE
To specify the maximum cutting feedrate for each axis, use parameter No. 1432 instead.

1432	Maximum cutting feedrate for each axis in the advanced preview control mode

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 – 240000	0 – 100000
Inch machine	0.1 inch/min	0 – 96000	0 – 48000
Rotation axis	1 deg/min	0 – 240000	0 – 100000

Specify the maximum cutting feedrate for each axis in the advanced preview control mode.

A feedrate for each axis is clamped during cutting feed so that it does not exceed the maximum cutting feedrate specified for each axis.

WARNING
In a mode other than the advanced preview mode, the maximum cutting feedrate specified in parameter No. 1422 or No. 1430 is applied and the feedrate is clamped at the maximum feedrate.

NOTE
1 This parameter is effective only in linear and circular interpolation. In polar coordinate, cylindrical, and involute interpolation, the maximum feedrate for all axes specified in parameter No. 1431 is effective.
2 If a setting for each axis is 0, the maximum feedrate specified in parameter No. 1431 is applied to all axes and the feedrate is clamped at the maximum feedrate.

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2		CSD				FWB

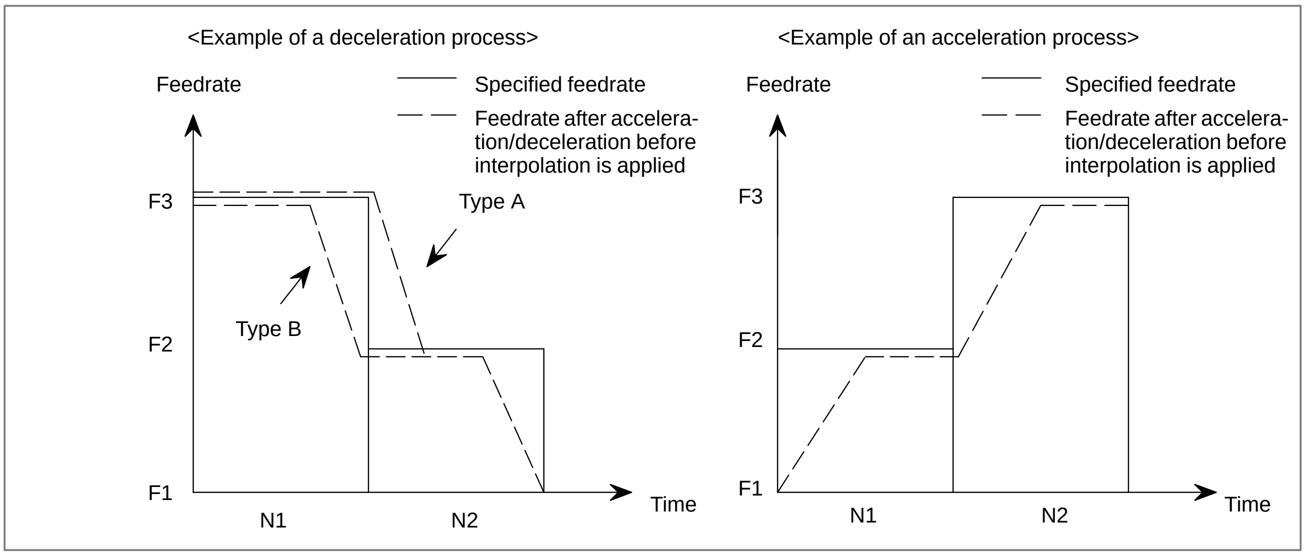
[Data type] Bit

FWB Cutting feed acceleration/deceleration before interpolation

0 : Type A of acceleration/deceleration before interpolation is used.
1 : Type B of acceleration/deceleration before interpolation is used.

Type A: When a feedrate is to be changed by a command, acceleration/deceleration starts after the program enters the block in which the command is specified.

Type B: When a feedrate is to be changed by a command, deceleration starts and terminates at the block before the block in which the command is specified.



CSD In the function for automatically reducing a feedrate at corners,

0 : Angles are used for controlling the feedrate.
1 : Differences in feedrates are used for controlling the feedrate.

LS2 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.)

1762	Exponential acceleration/deceleration time constant for cutting feed in the advanced preview control mode

[Data type] Word axis

[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.

1763	Minimum speed in exponential acceleration/deceleration for cutting feed in the advanced preview control mode

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set a minimum speed (FL) in exponential acceleration/deceleration for cutting feed in the advanced preview control mode.

1768	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 512

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.

1770	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)

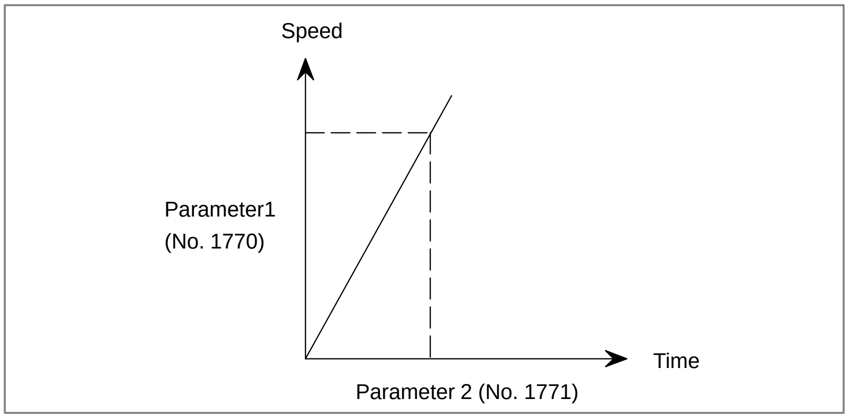
[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 48000

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. 1771.



CAUTION
When 0 is set in parameter No. 1770 or parameter No. 1771, linear acceleration/deceleration before interpolation is disabled.

1771	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. 1770.

CAUTION
1 When 0 is set in parameter No. 1770 or parameter No. 1771, linear acceleration/deceleration before interpolation is disabled.
2 In parameter Nos. 1770 and 1771, set values that satisfy the following: Parameter No. 1770/Parameter No. 1771 ≥ 5

1777	Minimum speed for the automatic corner deceleration function (for the advanced preview control)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

Set a speed at which the number of buffered pulses in deceleration is assumed to be 0 when linear acceleration/deceleration before interpolation is used.

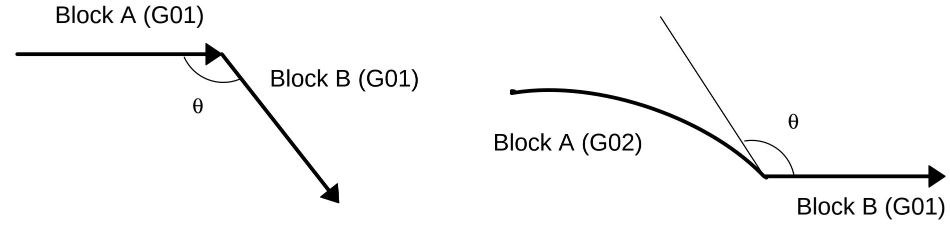
1779	Critical angle subtended by two blocks for automatic corner deceleration (for the advanced preview control)

[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

1780	Allowable speed difference for the speed difference based corner deceleration function (for linear acceleration/deceleration before interpolation)

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Set the speed difference for the speed difference based automatic corner deceleration function when linear acceleration/deceleration before interpolation is used.

1783	Allowable speed difference for the speed difference based corner deceleration function (for linear acceleration /deceleration before interpolation)

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotation axis	1 deg/min	6 – 15000	6 – 12000

A separate allowable feedrate difference can be set for each axis. Among the axes that exceed the specified allowable feedrate difference, the axis with the greatest ratio of the actual feedrate difference to the allowable feedrate difference is used as the reference to calculate the reduced feedrate at the corner.

1784	Speed when overtravel alarm has generated during acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Deceleration is started beforehand to reach the feedrate set in the parameter when an overtravel alarm is issued (when a limit is reached) during linear acceleration/deceleration before interpolation. By using this parameter, the overrun distance that occurs when an overtravel alarm is output can be reduced.

WARNING
The control described above is applicable only to stored stroke check1.

NOTE
1 When 0 is set in this parameter, the control described above is not exercised.
2 Use type–B linear acceleration/deceleration before interpolation (by setting bit 0 (FWB) of parameter No. 1602 to 1).

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)	FWB (1602#0)	←
Acceleration (Parameter 1)	1630	1770
Acceleration (Parameter 2)	1631	1771
Speed when overtravel alarm has generated	1784	←

· Parameters for automatic corner deceleration

Parameter description	Parameter No.	
	Normal mode	Advanced preview control mode
Automatic corner deceleration according to the corner angle or the speed difference	CSD (1602#4)	←
Minimum speed (according to the corner angle)	1778	1777
Critical angle (according to the corner angle)	1740	1779
Allowable speed difference for all axes (according to speed difference)	1780	←
Allowable speed difference for each axis (according to speed difference)	1783	←

Alarm and message

Number	Message	Description
109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.

Note

NOTE

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 contouring control function
·Feed per revolution
·One–digit F code feed
·Polar coordinate interpolation function
·Cylindrical interpolation function
·Involute interpolation function
·Exponential interpolation
·Three–dimensional coordinate conversion
·Retrace function
·Normal direction control
·Polar coordinate command
·Index table indexing
·Tool retract and recover
·Threading and synchronous cutting
·High–speed cycle cutting
·Handle interruption
·Program restart
·Simple synchronous control
·Feed stop
·High–speed skip function
·Constant surface speed control
·Interrupt type custom macro
·Small–hole peck drilling cycle
·High–speed remote buffer A/B
·Automatic tool length measurement
·Skip cutting
·G28 (low–speed reference position return)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.19.3	advanced preview CONTROL (G08)
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.19.2	advanced preview CONTROL (G08)

7.1.14 High–precision Contour Control by RISC (M series)

General

Some machining errors are due to the CNC. Such errors include machining errors caused by acceleration/deceleration after interpolation. To eliminate these errors, the following functions are performed at high speed by an RISC processor. These functions are called high–precision contour control functions.

- Function for multiple–block look–ahead acceleration/deceleration before interpolation. This function eliminates machining errors due to acceleration/deceleration.
- Automatic feedrate control function which enables smooth acceleration/ deceleration by considering changes in the figure and speed and allowable acceleration for the machine. This is performed by reading multiple blocks in advance.

Furthermore, smoother acceleration/deceleration is achieved, enabling the feed–forward factor to be increased. This feature also reduces follow–up error in the servo system.

- **Specification table**

Name	Function
No. of controlled axes	1 axis to 8 axes
No. of simultaneously controlled axes	Up to max. controlled axes
Axis names	Any of A, B, C, U, V, W, X, Y, Z
Increment system	0.01, 0.001, 0.0001 mm 0.001, 0.0001, 0.00001 inch
Max. programming dimensions	±8 digits
Positioning	Yes (Available with parameter MSU (No. 8403#1)=1)
Linear interpolation	Yes
Multi–quadrant circular interpolation	Yes
Helical interpolation	Provided (when bit 2 (G02) of parameter No. 8485 is set to 1)
Involute interpolation	Provided (when bit 4 (INV) of parameter No. 8485 is set to 1)
Feed per minute	Yes
Feedrate clamp	Yes
Feedrate override	0 – 254%, Every 1%
2nd feedrate override	0 – 254%, Every 1%
Workpiece coordinate system	Yes (Unchangeable in G05P10000 mode)

Name	Function
Absolute/incremental command	Combined use possible in the block
Sequence number	5 digits
Tape code	EIA, ISO
Tape format	Word address format
Control in/out	Yes
Optional block skip	Yes
Circle radius R specification	Yes
Automatic operation	Memory operation, Tape operation
Method of tape operation	RS–232–C, RS–422, DNC1, and remote buffer
Manual absolute on/off	Yes (FS15 type)
Cycle start, Feed hold	Yes
Dry run	Yes
Feedrate override under dry run	0 – 655.34%, Every 0.01%
Single block	Yes
Inch/metric conversion	Yes (Unchangeable in G05P10000 mode)
Multi–buffer	Yes
Cutter compensation C	Yes
Interlock (all axes)	Yes
Machine lock	Yes
Subprogram call (M98, M198)	Yes (Usable with parameter MSU (No. 8403#1)=1)
Auxiliary function	Yes (Usable with parameter MSU (No. 8403#1)=1)
Scaling	Provided (when bit 0 (G51) of parameter No. 8485 is set to 1)
Coordinate rotation	Provided (when bit 0 (G51) of parameter No. 8485 is set to 1)
Canned drilling cycle	Provided (when bit 1 (G81) of parameter No. 8485 is set to 1)
Rigid tapping	Provided (when bit 1 (G81) of parameter No. 8485 is set to 1)

- **Data that can be specified**

G00 : Positioning (Note)
G01 : Linear interpolation
G02 : Circular interpolation, Helical interpolation (CW) (Note)
G03 : Circular interpolation, Helical interpolation (CCW) (Note)
G02.2 : Involute interpolation (CW) (Note)
G03.2 : Involute interpolation (CCW) (Note)
G17 : Plane selection (XpYp plane)
where, Xp is the X–axis or its parallel axis;

G18	:	Plane selection (ZpXp plane) where, Yp is the Y–axis or its parallel axis;
G19	:	Plane selection (YpZp plane) where, Zp is the Z–axis or its parallel axis.
G38	:	Cutter compensation C with vector held
G39	:	Cutter compensation C corner arc
G40	:	Cutter compensation C cancel
G41	:	Cutter compensation C, left
G42	:	Cutter compensation C, right
G50	:	Scaling cancel (Note)
G51	:	Scaling command (Note)
G68	:	Coordinate rotation command (Note)
G69	:	Coordinate rotation cancel (Note)
G73, G74, G76, G80 to G89:		Canned drilling cycle, rigid tapping (Note)
G90	:	Absolute command
G91	:	Incremental command
Dxxx	:	D code
Fxxxxx	:	F code
Nxxxxx	:	Sequence number
G05P10000	:	Setting the HPCC mode
G05P0	:	Canceling the HPCC mode
I, J, K, R	:	I, J, K, and R specified for circular interpolation
Axial movement data:		Axial movement data specified with an axis name set in parameter No. 1020 (any of X, Y, Z, U, V, W, A, B, and C)
( )	:	Control in / out
/n	:	Optional block skip
Mxxxx	:	Miscellaneous function (Note)
Sxxxx	:	Spindle speed function (Note)
Txxxx	:	Tool function (Note)
Bxxxx	:	Second miscellaneous function (Note)

NOTE

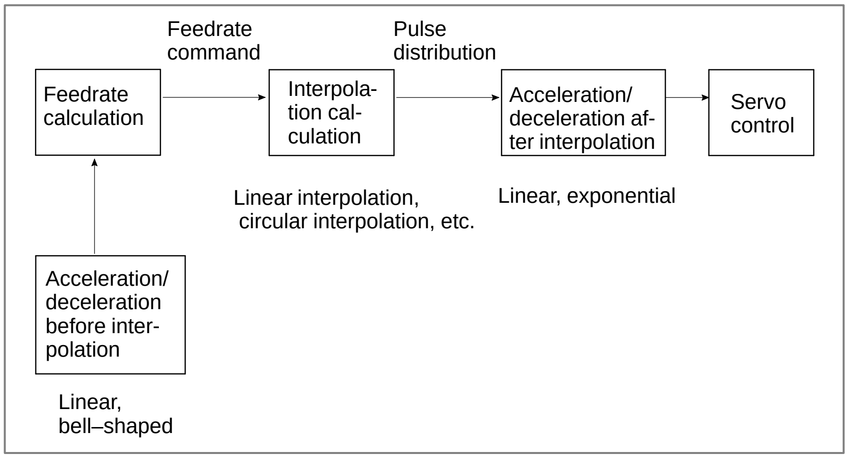
To specify positioning, helical interpolation, involute interpolation, scaling, coordinate rotation, a canned drilling cycle, rigid tapping, a miscellaneous function, a spindle function, a tool function, or a second auxiliary function in high–precision contour control (HPCC) mode, set the corresponding parameter, described in the specification list, to 1. Specifying any of the above functions without setting the corresponding parameter to 1 causes a P/S alarm (No. 5000).

7.1.14.1 Look–ahead acceleration/deceleration before interpolation

When feed per minute is specified, this function reads several tens of blocks ahead to perform acceleration/deceleration before interpolation, that is, to apply acceleration/deceleration to the specified feedrate.

When acceleration/deceleration after interpolation is used, acceleration/deceleration is applied to the interpolated data. Consequently, the interpolated data is changed by acceleration/deceleration. When acceleration/deceleration before interpolation is used, however, acceleration/deceleration is applied to the feedrate data before interpolation. Consequently, the interpolated data is not changed by acceleration/deceleration.

Accordingly, interpolation data ensures that machining follows a specified line or curve at all times, thus eliminating the machining profile errors that result from delays in acceleration/deceleration.

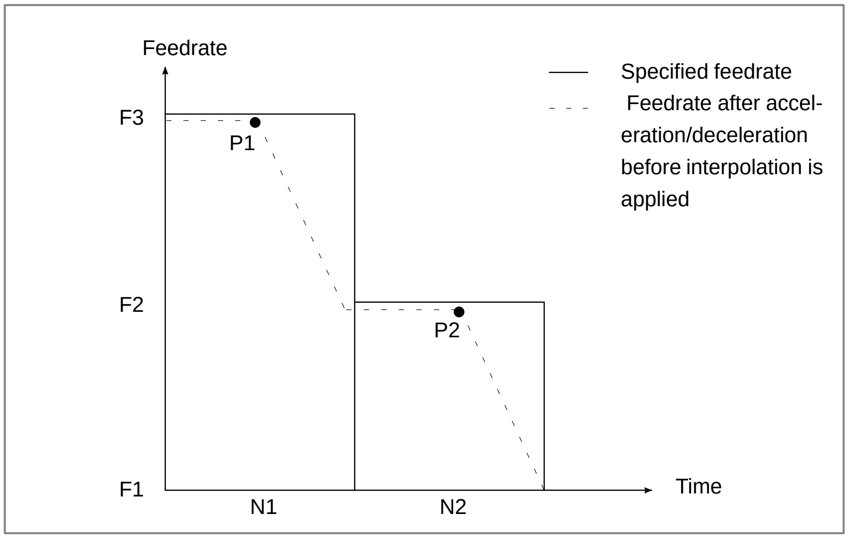


If a feedrate change along any axis is greater than the value set in a parameter for the joint (corner) between two successive blocks, a feedrate is calculated so that the difference in the feedrates does not exceed the specified value. The feedrate is automatically reduced to this calculated value at the corner.

Before this function can be used, specify parameter Nos. 8400 and 8401 for determining an acceleration for acceleration/deceleration before interpolation.

Example of deceleration

To ensure that the feedrate specified for a block is reached when the block is executed, deceleration is started in the previous block.

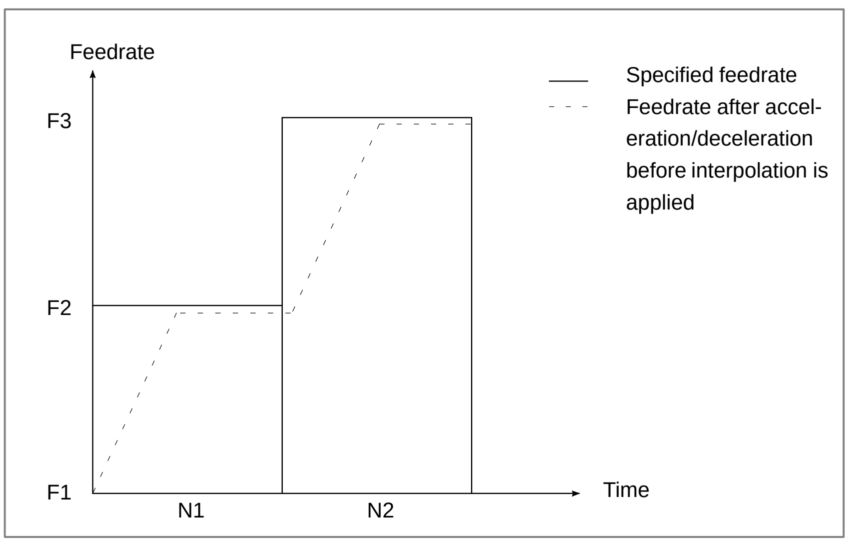


To reduce feedrate F3 to feedrate F2, deceleration must be started at P1. To reduce feedrate F2 to feedrate F1, deceleration must be started at P2.

The tool can be decelerated over several blocks, because several tens of blocks are read in advance.

Example of acceleration

Acceleration is started to reach the specified feedrate for a block when the block is executed.



Look–ahead bell–shaped acceleration/deceleration before interpolation

To use this function, set bit 7 (BDO) and bit 1 (NBL) of parameter No. 8402 to 1, and also set the following parameters:

Parameter No. 8400: Parameter 1 for setting the acceleration used for acceleration/deceleration before interpolation
Parameter No. 8401: Parameter 2 for setting the acceleration used for acceleration/deceleration before interpolation
Parameter No. 8402, bit 5 (DST) = 1, bit 4 (BLK) = 0
Parameter No. 8416: Time needed to reach maximum acceleration

For details, see the description of the parameters.

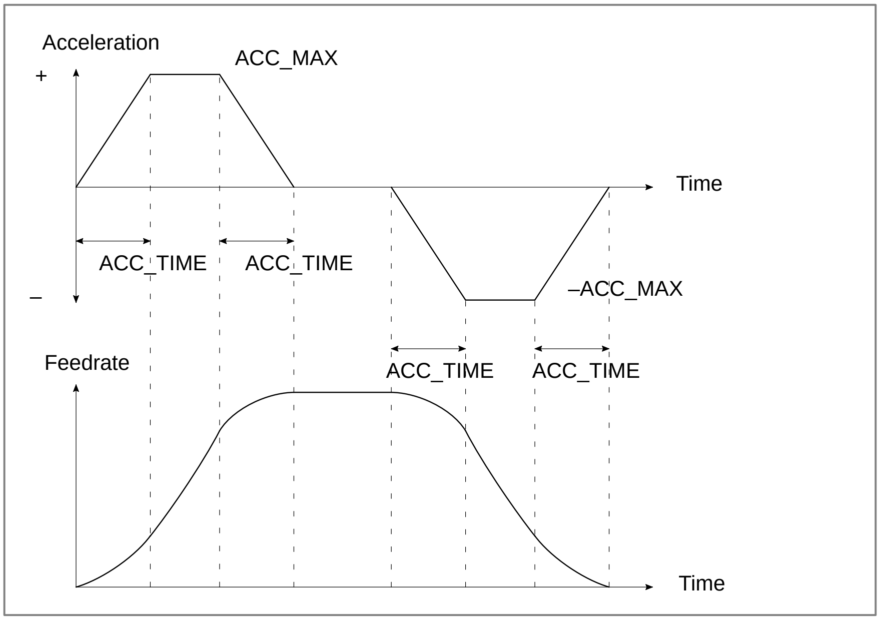
Description

Look–ahead bell–shaped acceleration/deceleration before interpolation controls acceleration as described below.

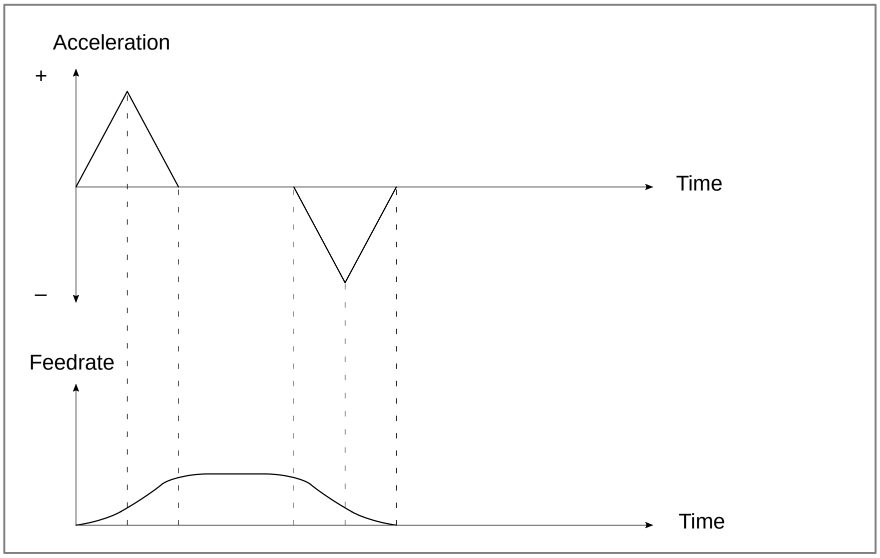
$$\text{Maximum acceleration ACC\_MAX} = \frac{\text{Setting of parameter No. 8400 [mm/min, inch/min]}}{\text{Setting of parameter No. 8401 [ms]}}$$

Time needed to reach maximum acceleration: ACC_TIME = Setting in parameter No. 8416 [ms]

- When maximum acceleration is reached

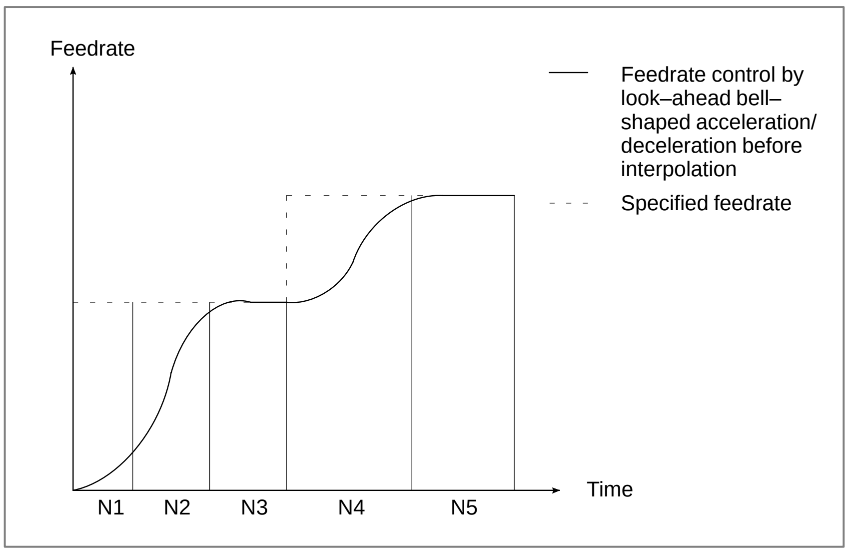


- When maximum acceleration is not reached



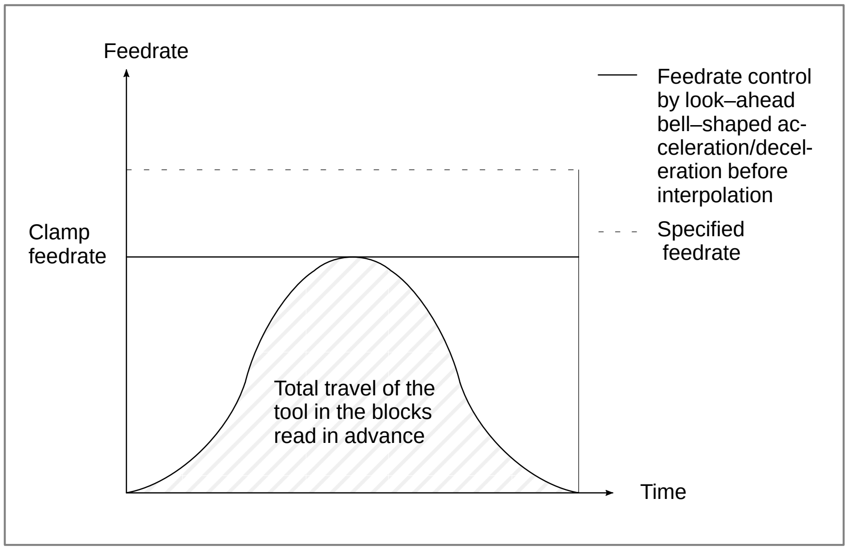
Acceleration

The tool is accelerated to a specified feedrate, starting at the beginning of a block.
The tool can be accelerated over multiple blocks.



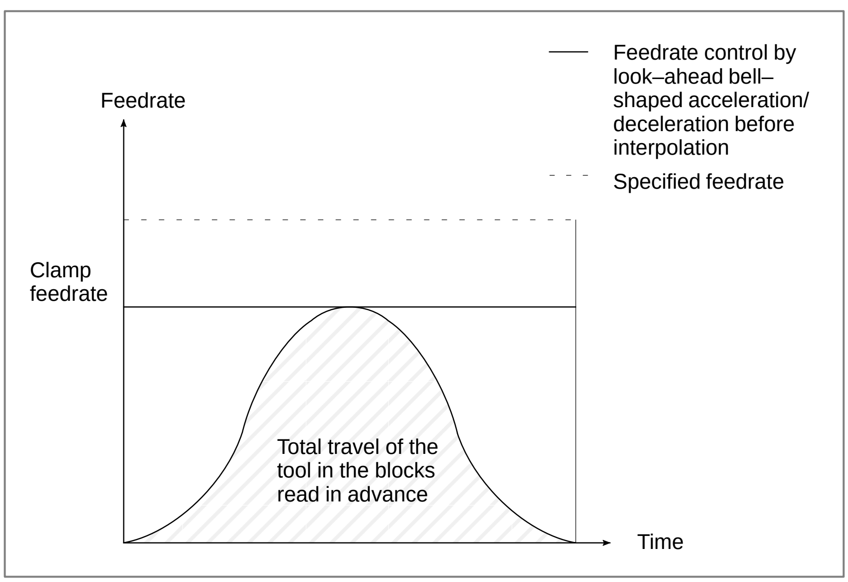
Feedrate clamping based on the total travel of the tool in look–ahead blocks

When the distance required to decelerate the tool from a specified feedrate is less than the total travel of the tool in the blocks read in advance, the feedrate is automatically clamped to a feedrate from which the tool can be decelerated to a feedrate of zero.

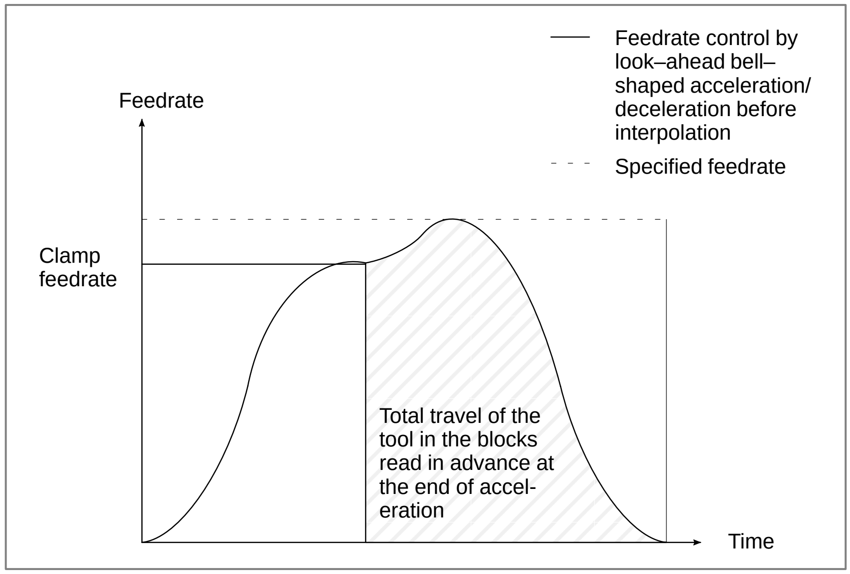


When several blocks, each specifying a short travel, are specified in succession, the following situation can occur:
The total travel of the tool in the blocks read in advance at the start of acceleration is less than the distance required to decelerate the tool from a specified feedrate, but the total travel of the tool in the blocks read in advance at the end of acceleration is greater than the distance required to decelerate the tool from a specified feedrate.
In such a case, the tool is accelerated once and clamped to the feedrate obtained based on the total travel of the tool in the blocks read in advance. Then, the tool is accelerated to a specified target feedrate.

- At the start of acceleration



- At the end of acceleration

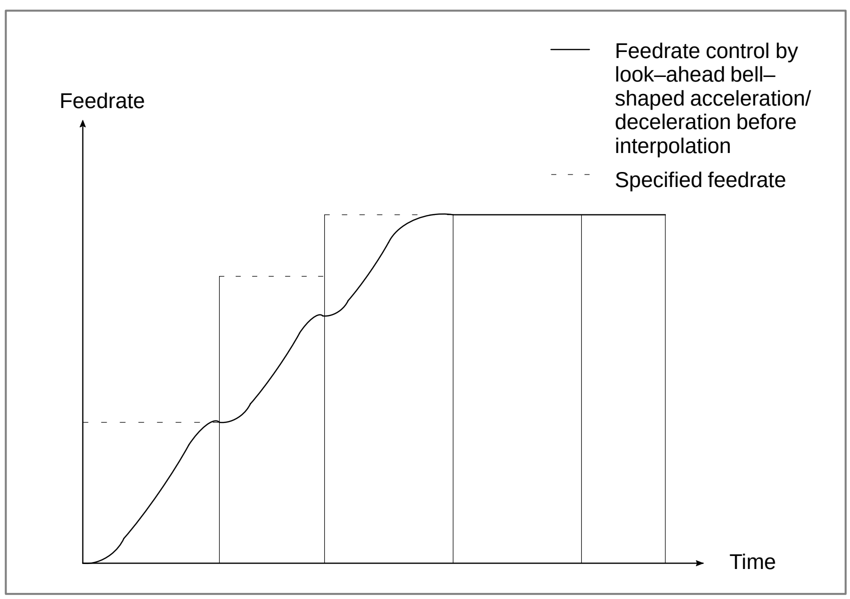


Feedrate command and feedrate

If an F command is changed by, for example, another F command, the corner deceleration function, or the automatic feedrate determination function, look–ahead bell–shaped acceleration/deceleration before interpolation treats the changed feedrate as a new target feedrate, and restarts acceleration/deceleration.

Whenever an F command is changed, bell–shaped acceleration/deceleration is performed.

Bell–shaped acceleration/deceleration is performed each time a different feedrate command is specified, for example, in a program containing successive blocks, each specifying a short travel.



When the feed hold function is used during acceleration

When the feed hold function is used during acceleration, control is performed as described below.

- While applying constant or increasing acceleration

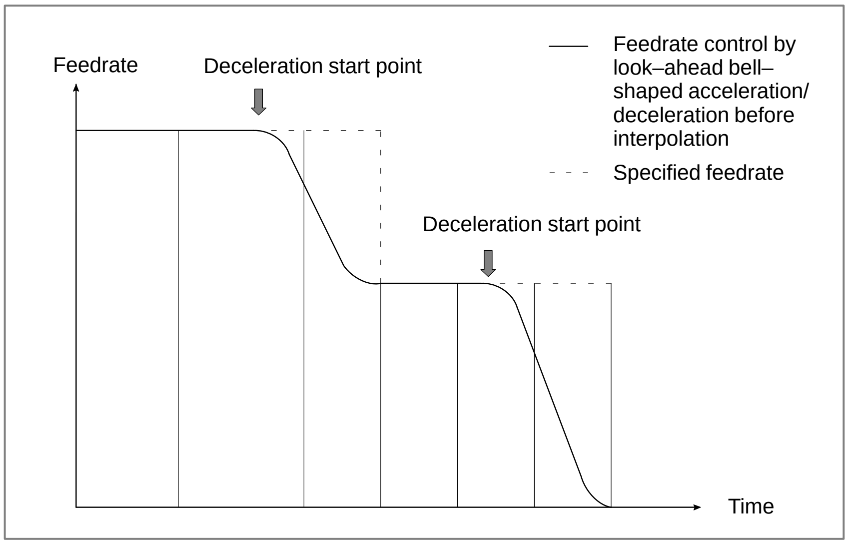
  Starting at the point where the feed hold function is specified, the acceleration is gradually reduced to 0. Then, the feedrate for the tool is gradually reduced to 0. Thus, the feed hold function does not always immediately reduce the feedrate of the tool; it instead may sometimes increase the feedrate for a brief instant before reducing the feedrate.

- While applying decreasing acceleration

  First, the acceleration is gradually reduced to 0. Then, the feedrate is gradually reduced to 0.

Deceleration

The tool is decelerated to the feedrate specified for a block, starting at the previous block.
The tool can be decelerated over multiple blocks.

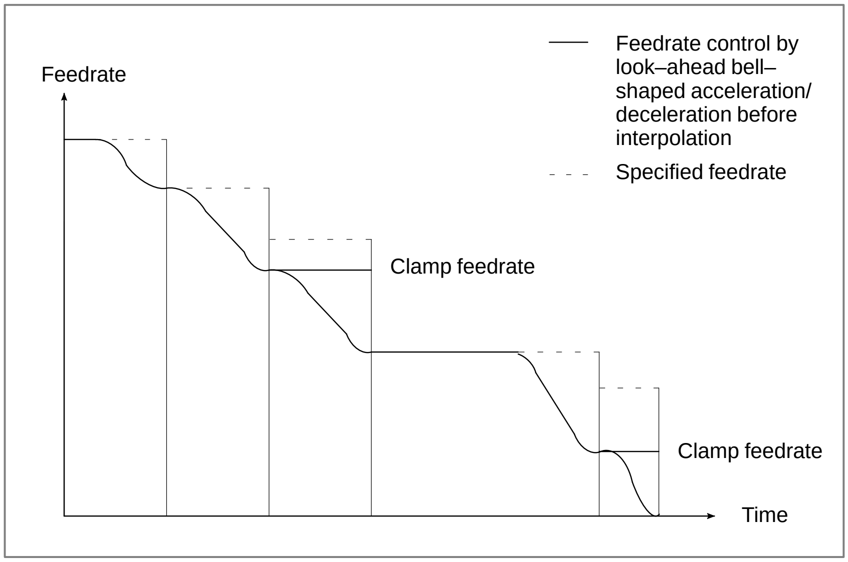


Feedrate command and deceleration

If an F command is changed by, for example, another F command, the corner deceleration function, or the automatic feedrate determination function, look–ahead bell–shaped acceleration/deceleration before interpolation treats the changed feedrate as a new target feedrate, and restarts acceleration/deceleration.

Whenever an F command is changed, bell–shaped acceleration/deceleration is performed.

When the distance required to decelerate the tool from a specified feedrate is longer than the total travel of the tool in the blocks read in advance, the feedrate is automatically clamped, as in the case of acceleration.



Deceleration based on tool travel

The deceleration of the tool is started when the total travel of the tool in the blocks read in advance is less than the distance required to decelerate the tool from the current feedrate.

When the total travel of the tool in the blocks read in advance increases at the end of deceleration, the tool is accelerated.

When blocks specifying a short travel are specified in succession, the tool may be decelerated, then accelerated, then decelerated, and so on, resulting in an unstable feedrate. In such a situation, specify a smaller feedrate.

Feed hold during deceleration

When the feed hold function is used during deceleration, control is performed as described below.

- While applying constant or increasing deceleration

  The point where the deceleration starts being reduced to 0 is shifted from the usually used point (i.e., that used when feed hold is not applied) to ensure that the feedrate for the tool is gradually reduced to 0.

- While applying decreasing deceleration

  The deceleration is gradually reduced to 0, after which the feedrate is reduced to 0.

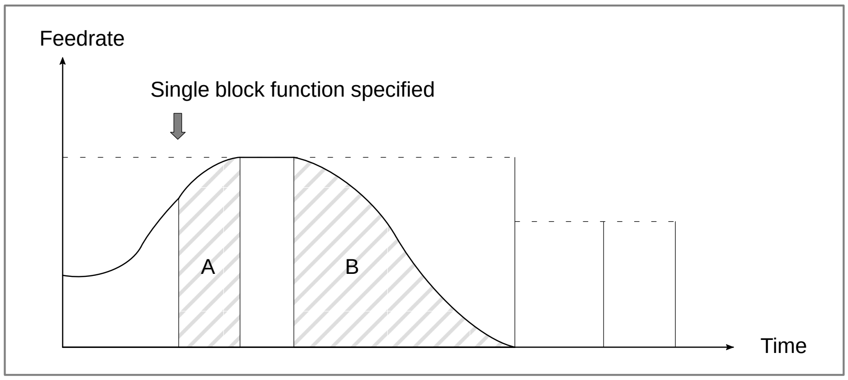
Single block function while look–ahead bell–shaped acceleration/deceleration before interpolation is used

When the single block function is specified while look–ahead bell–shaped acceleration/deceleration before interpolation is used, control is performed as described below.

While the tool is being accelerated or decelerated when the single block function is specified

(1) A + B ≦ Remaining travel for the tool in the block being executed when the single block function is specified

The tool is gradually decelerated so that the feedrate is 0 upon completion of the execution of the block that was being executed when the single block function was specified.



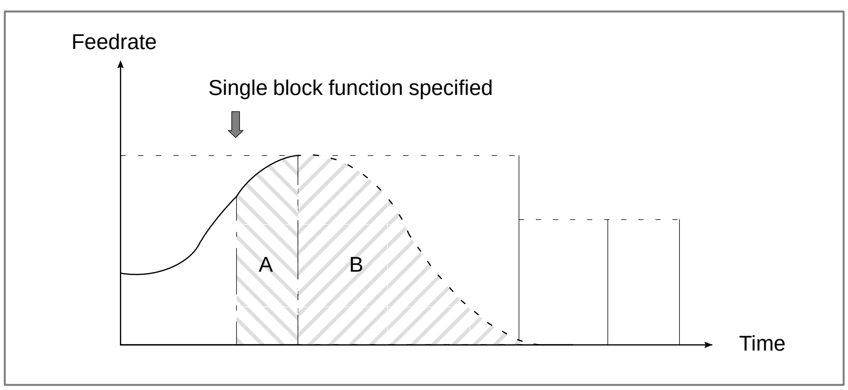
A: Distance traveled before the tool reaches the specified feedrate from the current acceleration/deceleration

B: Distance traveled before the feedrate falls to 0 from a feedrate to which no acceleration/deceleration is applied

(2) A + B > Remaining travel for the tool in the block being executed when the single block function is specified

The tool may be decelerated over multiple blocks until it stops.

How the tool is stopped is described later.



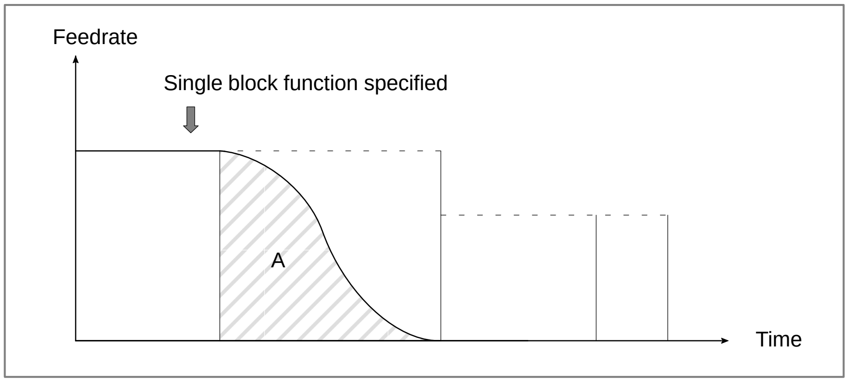
A: Distance traveled before the tool reaches the specified feedrate with the current acceleration/deceleration

B: Distance traveled until the feedrate falls to 0 from a feedrate to which no acceleration/deceleration is applied

While the tool is not being accelerated or decelerated when the single block function is specified

(1) A ≦ Remaining travel for the tool in the block being executed when the single block function is specified

The tool is gradually decelerated so that the feedrate is 0 upon completion of the execution of the block that was being executed when the single block function was specified.

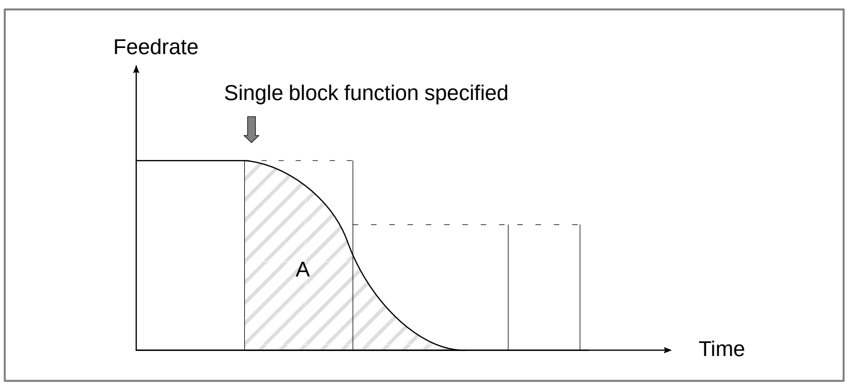


A: Distance traveled until the feedrate falls from the current feedrate value to 0

(2) A > Remaining travel of the tool in the block being executed when the single block function is specified

The tool may be decelerated over multiple blocks until it stops.

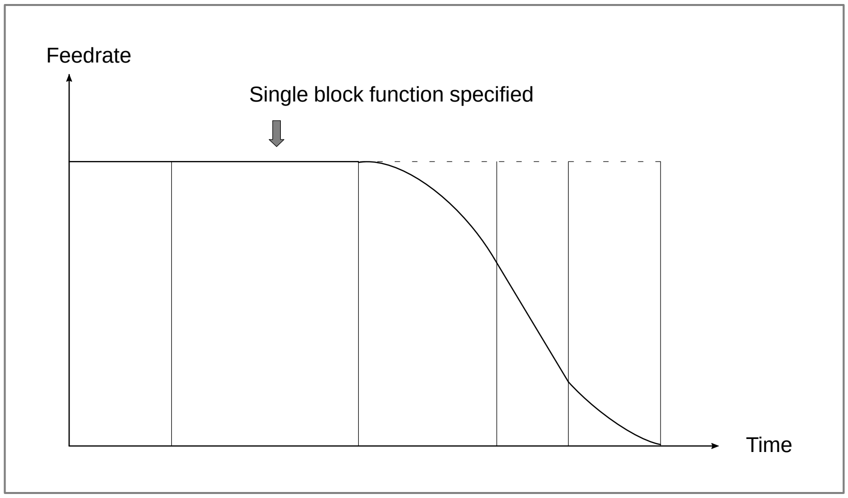
How the tool is stopped is described later.



A: Distance traveled until the feedrate falls from the current feedrate value to 0

How the tool is stopped when decelerated over multiple blocks

The tool is decelerated (or accelerated) over multiple blocks until the feedrate becomes 0.



CAUTION
1 Depending on the stop point and remaining blocks, two or more acceleration/deceleration operations may be performed.
2 When the single block function is specified, an acceleration/deceleration curve recalculation is required while the tool is moving along an axis. So, the tool is not always decelerated over the minimum number of blocks before stopping.

Dryrun/feedrateoverride

When a change in the specification of the dry run function or feedrate override function results in a change in the specified feedrate (feedrate change due to an external cause) while look–ahead bell–shaped acceleration/deceleration before interpolation is being used, control is performed as described below.

While the tool is being accelerated or decelerated when the specification of the dry run function or feedrate override function is changed

After the current acceleration/deceleration operation brings the tool to a specified feedrate and is terminated, the tool is accelerated or decelerated to the new target feedrate.

While the tool is not being accelerated or decelerated when the specification of the dry run function or feedrate override function is changed

The tool is accelerated or decelerated from the current feedrate to the specified feedrate.

Caution

CAUTION

1 When the specification of the dry run function or feedrate override function is changed, the acceleration/deceleration curve must be recalculated while the tool is actually moving along an axis. For this reason, there will be a slight delay before a feedrate change is actually started after the specification of the dry run function or feedrate override function is changed.

2 When the specification of the dry run function or feedrate override function is changed, the tool may be decelerated to below a specified feedrate and then accelerated, depending on the remaining amount of travel, current feedrate, and target feedrate.

7.1.14.2 Automatic feedrate control function

This function reads several tens of blocks ahead to exercise automatic feedrate control.

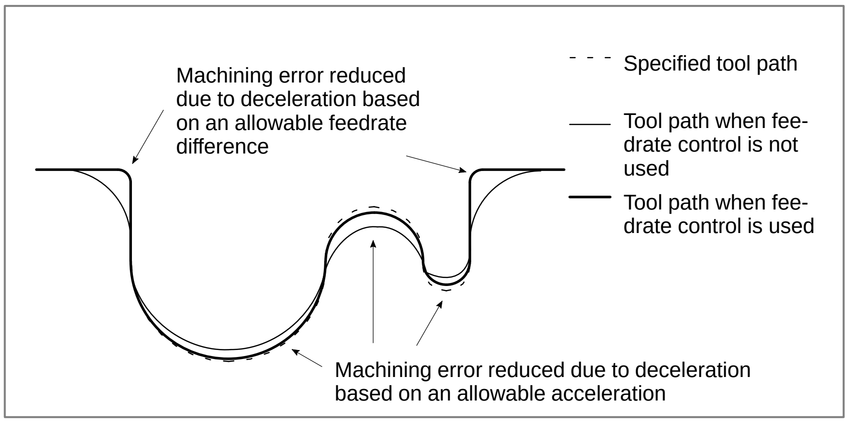
A feedrate is determined on the basis of the conditions listed below. If a specified feedrate exceeds a calculated feedrate, acceleration/deceleration before interpolation is used so that the calculated feedrate can be established.

(1) Feedrate change and specified allowable feedrate difference along each axis at a corner

(2) Anticipated acceleration and specified allowable acceleration along each axis

(3) Cutting load change anticipated from the direction of motion along the Z–axis

In automatic feedrate control mode, the feedrate is automatically reduced with acceleration/deceleration before interpolation to minimize the stress and strain applied to the machine.



To use this function, set bit 0 (USE) of parameter No. 8451 to 1, and set the following parameters:

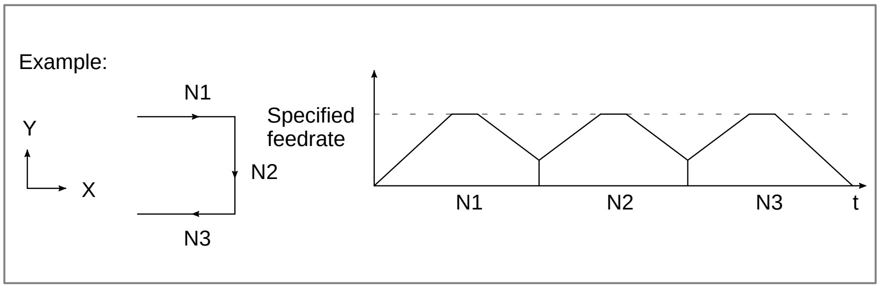
Parameter No. 8410: Allowable feedrate difference used for feedrate determination, based on a corner feedrate difference
Parameter No. 8475, bit 2 (BIP) = 1: Enables deceleration at a corner.
Parameter No. 8470: Parameter specifying an allowable acceleration for feedrate determination, based on acceleration
Parameter No. 8459, bit 1 (CTY) = 1, bit 0 (CDC) = 0
Parameter No. 8464: Initial feedrate for automatic feedrate control
Parameter No. 8465: Maximum allowable feedrate for automatic feedrate control

For details, see the description of each parameter.

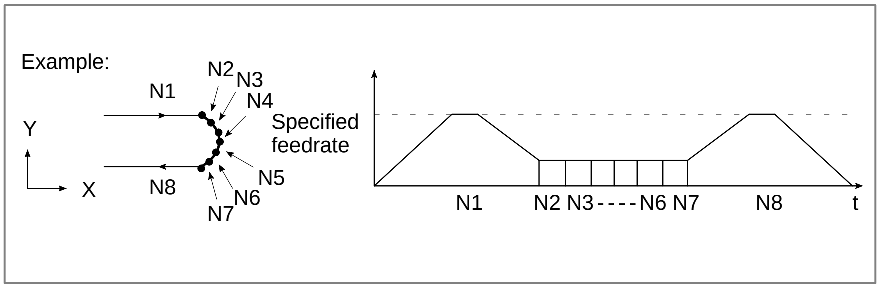
Feedrate control conditions

In automatic feedrate control mode, the feedrate for the tool is controlled as described below.

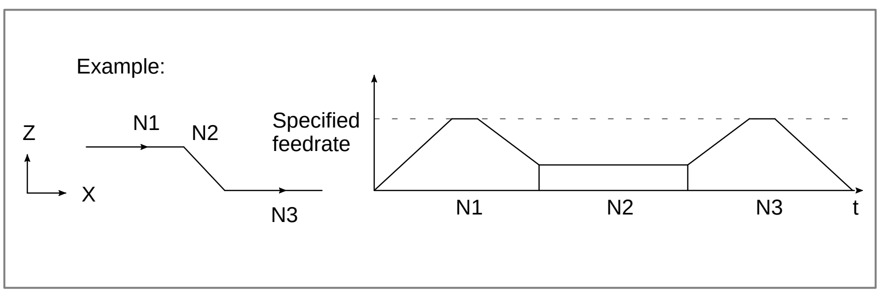
- The feedrate required at a corner is calculated from the specified feedrate difference at the corner along each axis, the tool being decelerated to the calculated feedrate at the corner.



- The feedrate required in a block is calculated from the specified acceleration along each axis at the start point and end point of the corner, the tool being decelerated so that the feedrate in the block does not exceed the calculated feedrate.



- The feedrate required in a block is calculated from the angle of downward movement along the Z–axis, the tool being decelerated so that the feedrate in the block does not exceed the calculated feedrate.



Example of feedrate determination based on a feedrate difference along each axis

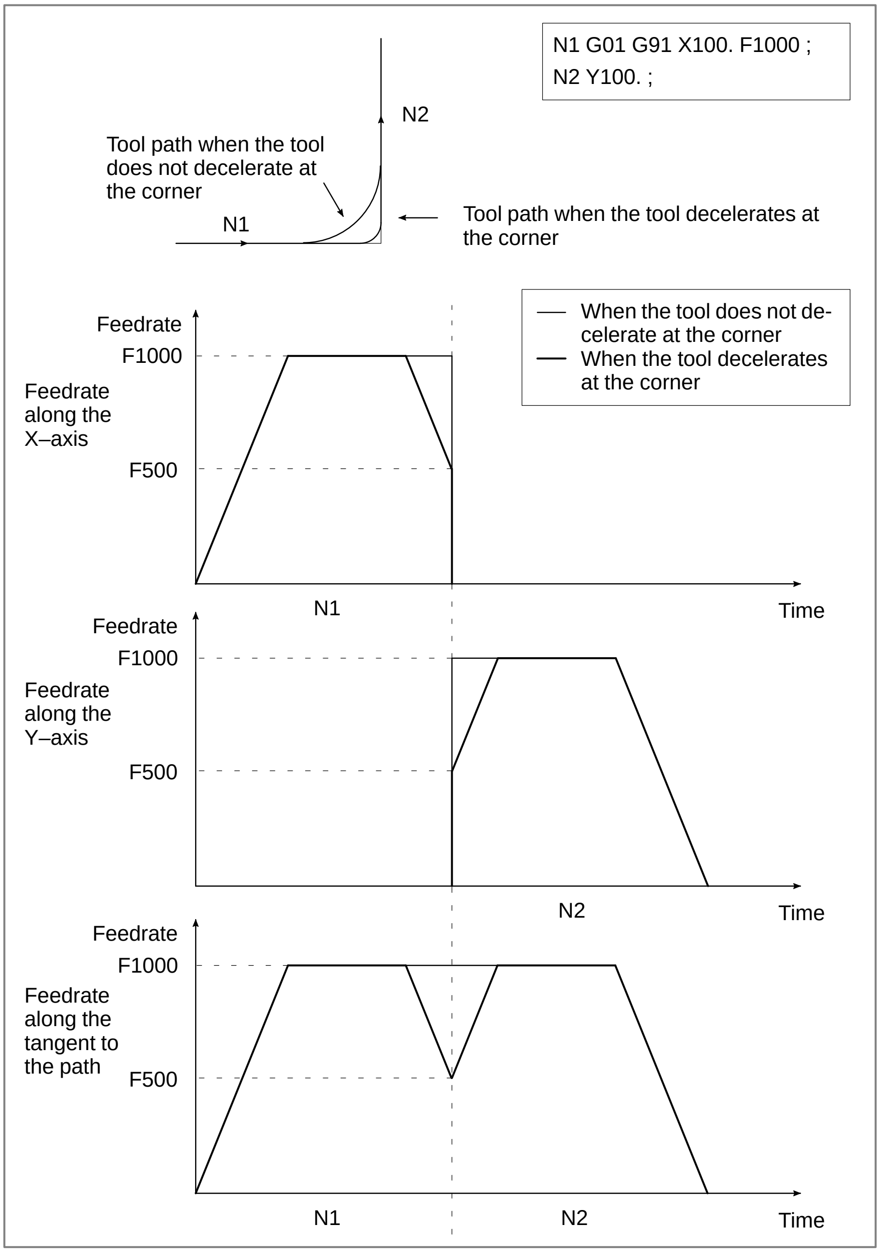
The feedrate required at a corner is calculated from the feedrate difference along each axis, as described below.

When the tool is to move at the specified feedrate F, a comparison is made between the feedrate change along each axis (Vc[X], Vc[Y], ...) and the value (Vmax) set in parameter No. 8410. If Vmax is exceeded by a feedrate change along any axis, the tool is decelerated at the corner to the required feedrate Fc:

$$Fc = F \times \frac{1}{Rmax}$$

where Rmax is the largest value of $R = \dfrac{Vc}{Vmax}$

Suppose that the specified feedrate for the tool is 1,000 mm/min, and that the direction of tool movement changes by 90 degrees (from along the X–axis to along the Y–axis). Suppose also that an allowable feedrate difference of 500 mm/min is set. Then, the tool will decelerate as shown below.



Example of feedrate determination based on acceleration along each axis

As shown below, when a curve is formed by very short successive line segments, there is no significant feedrate difference along each axis at each corner. Consequently, the tool need not be decelerated to compensate for feedrate differences. When taken as a whole, however, successive feedrate differences generate a large acceleration along each axis.

In this case, the tool must be decelerated to minimize the stress and strain imposed on the machine, as well as the machining error that may result from such excessive acceleration. The tool is decelerated to the feedrate at which the acceleration along each axis, found from the formula below, is equal to or less than a specified allowable acceleration.

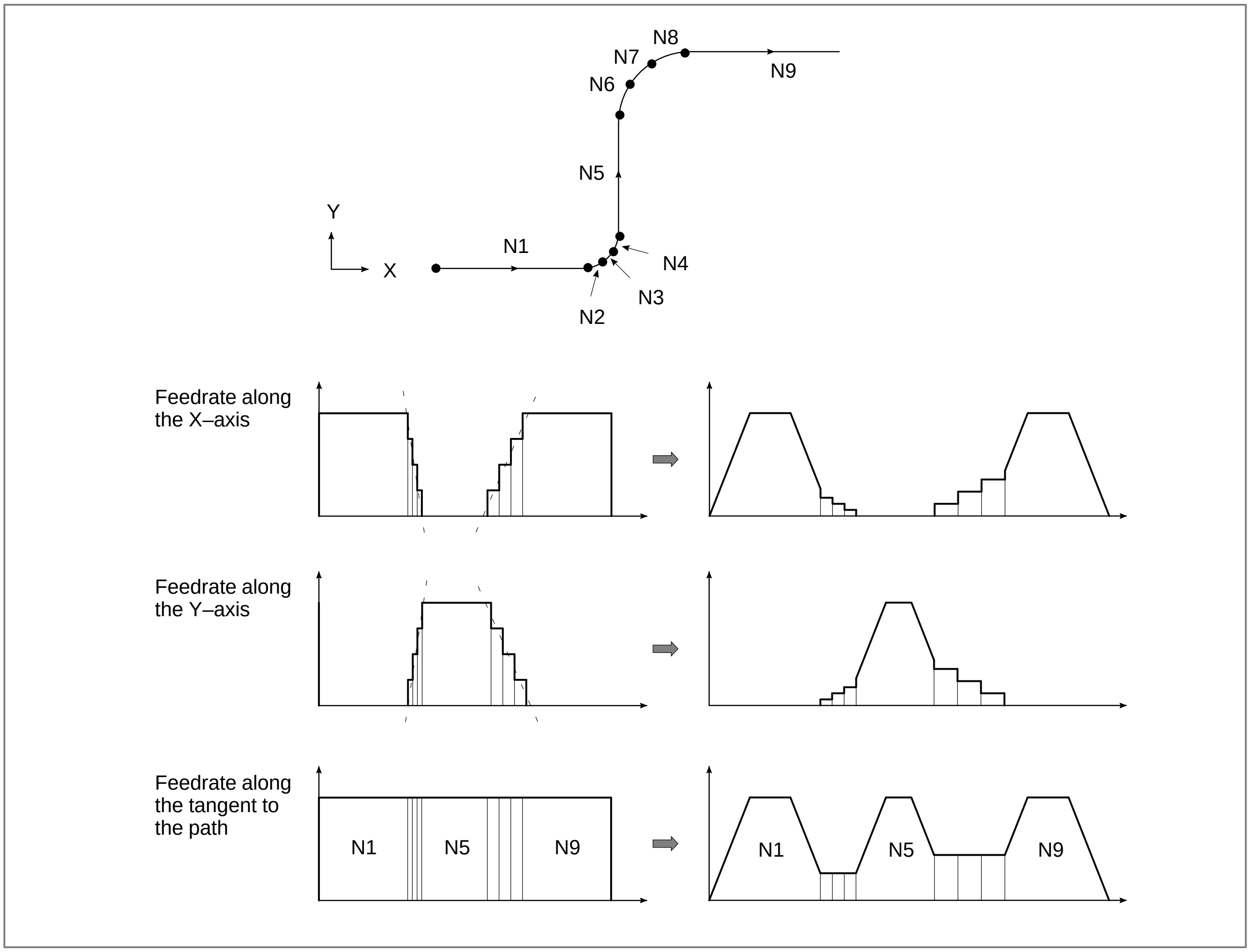
The allowable acceleration is determined from a maximum cutting feedrate (set in parameter No. 1432, No. 1430, or No. 1422) and the time needed to reach the maximum cutting feedrate (set in parameter No. 8470).

Acceleration along each axis =

$$\frac{\text{Feedrate difference along each axis at a corner}}{\max\left(\dfrac{\text{Travel in the previous block}}{F},\ \dfrac{\text{Travel in the next block}}{F}\right)}$$

The reduced feedrate required for each corner is calculated. The tool is decelerated to the decreased feedrate found at either the start point or the end point of each block, whichever is smaller.

Example: In the example shown below, the tool is accelerated too quickly from N2 to N4 and from N6 to N8 (as indicated by the dashed–line inclinations in the feedrate graphs) when automatic feedrate control is not used. So, the tool is decelerated.


N8
N7
N6
N9
N5
Y
X
N1
N4
N3
N2
Feedrate along the X–axis
Feedrate along the Y–axis
Feedrate along the tangent to the path
N1
N5
N9
N1
N5
N9

Feedrate determination based on an allowable acceleration during circular interpolation

When a block specifies circular feed per minute and bit 3 (CIR) of parameter No. 8475 is set to 1, the feedrate of the tool is automatically determined so that the acceleration along each axis does not exceed an allowable acceleration.

The allowable acceleration is determined from the maximum cutting feedrate (set in parameter No. 1432, No. 1430, or No. 1422) and the time needed to reach the maximum cutting feedrate (set in parameter No. 8470).

During circular interpolation, the tool is controlled so that it always moves along the path at the specified feedrate. At this time, the total acceleration of the tool, consisting of the acceleration along each axis, is calculated as follows:

$$\text{Acceleration} = \frac{F^2}{R}$$

F: Feedrate
R: Arc radius

A feedrate is calculated, as shown below, so that the total acceleration does not exceed the smaller of the allowable accelerations along the two axes of circular interpolation. If a specified feedrate is greater than the calculated feedrate, the tool is decelerated to the calculated feedrate.

$$\frac{F^2}{R} = \min(\alpha_x, \alpha_y)$$

$$F = \sqrt{R \times \min(\alpha_x, \alpha_y)}$$

α_x, α_y : Allowable accelerations along X–axis and Y–axis

Example of feedrate determination based on cutting load

This function can be used when bit 4 (ZAG) of parameter No. 8451 is set to 1.

Cutting the workpiece with the end of the cutter (Fig. 7.1.14.2 (b)) incurs a greater resistance than when cutting the workpiece with the side of the cutter (Fig. 7.1.14.2 (a)). Therefore, for (Fig. 7.1.14.2 (b)), the tool must be decelerated. To calculate the required degree of feedrate deceleration, the automatic feedrate control function uses the angle of downward movement of the tool along the Z–axis.

When the tool is moving down along the Z–axis, the angle (θ) of downward movement formed by the XY plane and cutter path is as shown in the Fig. 7.1.14.2 (b). The angle of downward movement is divided into four areas, with an override value for each area specified in a parameter, as follows:
Area 2: Parameter No. 8456
Area 3: Parameter No. 8457
Area 4: Parameter No. 8458

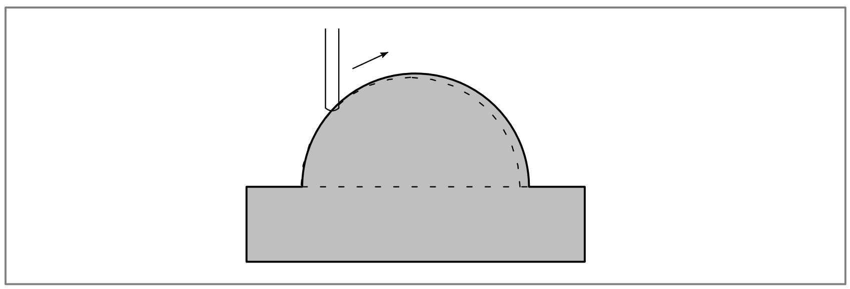


Fig. 7.1.14.2 (a) When the tool is moving up along the Z–axis

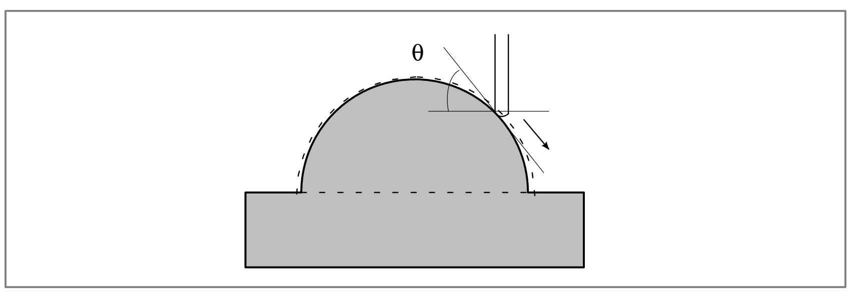
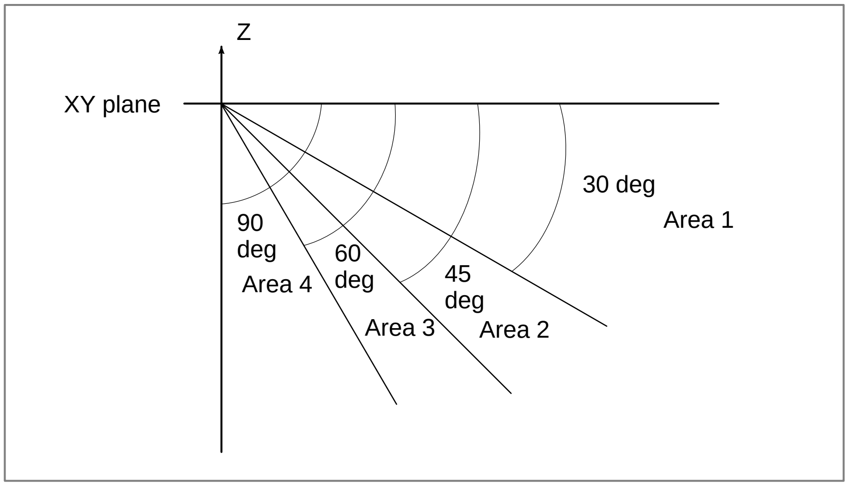


Fig. 7.1.14.2 (b) When the tool is moving down along the Z–axis

No override parameter is provided for area 1; the override value for area 1 is always 100%. A feedrate determined with a separate feedrate control function is multiplied by the override value specified for the area to which the angle θ of downward movement belongs.

Area 1: $0° \leq \theta < 30°$
Area 2: $30° \leq \theta < 45°$
Area 3: $45° \leq \theta < 60°$
Area 4: $60° \leq \theta \leq 90°$



CAUTION

The feedrate determination function that is based on cutting load uses an NC command to determine the direction of movement along the Z–axis. This means that the direction of movement along the Z–axis cannot be found if the movement along the Z–axis is subject to manual intervention with manual absolute on/off function set to on, or if the mirror image function is used with the Z–axis. So, never use these functions when using feedrate determination based on cutting load.

Ignoring F code commands

In a block for which the automatic feedrate control function is enabled, the ignoring of all feed commands (F commands) can be specified by setting bit 7 (NOF) of parameter No. 8451. The feed commands are:

(1) Modal F command specified before a block for which the automatic feedrate control function is enabled

(2) Modal F command and F command specified in a block for which the automatic feedrate control function is enabled

Note, however, that specified F commands and modal F commands are stored in the CNC.

This means that in a block for which the automatic feedrate control function is disabled, a modal F command of (1) or (2) is used instead of a modal F command calculated by the automatic feedrate control function.

Other examples of feedrate determination conditions

If a calculated feedrate exceeds the maximum allowable feedrate for automatic feedrate control, specified in parameter No. 8465 or with an F command, the feedrate is clamped to the maximum allowable feedrate or F command, whichever is smaller.

Automatic speed control in involute interpolation

Involute interpolation automatic speed control overrides a specified feedrate automatically, in the following two ways, during involute interpolation to obtain a high–quality surface with improved machining precision.

- Override with cutter compensation inside offset
- Override near the basic circle

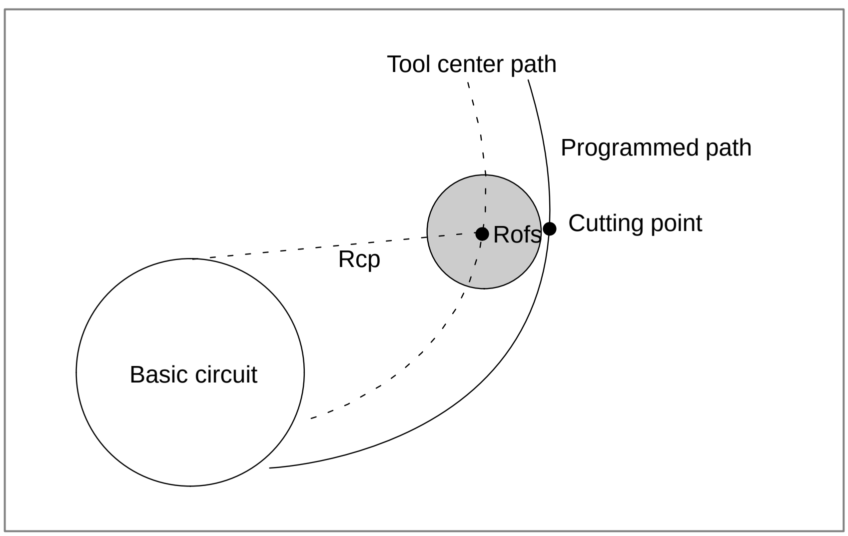
(1) Override with cutter compensation inside offset

In standard involute interpolation mode, when cutter compensation is applied to involute interpolation, control is provided so that the speed in the direction tangential to the path of the tool center (tool center path) is always set to a specified feedrate.

In this case, the actual cutting speed, which is the speed of the tool periphery point (cutting point) on the programmed path, varies because the curvature of the involute curve is constantly changing.

In particular, when the tool offset is inside the involute curve, as the tool approaches the basic circle, the actual cutting speed increases relative to a specified feedrate.

For smooth machining, the actual cutting speed should be controlled to match a specified feedrate. This function calculates an override according to the curvature of the involute curve which changes from moment to moment during involute interpolation, especially when an inside offset is used. Then, the function controls the actual cutting speed which is the speed in the direction tangential to the curve at the cutting point, so that the actual cutting speed always matches a specified feedrate.



The override value is calculated as follows:

$$OVRa = \frac{Rcp}{Rcp + Rofs} \times 100$$

Rcp : Radius of curvature at the tool center on the involute curve that passes through the tool center

Rofs : Tool radius

(2) Override near the basic circle

Near the basic circuit, the change in curvature of an involute curve is relatively large. If such areas are cut at a programmed feedrate, a heavy load may be placed on the cutter, preventing a smooth surface from being produced.

When areas near the basic circuit where the change in curvature of an involute curve is relatively large are cut, this function can reduce the load on the cutter, thus enabling a smooth surface to be obtained by automatically decelerating the tool movement according to the parameter setting.

When the radius of curvature at the cutting point is in the range specified by parameters (Rlmt1) to (Rlmt5), an override is applied as follows:

When Rlmt1 > Rcp $\pm$ Rofs $\geqq$ Rlmt2

$$\mathrm{OVRb} = \frac{100 - \mathrm{OVR2}}{\mathrm{Rlmt1} - \mathrm{Rlmt2}} \times (\mathrm{Rcp} \pm \mathrm{Rofs} - \mathrm{Rlmt2}) + \mathrm{OVR2}$$

When Rlmt2 > Rcp $\pm$ Rofs $\geqq$ Rlmt3

$$\mathrm{OVRb} = \frac{\mathrm{OVR2} - \mathrm{OVR3}}{\mathrm{Rlmt2} - \mathrm{Rlmt3}} \times (\mathrm{Rcp} \pm \mathrm{Rofs} - \mathrm{Rlmt3}) + \mathrm{OVR3}$$

When Rlmt3 > Rcp $\pm$ Rofs $\geqq$ Rlmt4

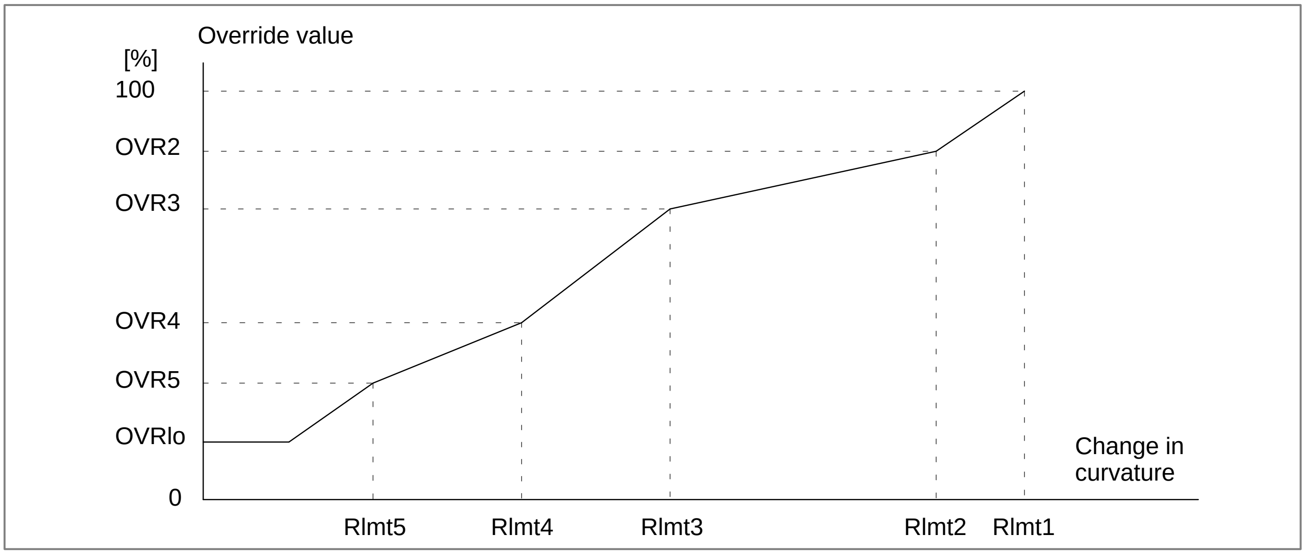
$$\mathrm{OVRb} = \frac{\mathrm{OVR3} - \mathrm{OVR4}}{\mathrm{Rlmt3} - \mathrm{Rlmt4}} \times (\mathrm{Rcp} \pm \mathrm{Rofs} - \mathrm{Rlmt4}) + \mathrm{OVR4}$$

When Rlmt4 > Rcp $\pm$ Rofs $\geqq$ Rlmt5

$$\mathrm{OVRb} = \frac{\mathrm{OVR4} - \mathrm{OVR5}}{\mathrm{Rlmt4} - \mathrm{Rlmt5}} \times (\mathrm{Rcp} \pm \mathrm{Rofs} - \mathrm{Rlmt5}) + \mathrm{OVR5}$$

Rlmt1 to Rlmt5 are set in parameters 5611 to 5615, and OVR2 to OVR5 are set in parameters 5616 to 5619. OVRlo in the graph shown below denotes the lower limit imposed on the override during involute interpolation. It is set in parameter 5620.

Rcp $\pm$Rofs indicates Rcp + Rofs for an inside offset and Rcp – Rofs for an outside offset.



If the override calculation result is below the parameter–set lower limit, the override is clamped to the lower limit.

7.1.14.3 Signal

HPCC mode signal MHPCC (F066#6)

[Classification] Output signal

[Function] Indicates that the system is set to high–precision contour control mode (HPCC mode).

[Output condition] The signal is set to 1 if G05 P10000 (HPCC mode ON) is specified in a program. The signal is set to 0 if G05 P0 is specified in a program or if HPCC mode is canceled by a reset.

HPCC operation signal EXHPCC (F066#7)

[Classification] Output signal

[Function] Indicates that the system is operating in high–precision contour control mode (HPCC operation is in progress).

[Output condition] The signal is set to 1 if G05 P10000 (HPCC mode ON) is specified in a program and if specifiable data of except G00, M, S, T, B is executed.
The signal is set to 0 when:

(1) Automatic operation is halted.

(2) Automatic operation is stopped.

(3) Specifiable data of G00, M, S, T or B is executed.

(4) HPCC mode is canceled.

For the specifiable date, refer to 7.1.14.

Signal addresses

	#7	#6	#5	#4	#3	#2	#1	#0
F066	EXHPCC	MHPCC						

7.1.14.4 Parameter

Parameters of linear acceleration and deceleration before interpolation

8400	Parameter 1 for determining a linear acceleration/deceleration before interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

This parameter determines a linear acceleration and deceleration before interpolation. Usually, set the maximum cutting speed (parameter No. 1422).

8401	Parameter 2 for determining a linear acceleration/deceleration before interpolation

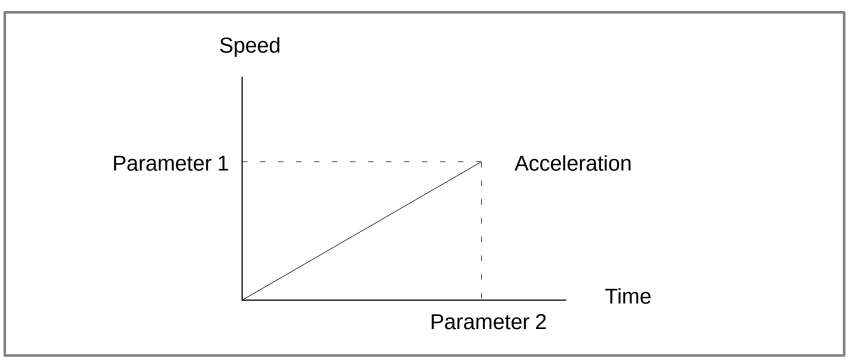
[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter specifies the time required until the speed specified in parameter 1 is achieved.

> **NOTE**
> The function for linear acceleration/deceleration before interpolation is canceled when either parameter no. 8400 or 8401 is set to 0.



	#7	#6	#5	#4	#3	#2	#1	#0
8402	BDO		DST	BLK			NBL	

[Data type] Bit

BDO, NBL Set the type of acceleration/deceleration before interpolation.

BDO	NBL	Meaning
0	0	Acceleration/deceleration prior to interpolation is of linear type
1	1	Acceleration/deceleration prior to interpolation is of bell shape type

BLK Be sure to set to 0.

DST Be sure to set to 1.

Parameters of advanced preview bell–shaped acceleration/deceleration before interpolation

8400	Parameter 1 for determining a acceleration/deceleration before interpolation

[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

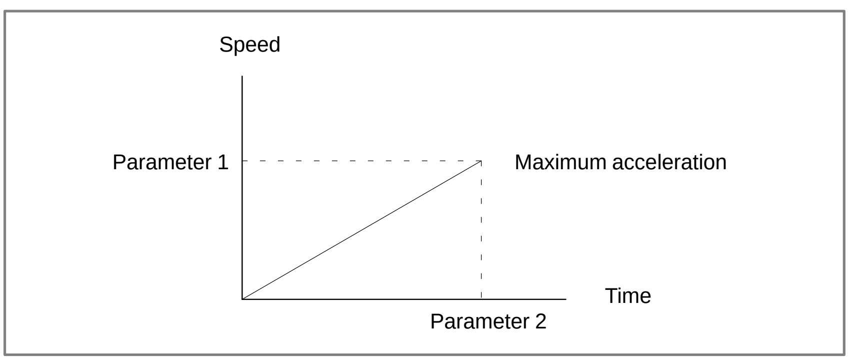
8401	Parameter 2 for determining a acceleration/deceleration before interpolation

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 0 to 4000

This parameter specifies the time required until the speed specified in parameter 1 is achieved.



8416	The time required to the maximum acceleration in advanced preview bell–shaped acceleration/deceleration before interpolation

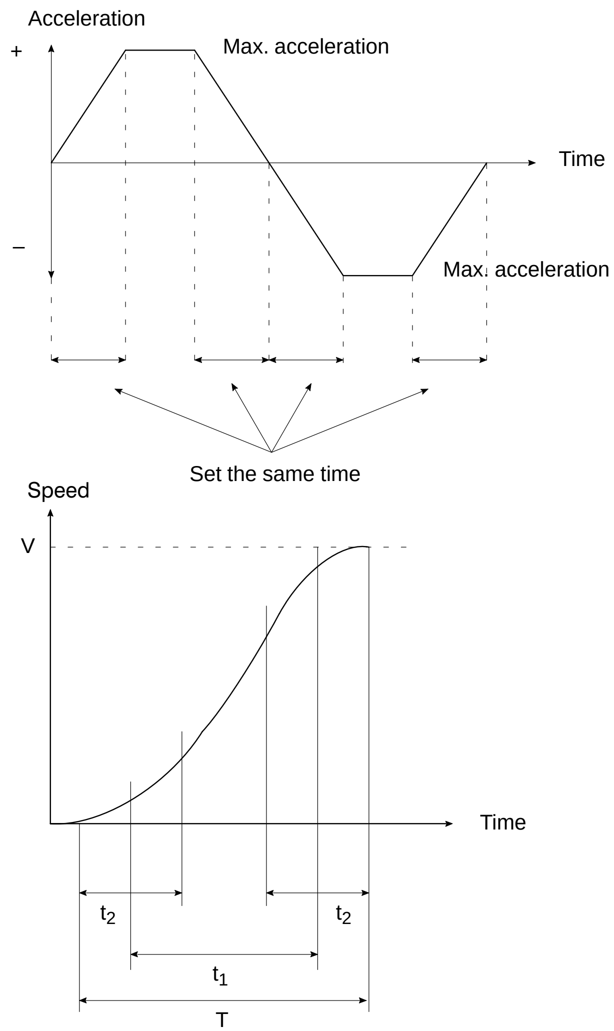
[Data type] Two–word

[Unit of data] mec

[Valid data range] 0 to 99999999

This parameter sets the time required to reach the maximum acceleration in advanced preview bell–shaped acceleration/deceleration before interpolation.

Also, this parameter is used for deceleration time taken from the maximum acceleration to zero.



t1: Time constant assumed in linear acceleration/deceleration (Parameter No. 8401)
t2: Time for corner rounding (parameter No. 8416)
V: Speed to set acceleration time t1 (Parameter No. 8400)
Usually set the max. cutting speed.

Total time =T
Time of linear part $=T-2 * t_2$
Time of curved part $=t_2$

When target speed is different, total time also changes (constant acceleration).

Parameters of automatic feedrate control

8410	Allowable velocity difference in velocity determination considering the velocity difference at corners

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	10 – 60000	1 – 6000
Inch machine	0.1 inch/min	10 – 60000	1 – 6000
Rotation axis	1 deg/min	10 – 60000	1 – 6000

If zero is specified for all axes, the machine does not decelerate at corners.

When the function for determining the velocity considering the velocity difference at corners is used, the system calculates the feedrate whereby a change in the velocity element of each axis does not exceed this parameter value at the interface between blocks. Then the machine decelerates using acceleration/ deceleration before interpolation.

	#7	#6	#5	#4	#3	#2	#1	#0
8451	NOF			ZAG				USE

[Data type] Bit

USE Automatic velocity control is:

0 : Not applied.
1 : Applied.

ZAG The velocity is:

0: Not determined according to the angle at which the machine descends along the Z–axis.
1: Determined according to the angle at which the machine descends along the Z–axis.

NOF In a block where automatic velocity control is validated, the F command is:

0 : Validated.
1 : Ignored.

(Maximum speed of automatic feedrate control set by parameter No. 8465 is used for command speed in spite of F command)

8452	Range of velocity fluctuation to be ignored

[Data type] Byte

[Unit of data] %

[Valid data range] 0 to 100 (Standard setting: 10)

8456	Area–2 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 80)

This parameter specifies an override in area 2 of velocity calculation considering the cutting load.

8457	Area–3 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 70)

This parameter specifies an override in area 3 of velocity calculation considering the cutting load.

8458	Area–4 override

[Data type] Word

[Unit of data] %

[Valid data range] 1 to 100 (Standard setting: 60)

This parameter specifies an override in area 4 of velocity calculation considering the cutting load.

	#7	#6	#5	#4	#3	#2	#1	#0
8459							CTY	CDC

[Data type] Bit

CDC Be sure to set this value to 0.

CTY Be sure to set this value to 1.

8464	Initial feedrate for automatic feedrate control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	0 – 600000	0 – 6000
Inch machine	0.1 inch/min	0 – 600000	0 – 6000
Rotation axis	1 deg/min	0 – 600000	0 – 6000

This parameter sets the initial feedrate for automatic feedrate control.

In automatic feedrate control, the initial feedrate set with this parameter is used at the beginning if no F command is specified in the program. Usually, set the maximum cutting feedrate (specified in parameter No. 1422).

8465	maximum allowable feedrate for automatic feedrate control

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	10 – 240000	1 – 100000
Inch machine	0.1 inch/min	10 – 96000	1 – 48000
Rotation axis	1 deg/min	10 – 240000	1 – 100000

This parameter sets the maximum allowable feedrate for automatic feedrate control. Usually, set the maximum allowable cutting feedrate (set in parameter No. 1422).

8470	Parameter for determining allowable acceleration in feedrate calculation considering acceleration

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 32767

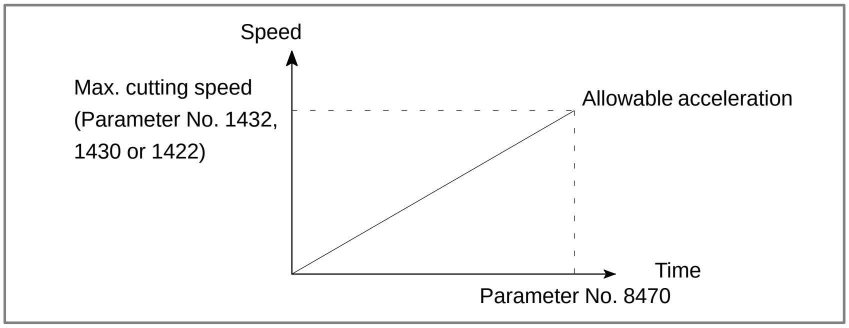
When the function for calculating the feedrate considering the acceleration is used under automatic feedrate control, this parameter is used to determine the allowable acceleration. The time required until the maximum cutting feedrate is reached must be specified here.

Allowable acceleration is determined from the maximum cutting feedrate and the value set in this parameter. Where, the maximum cutting feedrate is any of value set in parameter No. 1432, 1430 or 1422. Which parameter No. is used depends on the following conditions:

· When a value other than 0 is set to No. 1432, the value set to No. 1432 is used.

· When 0 is set to No. 1432 and a value other than 0 is set to No. 1430, the value set to No. 1430 is used.

· When 0 is set to No. 1432 and 1430, the value set to No. 1422 is used.



	#7	#6	#5	#4	#3	#2	#1	#0
8475					CIR	BIP		

[Data type] Bit

CIR The function of automatic feedrate control considering acceleration and deceleration during circular interpolation is:

0 : Not used.
1 : Used.

When 1 is set, parameter NO. 8470 for determining the allowable acceleration must be specified.

BIP The function of deceleration at corners is:

0 : Not used.
1 : Used. (Always set 1.)

Parameters of axis control

7510	Maximum number of axes in High Precision Contour Control

[Data type] Byte

[Valid data range] 1, 2, 3, … to the maximum number of control axes

This parameter specifies the maximum number of axes to controlled by High Precision Contour Control.

Example) Axis configuration is X, Y, Z, A, B, and C from the 1st axis in this order and to make HPCC valid to the 4th axis (A), set this parameter to 4. In this case, HPCC is also effective for the X, Y, Z axes.

X, Y, Z, A axes Axes on which HPCC is valid
B, C axes . Axes on which HPCC is not valid.

	#7	#6	#5	#4	#3	#2	#1	#0
8480		RI2	RI1	RI0				

[Data type] Bit

Set the interpolation frequency during the high precision contour control mode (HPCC mode).
Be sure to set the following values:

RI2	RI1	RI0
0	1	0

Parameters of acceleration/deceleration after interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
1602		LS2						

[Data type] Bit

LS2 Acceleration/deceleration after interpolation for cutting feed in the high precision contour control mode (HPCC mode) is:

0 : Not used. (Exponential acceleration/deceleration)
1 : Used. (The function for linear acceleration/deceleration after interpolation for cutting feed is required.)

1768	Time constant for linear acceleration/deceleration during cutting feed in HPCC mode

[Data type] Word axis

[Unit of data] ms

[Valid data range] 8 to 512

> **NOTE**
> The function for linear acceleration/deceleration after interpolation is required.

Parameters of cutter compensation C

	#7	#6	#5	#4	#3	#2	#1	#0
5000								SBK

[Data type] Bit

SBK An internally created block for cutter compensation C:
0 : Does not cause a single block stop.
1 : Cause a single block stop.

	#7	#6	#5	#4	#3	#2	#1	#0
5003				BCK	ICK			

[Data type] Bit

ICK In HPCC mode, when cutter compensation C interference check is:
0 : Done
1 : Not done

BCK In HPCC mode, when cutter compensation C interference check determines that the programmed move direction differs from the offset move direction by between 90 and 270 degrees:
0 : An alarm is issued.
1 : No alarm is issued.

Parameters related to involute interpolation

5611	Radius of curvature at the cutting point for the start point of override 1 near the basic circle
5612	Radius of curvature at the cutting point for the start point of override 2 near the basic circle
5613	Radius of curvature at the cutting point for the start point of override 3 near the basic circle
5614	Radius of curvature at the cutting point for the start point of override 4 near the basic circle
5615	Radius of curvature at the cutting point for the start point of override 5 near the basic circle

[Data type] 2–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] 1 to 99999999

5616	Override value at the start of override 2 near the basic circle
5617	Override value at the start of override 3 near the basic circle
5618	Override value at the start of override 4 near the basic circle
5619	Override value at the start of override 5 near the basic circle

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100

5620	Lower limit imposed on the override during involute interpolation

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 100

5621	Allowable acceleration in constant acceleration control during involute interpolation

[Data type] Word

[Unit of data] ms

[Valid data range] 1 to 32767

5622	Minimum speed in constant acceleration control during involute interpolation

[Data type] Word

[Unit of data] ms

[Valid data range] 1 to 32767

The other parameters

	#7	#6	#5	#4	#3	#2	#1	#0
8403	SG0				LM2	LM1	MSU	

[Data type] Bit

MSU When G00, or an M, S, T, or B code is specified in HPCC mode:

0 : An alarm is issued.
1 : The CNC executes the command.

LM1 In HPCC mode, a stroke check before movement for stored stroke limit 1 is:

0 : Not performed.
1 : Performed.

LM2 In HPCC mode, a stroke check before movement for the second stored stroke limit is:

0 : Not performed.
1 : Performed.

SG0 When G00 is specified in HPCC mode:

0 : The setting of bit 1 (MSU) of parameter No. 8403 is followed.
1 : The tool is moved along the axis at the feedrate set with parameter No. 8481, replacing the G00 command with the G01 command, regardless of the setting made for bit 1 (MSU) of parameter No. 8403. Refer to note 1 in description of parameter No. 8481.

8481	Rapid–traverse rate in HPCC mode

[Data type] Two–word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	0 – 600000	0 – 60000
Inch machine	0.1 inch/min	0 – 600000	0 – 60000
Rotation axis	1 deg/min	0 – 600000	0 – 60000

When bit 7 (SG0) of parameter No. 8403 is set to 1, this parameter sets the rapid traverse rate in the HPCC mode.

CAUTION
The G00 command is replaced with the G01 command before execution. So, even if a feedrate is specified for two axes, the rapid traverse rate set with this parameter is always used.
Example:
If the following command is specified when a rapid traverse rate of 1000 mm/min is set F1000, rather than F1414, is used:
G00 X100.Y100.;
For details, refer to Notes on positioning (G00).

	#7	#6	#5	#4	#3	#2	#1	#0
8485			CDS	INV	PRW	G02	G81	G51

[Data type] Bit

G51 In high–precision contour control (HPCC) mode, the scaling/coordinate system rotation functions are:

0 : Disabled.
1 : Enabled.

G81 In high–precision contour control (HPCC) mode, the hole machining canned cycle is:

0 : Disabled.
1 : Enabled.

G02 In high–precision contour control (HPCC) mode, helical interpolation is:

0 : Disabled.
1 : Enabled.

PRW In high–precision contour control (HPCC) mode, parameter rewriting by PMC WINDOW is:

0 : Disabled.
1 : Enabled.

INV In high–precision contour control (HPCC) mode, involute interpolation is:

0 : Disabled.
1 : Enabled.

CDS In high–precision contour control (HPCC) mode, smooth interpolation is:

0 : Disabled.
1 : Enabled.

7.1.14.5 Alarm and message

Number	Message	Description
5000	ILLEGAL COMMAND CODE(HPCC)	An invalid code was specified in HPCC mode.
5003	ILLEGAL PARAMETER (RISC)	Parameter setting is erroneous.
5004	RISC NOT READY	RISC processor board is not ready state.
5006	TOO MANY WORD IN ONE BLOCK	The number of words in a block exceeds allowable range (HPCC mode).
5012	G05 P10000 ILLEGAL START UP	G05P10000 was specified in a mode from which HPCC mode cannot be entered.
5013	HPCC : CRC OFS REMAIN AT CANCEL	G05P0 was specified in G41/G42 mode or a state in which an offset value remains.

7.1.14.6 Note

- **Acceleration/deceleration before interpolation in look–ahead blocks**

> **NOTE**
> If there is a series of very short blocks, for each of which the rate of acceleration/deceleration before interpolation is low, the actual feedrate may not reach the programmed feedrate.

- **Automatic feedrate control**

> **NOTE**
> 1 If the upper limit for automatic feedrate control is set to 0 in parameter No. 8465, no feedrate exceeding 0 is permitted, such that the issue of an F command causes PS alarm 011 (FEED ZERO). To prevent this, specify a value other than zero in the parameter.
> 2 If the override is changed while the automatic feedrate control function is enabled, the calculated clamp feedrate is overridden.

- **Notes on operation**

> **WARNING**
> 1 In HPCC mode, axial interlocking (each axis, each direction) is inhibited (signals G130, G132, G134).
> 2 In HPCC mode, the external mirror image (DI) signal, a mirror image determined by setting data, or axial machine lock must not be changed. Pocket calculator type decimal point input is disabled in HPCC mode (when bit 0 of parameter No. 3401 is set to 1).

> **NOTE**
> 1 A single–block stop cannot be made at the end of the G05P10000 block.
> 2 External deceleration, the F1–digit command, and automatic corner override are disabled.
> 3 In HPCC mode, the operation mode cannot be switched to MDI mode. Also, MDI operation is not permitted.
> 4 A program including G50 P10000; cannot be resumed.

Notes on positioning (G00)

When executing a G00 command when bit 7 of parameter No. 8403 (SG0) is set to 1, note the following:

WARNING
Linear interpolation positioning is performed.

CAUTION
1 The G00 command is replaced with the G01 command upon being executed. Even if two axes are specified, movement is performed at the feedrate specified in parameter No. 8481.
Example)
If the following command is specified when parameter No. 8481 is set to 1000 mm/min, F1000 is executed instead of F1414:
G00 X100. Y100.;
2 Because the G00 command is replaced with the G01 command upon being executed, rapid traverse override is disabled, the cutting feedrate override being enabled instead.

NOTE
1 Because the G00 command is replaced with the G01 command upon being executed, acceleration/deceleration after interpolation is done using the time constant of acceleration/deceleration after interpolation for cutting feed.
2 Because the G00 command is replaced with the G01 command upon being executed, acceleration/deceleration before interpolation in RISC HPCC mode is enabled. (Both linear acceleration/deceleration and bell–shaped acceleration/deceleration are supported.)
3 In–position check is not executed.

7.1.14.7 Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.19.5	High–precision contour control

7.1.15 Positioning by Optimul Acceleration

General

When a rapid traverse command is specified during automatic operation, the function for positioning by optimul acceleration can be used to adjust the rapid traverse rate, time constant, and loop gain, according to the amount of travel for the block. This reduces the time required for positioning and position check, therefore reducing the cycle time.

When rapid traverse is specified in automatic operation, the function adjusts the rapid traverse rate, time constant, and loop gain to one of seven levels, according to the amount of travel for the block. The relationship between the amount of travel and the corresponding rapid traverse rate, time constant, and loop gain are specified in parameters. This function is not effective for cutting feed. (For cutting feed, the loop gain set in parameter 1825 is used.)

Valid commands

This function is effective for all commands that cause rapid traverse in automatic operation, including commands for canned cycles, automatic reference position return, and machine coordinate system selection, as well as the G00 command. This function, however, becomes ineffective in the following cases, and the feedrate, time constant, and loop gain set in parameters 1420, 1620 and 1825 are used:

(1) When an automatic reference position return (G28) is specified before the reference position is established

(2) When the G00 command is executed on the RISC side in high–precision contour control (when bit 7 (SG0) of parameter 8403 is set to 1)

(3) In simple high–precision contour control mode

(4) In Cs axis contour control mode

Cutting feed

This function is not effective for cutting feed.

Manual operation

This function is not effective in manual operation. The function is not effective for movement caused by manual numeric commands.

Manual intervention

When rapid traverse is stopped by feed hold or mode change during automatic operation, and manual operation is performed, the loop gain is changed to the value set in parameter 1825. When the program is restarted, the interrupted block is resumed without changing the loop gain used in manual operation. Then, when the next and subsequent blocks are executed, one of the values set in parameters 6181 to 6187 is used again as the loop gain.

PMC axis control

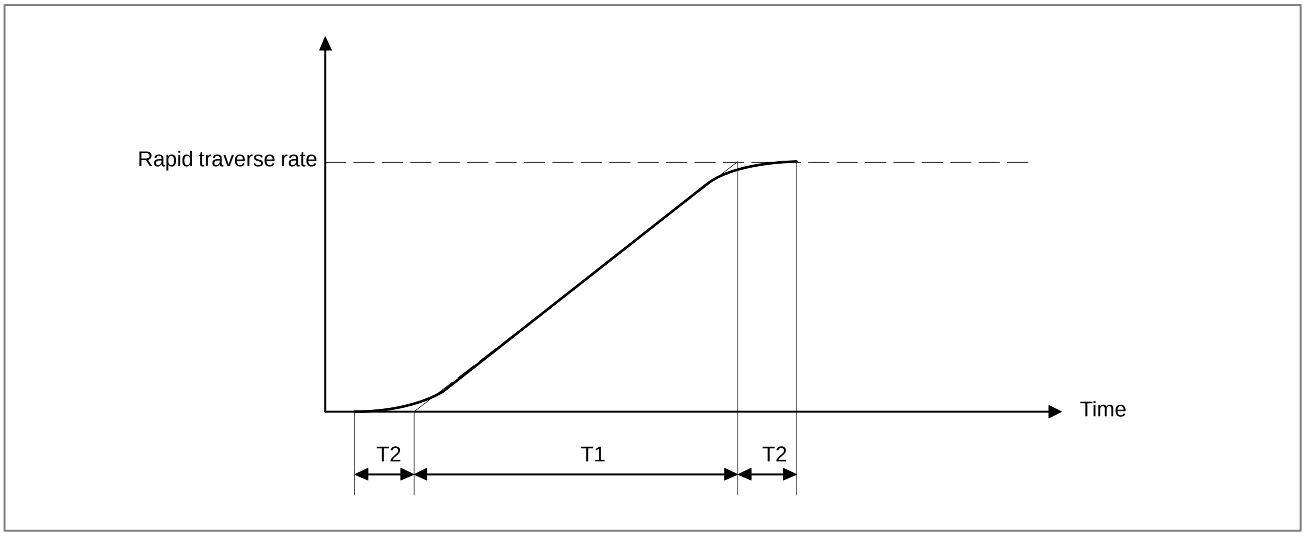
This function is not effective during PMC axis control.

Rapid traverse overlap

When this function is enabled, rapid traverse overlap (enabled when bit 4 (RTO) of parameter 1601 is set to 1) is not performed.

Rapid traverse bell–shaped acceleration/deceleration

When rapid traverse bell–shaped acceleration/deceleration is used, T1 in the figure below and the rapid traverse rate are adjusted. T2 is not adjusted.



Angular axis control

When angular axis control is used, adjustment is performed according to the amount of travel in the Cartesian coordinate system.

Coordinate system rotation, scaling

In coordinate system rotation mode, adjustment is performed according to the amount of travel after coordinate system rotation. In scaling mode, adjustment is performed in a similar manner.

Cutter compensation C, tool–tip radius compensation

In cutter compensation C or tool–tip radius compensation mode, adjustment is performed according to the amount of travel after compensation.

Three–dimensional coordinate conversion

In three–dimensional coordinate conversion mode, adjustment is performed according to the amount of travel before coordinate conversion.

Polar coordinate interpolation

In polar coordinate interpolation mode, adjustment is performed according to the amount of travel in the Cartesian coordinate system.

Simple synchronous control

When using this function with simple synchronous control, enable this function for both the master and slave axes (by setting bit 0 (OAD) of parameter 6131 to 1), and set the same rapid traverse rate, time constant, and loop gain values for both the master and slave axes.

Rapid traverse of linear interpolation type

Even when rapid traverse of linear interpolation type is being used (when bit 1 (LRP) of parameter 1401 is set to 1), adjustment is performed. In this case, however, the feedrate for each axis is determined so that the tool path becomes linear. Therefore, the tool does not always move at a parameter–set rapid traverse rate (parameters 6161 to 6167).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6131								OAD

[Data type] Bit axis

OAD The function for positioning by optimul acceleration is:

0 : Disabled.
1 : Enabled.

6141	Distance D1 for level 1 (metric input, or rotation axis)
6142	Distance D2 for level 2 (metric input, or rotation axis)
6143	Distance D3 for level 3 (metric input, or rotation axis)
6144	Distance D4 for level 4 (metric input, or rotation axis)
6145	Distance D5 for level 5 (metric input, or rotation axis)
6146	Distance D6 for level 6 (metric input, or rotation axis)

6151	Distance D1 for level 1 (inch input)
6152	Distance D2 for level 2 (inch input)
6153	Distance D3 for level 3 (inch input)
6154	Distance D4 for level 4 (inch input)
6155	Distance D5 for level 5 (inch input)
6156	Distance D6 for level 6 (inch input)

[Data type] 2–word

[Unit of data]

Increment system	IS–B	IS–C	Unit
Metric input, rotation axis	0.001	0.0001	mm, deg
Inch input	0.0001	0.00001	inch

[Valid data range] 0 to 99999999

These parameters set the positioning distances used when the function for adjusting the rapid traverse rate, time constant, and loop gain to one of seven levels according to the positioning distance is used. (The settings are common to all axes.)

NOTE

1 The settings must satisfy the relationship D1 < D2 < D3 < D4 < D5 < D6.

2 Up to seven levels can be used for adjustment. When using four levels, for example, set D4 to 99999999.

3 For diameter programming axes, set a diameter. For example, assume that 10.000 mm is set in a parameter for diameter programming axes. Then, when the amount of travel has reached 10.000 mm, adjustment is performed.

6161	Level 1 rapid traverse rate
6162	Level 2 rapid traverse rate
6163	Level 3 rapid traverse rate
6164	Level 4 rapid traverse rate
6165	Level 5 rapid traverse rate
6166	Level 6 rapid traverse rate
6167	Level 7 rapid traverse rate

[Data type] 2–word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	30 to 240000
Inch machine	0.1 inch/min	0 to 96000

The rapid traverse rate for each axis is set.

6171	Level 1 rapid traverse time constant
6172	Level 2 rapid traverse time constant
6173	Level 3 rapid traverse time constant
6174	Level 4 rapid traverse time constant
6175	Level 5 rapid traverse time constant
6176	Level 6 rapid traverse time constant
6177	Level 7 rapid traverse time constant

[Data type] Word axis

[Unit of data] ms

[Valid data range] 8 to 4000

The rapid traverse time constant for each axis is set.

6181	Level 1 servo loop gain
6182	Level 2 servo loop gain
6183	Level 3 servo loop gain
6184	Level 4 servo loop gain
6185	Level 5 servo loop gain
6186	Level 6 servo loop gain
6187	Level 7 servo loop gain

[Data type] Word axis

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

[Valid data range] 1 to 9999

The servo loop gain for each axis is set.

Positioning distances and corresponding parameter Nos.

Level	Positioning distance d	Rapid traverse rate	Rapid traverse time constant	Servo loop gain
1	0<d≦D1	6161	6171	6181
2	D1<d≦D2	6162	6172	6182
3	D2<d≦D3	6163	6173	6183
4	D3<d≦D4	6164	6174	6184
5	D4<d≦D5	6165	6175	6185
6	D5<d≦D6	6166	6176	6186
7	D6<d	6167	6177	6187

7.1.16 Simple High–precision Contour Control (M series)

General

By taking full advantages of high–precision contour control using a RISC processor, this function enables high–speed high–precision machining without the need for special hardware.

This function is designed for high–speed high–precision machining. The use of this function enables the suppression of the delay in the servo system and the delay in acceleration/deceleration which increases as the feedrate increases, therefore reducing the machining profile error.

This function enables look–ahead linear acceleration/deceleration before the interpolation of up to 15 blocks. This results in smooth acceleration/deceleration over many blocks, as well as high–speed machining.

In simple high–precision contour control mode, the following functions are effective:

(1) Look–ahead linear acceleration/deceleration before interpolation (look–ahead control of up to 15 blocks)
(2) Automatic corner deceleration
(3) Feedrate clamping by acceleration
(4) Feedrate clamping by arc radius
(5) Block overlap (up to 5 blocks)
(6) Look–ahead feed–forward

For details of the above functions, see the relevant explanations.

Specifications

Axis control

Name	Description
Controlled axes	3 to 8
Simultaneously controlled axes	Up to 6
Axis name	Basic three axes: Always X, Y, and Z Other axes: U, V, W, A, B, or C
Least input increment	0.001 mm, 0.001 deg, 0.0001 inch
Input increment 1/10	0.0001 mm, 0.0001 deg, 0.00001 inch

Interpolation function

○ **: Can be programmed**

× **: Cannot be programmed**

Name	Description
Positioning (G00)	○ (Positioning of linear interpolation type)
Single direction positioning (G60)	×
Exact stop (G09)	○
Exact stop mode (G61)	○
Tapping mode (G63)	×
Automatic corner override (G62)	×
Linear interpolation (G01)	○
Circular interpolation (G02, G03)	○ (Multiple quadrants allowed)
Helical interpolation (G02, G03)	○ (Circular interpolation + Up to four axes for linear interpolation) When the helical interpolation function is selected, up to two axes for linear interpolation can be specified. When the helical interpolation B function is selected, up to four axes for linear interpolation can be specified. A desired feedrate must be specified by also taking movement along the helical axis into consideration.
Spiral interpolation/Conical interpolation (G02, G03)	○

Name	Description
Involute interpolation (G02.2, G03.2)	×
Exponential interpolation (G02.3, G03.3)	×
Dwell (G04)	○ (For a specified number of seconds or revolutions) To specify a number of revolutions for the dwell, the thread cutting/synchronous feed function must be selected.
Polar coordinate interpolation (G12.1, G13.1)	×
Cylindrical interpolation (G07.1)	×
Thread cutting/synchronous feed (G33)	×
Skip function (G31)	○ *
High–speed skip function (G31)	○ *
Multistage skip function (G31 Px)	○ *
Reference position return (G28)	○ When the reference position is not established, P/S alarm No. 090 is issued. *
Reference position return check (G27)	○ *
2nd, 3rd, and 4th reference position return (G30)	○ *
Floating reference position return (G30.1)	○ *
Canned cycle (G73 to G89)	○ *
Rigid tapping	×
Return to initial point in canned cycle (G98)/Return to R point in canned cycle (G99)	○ *
Normal direction control (G41.1, G42.1)	×
Continuous dressing	×
In–feed control (G161)	×
Index table indexing	×
High–speed cycle machining (G05)	×
Absolute command (G90)/ Incremental command (G91)	○

Feed functions

○ : **Can be programmed**
× : **Cannot be programmed**

Name	Description
Rapid traverse rate	Up to 240 m/min (0.01 mm)
	Up to 100 m/min (0.0001 mm)
Rapid traverse rate override	F0, 25, 50, 100 %
Rapid traverse rate override in units of 1%	0% to 100 %
Feed per minute (G94)	○
Feed per rotation (G95)	×
Rapid traverse bell–shaped acceleration/deceleration	×
Cutting feed linear acceleration/deceleration before interpolation	○ (look–ahead control of up to 15 blocks)
Feedrate override	0% to 254%
Second feedrate override	×
Feed by F command with one digit	×
Inverse time feed (G93)	×
External deceleration	○

Tool compensation functions

○ : **Can be programmed**
× : **Cannot be programmed**

Name	Description
Cutter compensation C (G40, G41, G42)	○
Tool length compensation (G43, G44, G49)	○

Program input

○ : **Can be programmed**
× : **Cannot be programmed**

Name	Description
Plane selection (G17, G18, G19)	○
Local coordinate system (G52)	○ *
Workpiece coordinate system (G54 to G59) (G54.1Pxx)	○ *
Workpiece coordinate system (G92)	○ *
Workpiece coordinate system preset (G92.1)	○ *
Interrupt–type custom macro	×

Others

○ : Can be programmed

× : Cannot be programmed

Name	Description
Cycle start/Feed hold	○
Dry run	○
Single block	○
Interlock	○
Machine lock	○ When an axis machine lock signal (MLK1 to MLK8) is turned on or off, acceleration/deceleration is not performed on the axis held under machine lock.
Control–in/control–out command ( )	○
Optional block skip command (/n: n is a number)	○
Miscellaneous function (Mxxxx)	○ Only the function code signal and function strobe signal are output.
Spindle function (Sxxxx)	○
Tool function (Txxxx)	○ Only the function code signal and function strobe signal are output.
Second auxiliary function (Bxxxx)	○ Only the function code signal and function strobe signal are output.
Simple synchronous control	○ Synchronous control cannot be enabled or disabled.
Program restart	×
Retrace function	×
Tool life management	×
Macro executor (execution macro)	×
MDI operation	× When G05.1 Q1 is specified in MDI mode, P/S alarm No. 5113 is issued. The operation mode cannot be switched to MDI mode in simple high–precision contour control mode.
Manual intervention	× Upon a restart after manual intervention, the position at which manual intervention occurred must be restored. If the position is not restored, P/S alarm No. 5114 is issued.

Those functions marked with an asterisk (*) do not perform look–ahead control of multiple blocks.

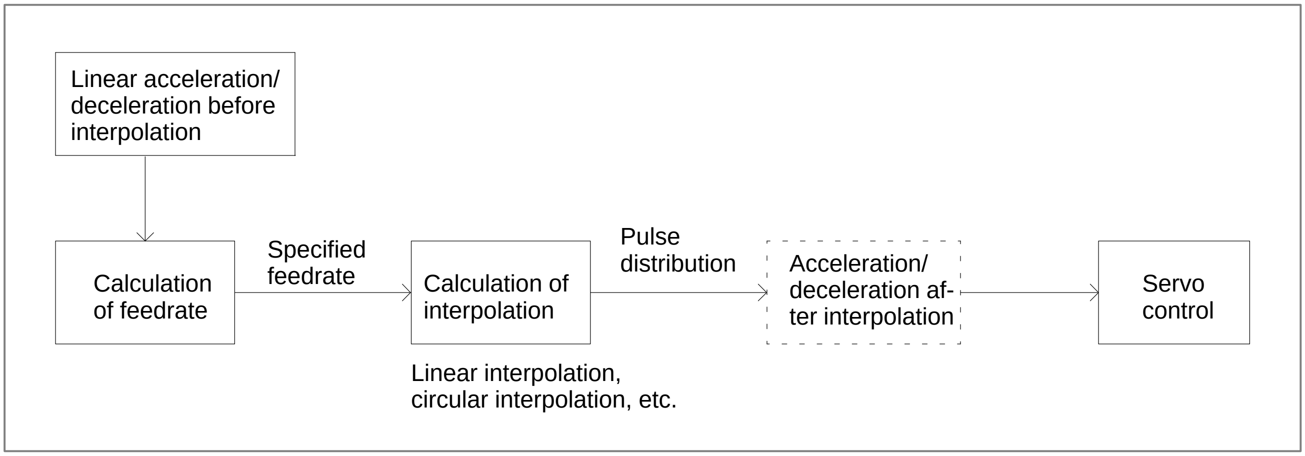
Conditions for entering simple high–precision contour control mode

Before G05.1 Q1, the following modal codes must be specified. If this condition is not satisfied, P/S alarm No. 5111 will be issued.

G code	Description
G00 G01 G02 G03	Positioning Linear interpolation Circular interpolation (CW) Circular interpolation (CCW)
G13.1	Polar coordinate interpolation cancel mode
G15	Polar coordinate command cancel
G25	Spindle speed fluctuation detection off
G40	Cutter compensation cancel
G40.1	Normal direction control cancel mode
G49	Tool length compensation cancel
G50	Scaling cancel
G50.1	Programmable mirror image cancel
G64	Cutting mode
G67	Macro modal call cancel
G69	Coordinate rotation cancel
G80	Canned cycle cancel
G94	Feed per minute
G97	Constant surface speed control cancel
G160	In–feed control function cancel

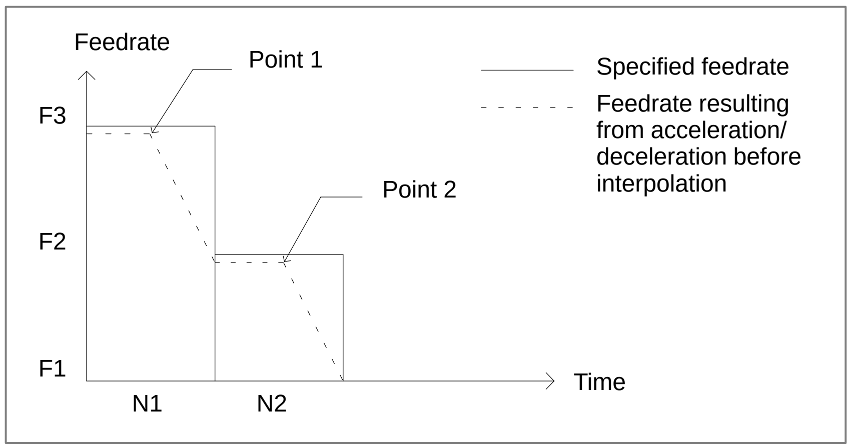
Advanced preview linear acceleration/deceleration before interpolation

For a cutting feed command in feed per minute mode, advanced preview control of up to 15 blocks can be performed, and linear acceleration/deceleration can be performed for a programmed feedrate (before interpolation). When acceleration/deceleration is performed after interpolation, acceleration/deceleration is performed for the data resulting from interpolation, which changes the interpolation data. For acceleration/deceleration before interpolation, on the other hand, acceleration/deceleration is performed for the feedrate data before interpolation, so that the interpolation data is not changed by acceleration/deceleration. Therefore, interpolation data can always be aligned with a specified line or curve to eliminate any machining profile errors caused by a delay in acceleration/deceleration.



(Example of deceleration)

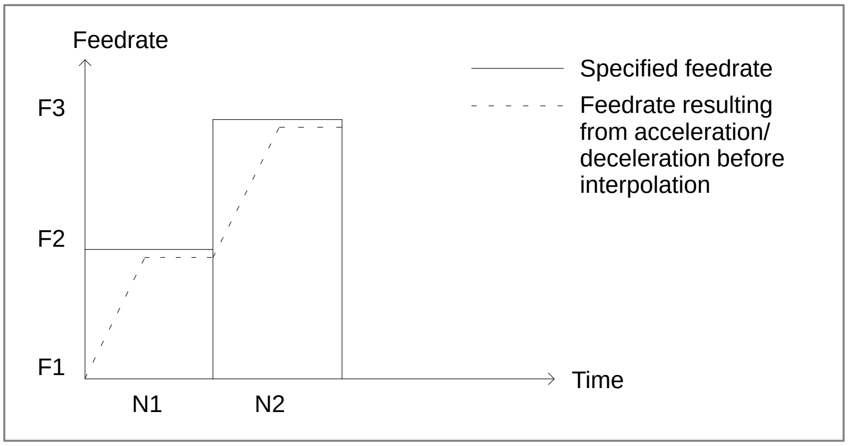
To execute a block at a specified feedrate, the feedrate is reduced from that of the previous block.



To reduce the feedrate from F3 to F2, deceleration must be started at Point 1.
To decrease the feedrate from F2 to F1, deceleration must be started at Point 2.
Since advanced preview control for up to 15 blocks is performed, deceleration over multiple blocks can be performed.

(Example of acceleration)

To execute a block at a specified feedrate, the feedrate is increased.



Automatic corner deceleration

When there are axes for which the difference in feedrate between blocks exceeds the allowable feedrate difference set in parameter 1783, the automatic corner deceleration function calculates the feedrate at a corner and performs deceleration so that the calculated feedrate is attained at the junction of the blocks. The feedrate at the corner is calculated as explained below, where the ratio of the difference in the actual feedrate to the allowable feedrate difference is obtained for each axis, after which the largest of these ratios is used as a reference.

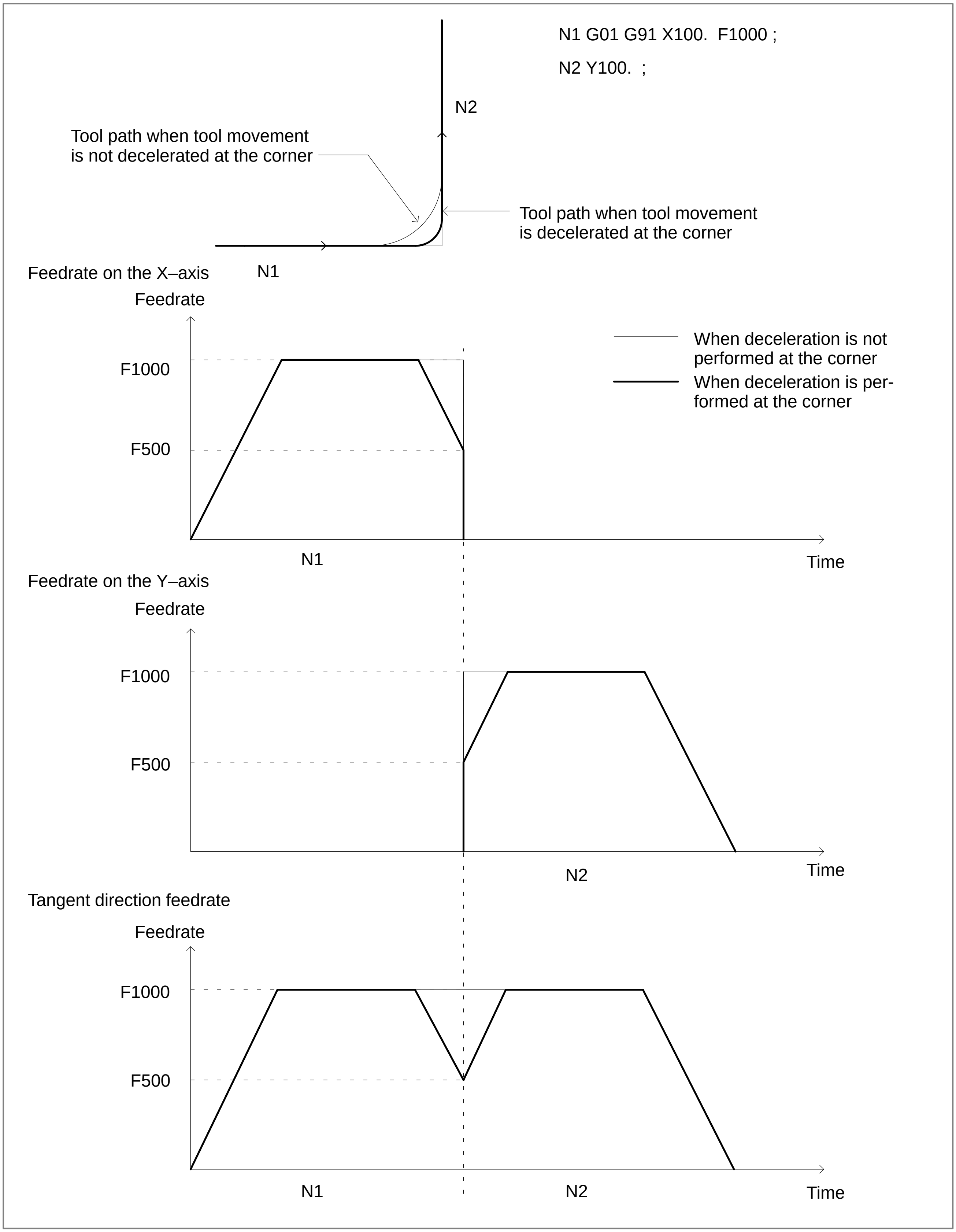
The change in feedrate on each axis (Vx, Vy, ...) when the tool moves at a specified feedrate F is compared with the value set in parameter 1783 (Vprm–x, Vprm–y, ...). If the change in the feedrate for any axis exceeds the parameter–set value, the following is obtained first:

$$\mathrm{Rmax} = \max\left[\frac{Vx}{Vprm\text{–}x},\ \frac{Vy}{Vprm\text{–}y},\ \ldots\right]$$

Then, the feedrate (Fc) is obtained from the following expression, after which corner deceleration is performed:

$$Fc = F \times \frac{1}{\mathrm{Rmax}}$$

For example, suppose that the direction of the tool movement changes through 90 degrees from X–axis movement to Y–axis movement, and also suppose that the specified feedrate is 1000 mm/min, and that the allowable feedrate difference (parameter 1783) is set to 500 mm/min. Then, deceleration is performed as shown in the figure below.


N1 G01 G91 X100. F1000 ;
N2 Y100. ;
N2
Tool path when tool movement is not decelerated at the corner
Tool path when tool movement is decelerated at the corner
N1
Feedrate on the X–axis
Feedrate
F1000
F500
N1
Time
When deceleration is not performed at the corner
When deceleration is performed at the corner
Feedrate on the Y–axis
Feedrate
F1000
F500
N2
Time
Tangent direction feedrate
Feedrate
F1000
F500
N1
N2
Time

Feedrate clamping by acceleration

When a curve is made up of a series of short straight lines as shown in the figure below, the difference in feedrate on each axis at each corner is not large. Therefore, deceleration by feedrate difference is not effective. Small feedrate differences occur successively, however, so a large acceleration will be generated on each axis as a whole.

In such cases, deceleration is performed to reduce the machining error and the stress imposed on the machine, both of which are caused by excessive acceleration. The target feedrate to be reached by deceleration is set so that the acceleration on each axis, as obtained from the expression below, does not exceed the allowable acceleration set for all axes.

The allowable acceleration is set using the maximum cutting feedrate (parameter 1432) and the time required to reach the maximum cutting feedrate (parameter 1785).

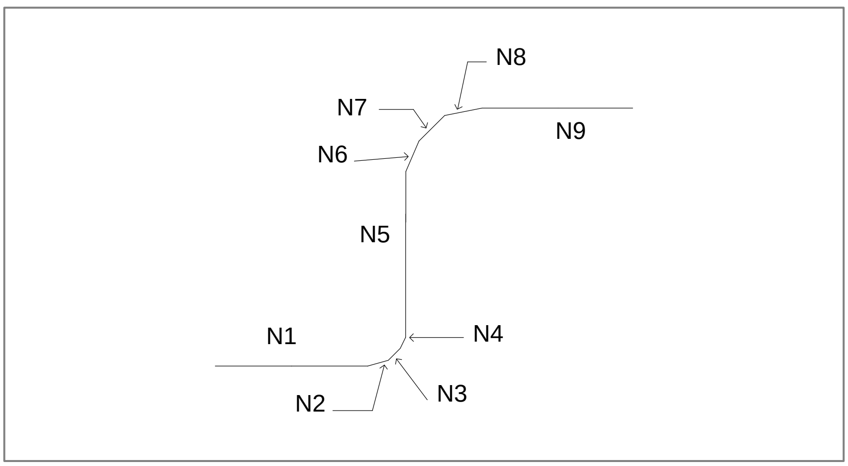
Acceleration on an axis =

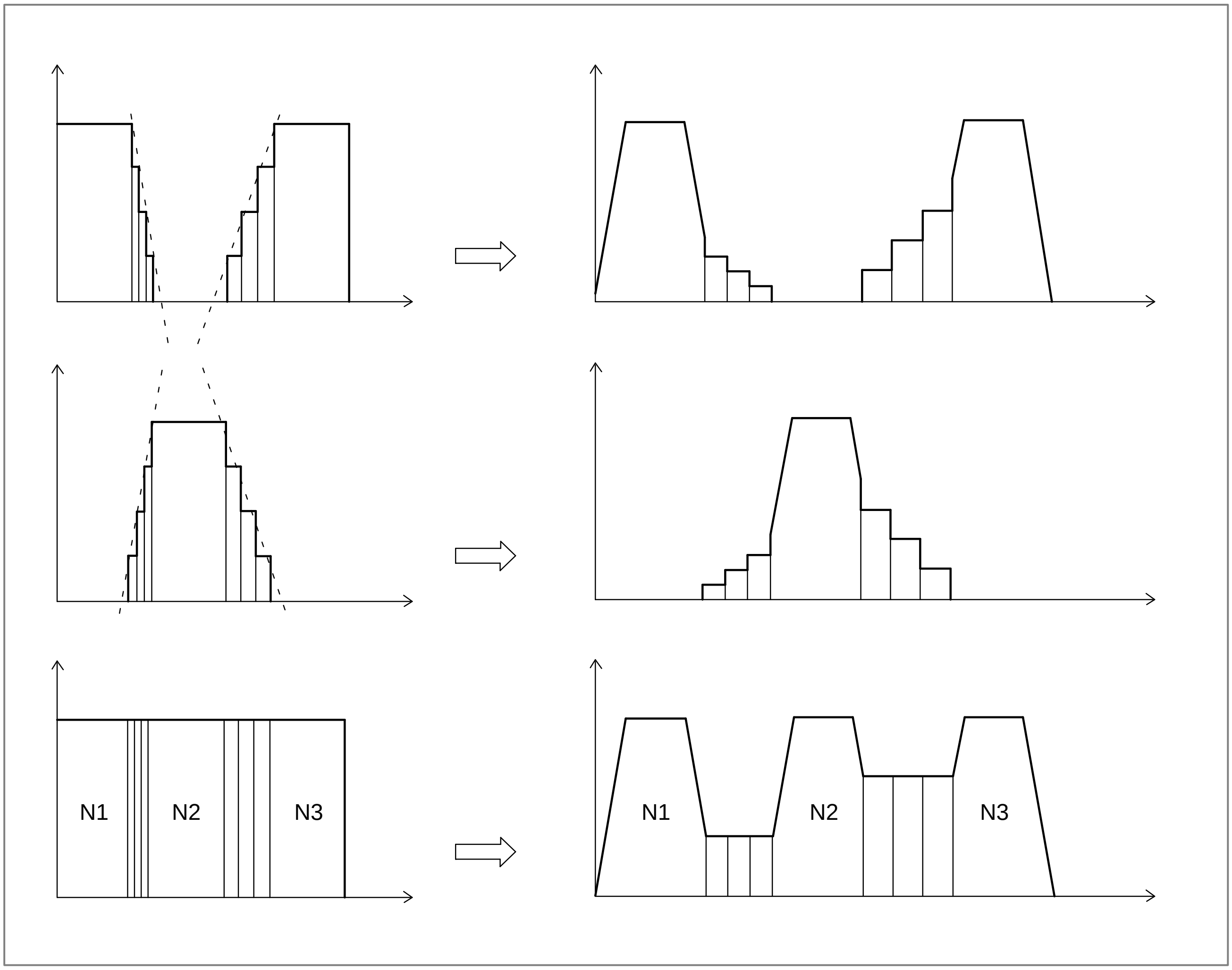
$$\frac{\text{Difference in feedrate on the axis at a corner}}{\max\left(\dfrac{\text{Amount of travel in the previous block}}{F},\ \dfrac{\text{Amount of travel in the next block}}{F}\right)}$$

A target feedrate to be reached by deceleration is obtained for each corner. The actual feedrate is the feedrate to be reached by deceleration, as obtained at the start point of a block, or the feedrate to be reached by deceleration as obtained at the end point of the block, whichever is the lower.

Example:

In the following example, too great an acceleration/deceleration (the inclination of the dotted lines in the feedrate graphs) is observed between N2 and N4 and between N6 and N8, so deceleration is performed in these areas.





Feedrate clamping by arc radius

This function can suppress acceleration in an arc machining block to an allowable level by clamping the feedrate. Based on the arc radius R and the maximum allowable feedrate V for that arc radius (which are set in parameters), this function calculates the maximum allowable feedrate v for an arc with a programmed radius r, as shown below. If the specified feedrate exceeds feedrate V, the feedrate can be clamped to feedrate v automatically.

$$\text{Maximum allowable acceleration} = \frac{V^2}{R}$$

R : Arc radius, V : Feedrate for arc radius R

Then, the maximum allowable feedrate v for an arc with radius r is obtained from the following expression:

$$v = \sqrt{r/R} \times V$$

As the specified arc radius becomes smaller, the maximum allowable feedrate v falls. To prevent the maximum allowable feedrate from becoming too small, the lower limit for feedrate clamping based on the arc radius can be set in parameter 1732. If the maximum allowable feedrate is less than the value set in parameter 1732, the parameter–set value can be used as the maximum allowable feedrate v.

Rapid traverse

In rapid traverse, acceleration/deceleration is performed using linear acceleration/deceleration before interpolation, and tool movement is performed by positioning based on linear interpolation. The tool movement feedrate is obtained from the rapid traverse rate for each axis which is set in parameter 1420. The acceleration in acceleration/deceleration before interpolation is obtained from the rapid traverse rate for each axis which is set in parameter 1420 and the linear acceleration/deceleration time constant for each axis in rapid traverse which is set in parameter 1620. They are obtained as follows:

(1) Tool movement feedrate

For each axis along which the tool moves, calculation is performed using the following expression. Then, from among the calculated values, the smallest value is used as the tool movement feedrate.

$$\text{Rapid traverse rate on an axis (parameter 1420)} \times \frac{\text{Amount of travel for a block}}{\text{Amount of travel along an axis}}$$

(2) Acceleration in linear acceleration/deceleration before interpolation

For each axis along which the tool moves, calculation is performed using the following expression. Then, from among the calculated values, the smallest value is used as the acceleration in linear acceleration/deceleration before interpolation during movement.

$$\frac{\text{Rapid traverse rate on an axis (parameter 1420)}}{\text{Time constant for an axis (parameter 1620)}} \times \frac{\text{Amount of travel for a block}}{\text{Amount of travel along an axis}}$$

> **NOTE**
> Any overlap of rapid traverse blocks is invalid.

Signal

Simple high–precision contour control mode signal SHPC <F062#0>

[Classification] Output signal

[Function] This signal indicates that the system is in simple high–precision contour control mode.

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

- The system is in simple high–precision contour control mode.

The signal is set to 0 when:

- The system is in other than simple high–precision contour control mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F062								SHPC

Parameter

- **Parameters related to linear acceleration/ deceleration before interpolation**

1770	Maximum machining feedrate during linear acceleration/deceleration before interpolation

[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.01 inch/min	6 to 9600	6 to 4800

This parameter sets the maximum machining feedrate for linear acceleration/deceleration before interpolation. (Parameter 1 for setting the acceleration in linear acceleration/deceleration before interpolation)

1771	Time required to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation (time constant)

[Data type] Word

[Unit of data] 1 ms

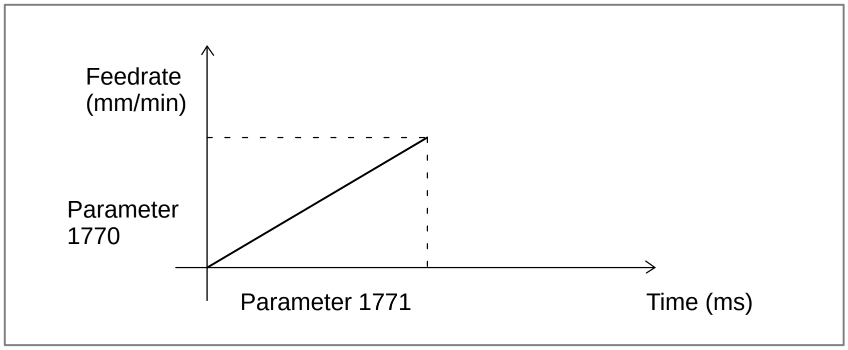
[Valid data range] 0 to 4000

This parameter sets the time (time constant) required to reach the feedrate set in parameter 1. (Parameter 2 for setting the acceleration in linear acceleration/deceleration before interpolation)

NOTE

1 When parameter 1770 or 1771 is set to 0, linear acceleration/deceleration before interpolation is not performed.

2 Set these parameters so that parameter 1770/parameter 1771 = 5 or more.



1784	Speed when an overtravel alarm is issued during linear acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.01 inch/min	6 to 6000	6 to 4800

This parameter sets the speed to be reached when an overtravel alarm is issued during linear acceleration/deceleration during interpolation. If an overtravel alarm is issued during linear acceleration/deceleration before interpolation, the movement is decelerated and halted after the issue of the alarm. Therefore, the tool overruns by an amount equal to the distance traveled during deceleration. The overrun varies depending on the feedrate observed when the overtravel alarm is issued. The overrun can be reduced by performing deceleration to the speed set in parameter 1784 in advance when an overtravel alarm is issued. In this case, deceleration is performed so that the feedrate at the instant when the overtravel alarm is issued does not exceed the parameter–set speed. So, deceleration may be completed earlier. Upon the completion of deceleration, the feedrate is set to the parameter–set speed.

NOTE
This parameter is invalid for rapid traverse blocks.

If the following condition is satisfied, deceleration is performed:

Distance to the stored stroke limit on an axis	<	Distance required for decelerating the current speed (tangent direction feedrate) to the speed set in parameter 1784

The overrun is expressed as follows:

$$\text{Overrun distance} \leqq \frac{\left[\mathrm{FIX}\left(\frac{F_{OT}}{F}\times\frac{T}{8}\right)+1.5\right]^{2}}{1875}\times\frac{F}{T}$$

F : Maximum machining feedrate during linear acceleration/deceleration before interpolation (parameter 1770)

T : Time required to reach the maximum machining feedrate during linear acceleration/deceleration before interpolation (parameter 1771)

F_{OT} : Speed when an overtravel alarm is issued during linear acceleration/deceleration before interpolation (parameter 1784)

FIX : Any fractional part is truncated.

NOTE
1 When 0 is set, the above control is not performed.
2 When stroke check is invalid, the above control is also invalid.
3 The above control is valid only for stored stroke check 1.

- **Parameter related to automatic corner deceleration**

1783	Allowable feedrate difference for each axis in the corner deceleration function by feedrate difference (for acceleration/deceleration before interpolation)

[Data type] Word axis

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

This parameter sets the allowable difference in feedrate on each axis in the automatic corner deceleration function by the difference in feedrate when liner acceleration/deceleration before interpolation is used.

- **Parameter related to feedrate clamping by acceleration**

1785	Parameter for determining the allowable acceleration in feedrate clamping by acceleration

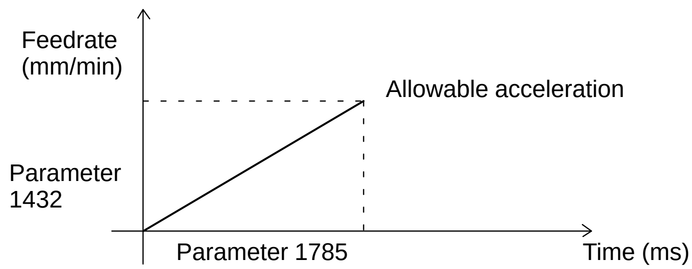
[Data type] Word axis

[Unit of data] 1 ms

[Valid data range] 0 to 32767

This parameter is used to set the time required to reach the maximum cutting feedrate and determine the allowable acceleration when feedrate clamping by acceleration is performed.

The allowable acceleration is determined from the maximum cutting feedrate and the data set in this parameter. Parameter 1432 (maximum cutting feedrate in simple high–precision contour control mode) is used as the maximum cutting feedrate.



Parameters related to feedrate clamping by arc radius

1731	Arc radius for the upper limit imposed on feedrate

[Data type] 2–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1000 to 99999999

This parameter sets the arc radius for the upper limit imposed on the feedrate set in parameter 1730.

1730	Upper limit imposed on feedrate for arc radius R

[Data type] Word

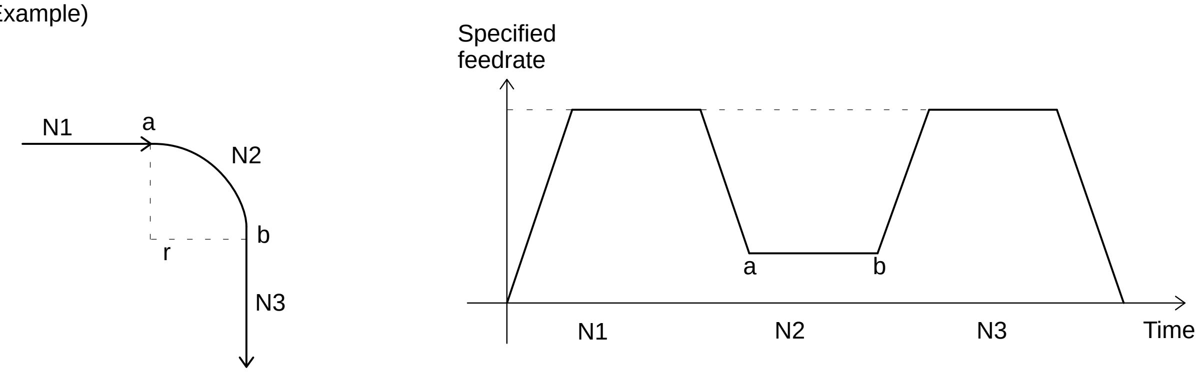
[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 to 15000	0 to 12000
Inch machine	0.1 inch/min	8 to 6000	0 to 4800

This parameter sets the upper limit imposed on the feed rate for the arc radius set in parameter 1731.

(Example)



1732	Lower limit RVmin for feedrate clamping by arc radius

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	8 to 15000	0 to 12000
Inch machine	0.1 inch/min	8 to 6000	0 to 4800

When the function for clamping the feedrate by arc radius is used, the upper limit imposed on the feedrate falls with the arc radius. If the upper limit imposed on the feedrate is less than the lower limit imposed on the feedrate clamping RVmin, the upper limit imposed on the feedrate is set as RVmin.

- **Other parameters**

1422	Upper limit imposed on cutting feedrate in simple high–precision contour control

[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 240000	6 to 100000
Inch machine	0.1 inch/min	6 to 96000	6 to 4800

This parameter is used to set the upper limit on the cutting feedrate in simple high–precision contour control mode.

1432	Maximum cutting feedrate in simple high–precision contour control mode (for each axis)

[Data type] 2–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min	0 to 96000	0 to 48000
Rotation axis	1 deg/min	0 to 240000	0 to 100000

This parameter sets the maximum cutting feedrate for each axis in simple high–precision contour control mode.

	#7	#6	#5	#4	#3	#2	#1	#0
7050		MI1	MI0					

[Data type] Bit

MI1, MI0 Set the following values

	MI1	MI0
Setting	0	1

	#7	#6	#5	#4	#3	#2	#1	#0
7052								NMI

[Data type] Bit axis

For the PMC–controlled axes and Cs axis, set 1.

- **Parameter numbers in standard mode, advanced preview control mode, and simple high–precision contour control mode**

(Parameters related to linear acceleration/deceleration before interpolation)

Parameter	Parameter No.		
	Standard mode	Advanced preview control	Simple high–precision contour control
Acceleration/deceleration type (type A/B)	FWB/1602#0		None
Parameter 1 for setting acceleration	1630	1770	
Parameter 2 for setting acceleration	1631	1771	
Speed when overtravel alarm is issued	1784		

(Parameters related to automatic corner deceleration)

Parameter	Parameter No.		
	Standard mode	Advanced preview control	Simple high–precision contour control
Method for determining automatic corner deceleration (angle/feedrate difference)	CSD/1602#4		None
Lower limit imposed on feedrate (control based on angle)	1778	1777	None
Angle to be determined (control based on angle)	1740	1779	None
Allowable feedrate difference for all axes (control based on feedrate difference)	1780		None
Allowable feedrate difference for each axis (control based on feedrate difference)	1783		

(Parameters related to feedrate clamping by arc radius)

Parameter	Parameter No.		
	Standard mode	Advanced preview control	Simple high–precision contour control
Arc radius for the upper limit of feedrate	1731		
Upper limit imposed on feedrate for arc radius R	1730		
Lower limit imposed on clamp feedrate	1732		

(Other parameters)

Parameter	Parameter No.		
	Standard mode	Advanced preview control	Simple high–precision contour control
Precision of radius error in circular interpolation	PCIR1/3403#0		None
Maximum cutting feedrate (for all axes)	1422	1431	1422
Maximum cutting feedrate (for each axis)	1430	1432	

Alarm and message

Number	Message	Description
5110	IMPROPER G–CODE (G05.1 G1 MODE) (M series)	An invalid G code is specified in simple high–precision contour control mode. A command is specified for the index table indexing axis in simple high–precision contour control mode.
5111	IMPROPER MODAL G–CODE (G05.1 G1) (M series)	When simple high–precision contour control mode is specified, a G code that cannot be used is placed in the model state.
5112	G08 CAN NOT BE COMMANDED (G05.1 G1) (M series)	An advanced preview control command (G08) is specified in simple high–precision contour control mode.
5113	CAN NOT ERROR IN MDI MODE (G05.1) (M series)	Simple high–precision contour control (G05.1) is specified in MDI mode.
5114	NOT STOP POSITION (G05.1 Q1) (M series)	Upon a restart after manual intervention, the coordinates at which manual intervention was performed are not restored.
5156	SPL: ERROR (M series)	The controlled axis selection signal (PMC axis control) changes in simple high–precision contour control mode. The simple synchronous axis selection signal changes in simple high–precision contour control mode.

Caution

CAUTION

1 When the total distance for the blocks under advanced preview control is equal to or less than the deceleration distance from the current feedrate, deceleration starts. When advanced preview control has proceeded upon the completion of deceleration, and the total distance for the blocks increases,acceleration starts. Especially, when a series of blocks containing very small amounts of travel are specified, deceleration and acceleration may be alternated, which prevents the feedrate from becoming constant. In such a case, specify a lower feedrate.

2 When the dry run signal is inverted from 0 to 1 or from 1 to 0 during movement along an axis, the speed of movement is increased or reduced to a specified speed without first being reduced to zero.

3 When a no–movement block or a one–shot G code such as G04 is encountered in simple high–precision contour control mode, the movement is decelerated and halted in the preceding block.

Note

NOTE

1 As acceleration after interpolation, use linear or bell–shaped acceleration. Exponential acceleration/deceleration cannot be used.

2 Upon switching to simple high–precision contour control mode, all manual handle interrupts are ignored.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.19.6	Simple high–precision contour control

7.1.17 High–speed Linear Interpolation (M series)

General

The high–speed linear interpolation function processes a move command related to a controlled axis not by ordinary linear interpolation but by high–speed linear interpolation. This function enables the high–speed execution of an NC program including a series of very small amounts of travel.

- **High–speed linear interpolation mode**

The high–speed linear interpolation start command G05 P2 places the system in high–speed linear interpolation mode, in which high–speed linear interpolation is executed. The high–speed linear interpolation end command G05P0 places the system in standard NC program operation mode.

At power–up or in the NC reset state, the system enters standard NC program operation mode.

After the end of high–speed linear interpolation mode, the system enters standard NC program operation mode.

- **Cutting feedrate**

Specify a cutting feedrate in high–speed linear interpolation mode. If no cutting feedrate is specified, the modal F value is assumed.

Maximum feedrate	Interpolation period: 8 ms		Interpolation period: 4 ms	
(IS–B mm input)	122848	mm/min	245696	mm/min
(IS–B inch input)	12284.8	inch/min	24569.6	inch/mim
(IS–C mm input)	12284	mm/min	24569	mm/min
(IS–C inch input)	1228.48	inch/min	2456.96	inch/min

(Maximum feedrate) =
$122{,}848 \times 8$/(interpolation period) (IS–B, metric input)

Minimum feedrate	Interpolation period: 8 ms		Interpolation period: 4 ms	
(IS–B mm input)	4	mm/min	8	mm/min
(IS–B inch input)	0.38	inch/min	0.76	inch/mim
(IS–C mm input)	4	mm/min	8	mm/min
(IS–C inch input)	0.38	inch/min	0.76	inch/min

(Minimum feedrate) = 4×8/(interpolation period) (IS–B, metric input)

- **Interpolation period**

In high–speed linear interpolation mode, the NC interpolation period can be changed. As the interpolation period decreases, the machining speed and precision increase.

IT2, IT1, and IT0 bits (bits 6, 5, and 4 of parameter 7501)

IT2	IT1	IT0	Interpolation period
0	0	0	8 ms in high–speed linear interpolation mode
0	1	0	4 ms in high–speed linear interpolation mode
0	0	1	2 ms in high–speed linear interpolation mode
0	1	1	1 ms in high–speed linear interpolation mode
1	1	1	0.5 ms in high–speed linear interpolation mode

- **Acceleration/ deceleration processing in high–speed linear interpolation mode**

In high–speed linear interpolation mode, acceleration/deceleration before interpolation is not effective. Acceleration/deceleration after interpolation is enabled or disabled by setting SUP (bit 0 of parameter No.7502). When the interpolation period is set to 4 ms or less, the time constant for linear acceleration/deceleration after interpolation and bell–shaped acceleration/deceleration after interpolation is restricted as follows:

Time constant for linear acceleration/deceleration after interpolation and bell–shaped acceleration/deceleration after interpolation (parameter No.1628)

Interpolation period 8 ms: Maximum setting = 512 ms
Interpolation period 4 ms: Maximum setting = 256 ms
Interpolation period 2 ms: Maximum setting = 128 ms
Interpolation period 1 ms: Maximum setting = 064 ms
Interpolation period 0.5 ms: Maximum setting = 032 ms

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7501		IT2	IT1	IT0				

[Data type] Bit

IT2	IT1	IT0	
0	0	0	: The interpolation period in high–speed linear interpolation mode is 8 ms.
0	1	0	: The interpolation period in high–speed linear interpolation mode is 4 ms.
0	0	1	: The interpolation period in high–speed linear interpolation mode is 2 ms.
0	1	1	: The interpolation period in high–speed linear interpolation mode is 1 ms.
1	1	1	: The interpolation period in high–speed linear interpolation mode is 0.5 ms.

	#7	#6	#5	#4	#3	#2	#1	#0
7502								SUP

[Data type] Bit

SUP 0 : Acceleration/deceleration processing is disabled in high–speed linear interpolation mode.

1 : Acceleration/deceleration after interpolation is enabled in high–speed linear interpolation mode.

7510	Number of controlled axes in high–speed linear interpolation mode

[Data type] Byte

[Unit of data] Number of controlled axes

[Valid data range] 1 to 4

This parameter sets the number of axes controlled in high–speed linear interpolation mode.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.19.8	High–speed linear interpolation

7.2 ACCELERATION/ DECELERATION CONTROL

7.2.1 Automatic Acceleration/ Deceleration

General

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 7.2.1).

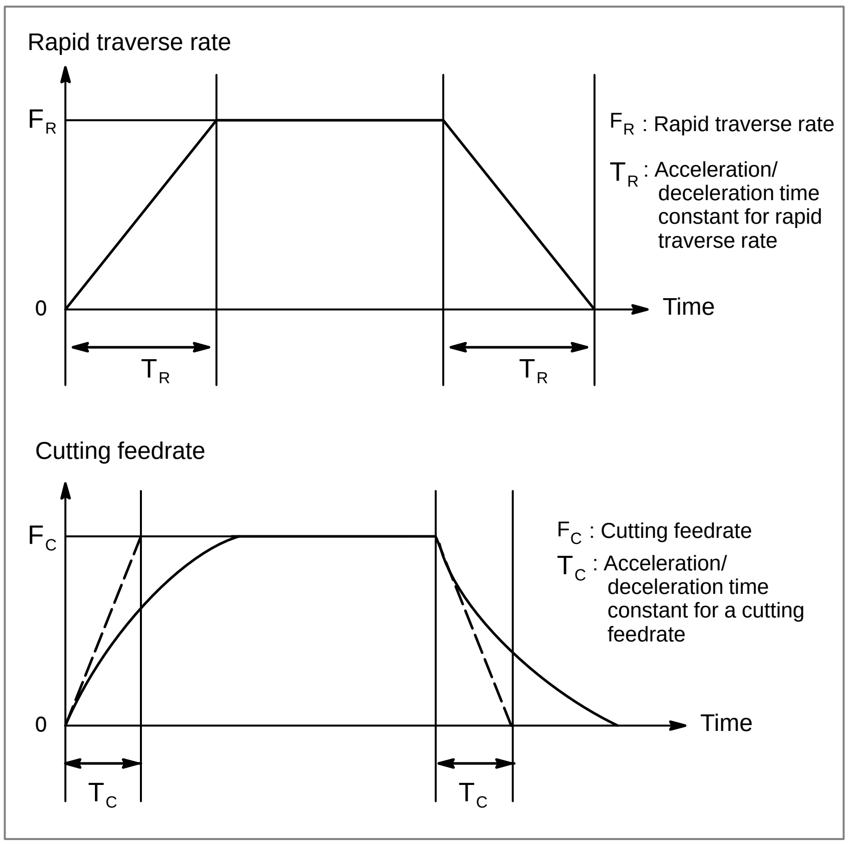


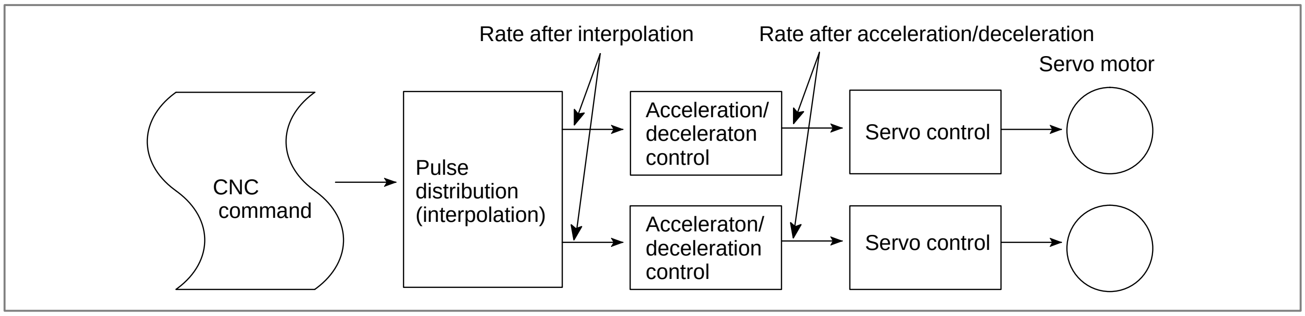
Fig. 7.2.1 Automatic acceleration/deceleration (example)

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so 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 parameter 1620)

Cutting feed: Exponential acceleration/deceleration (time constant per axis is set by parameter 1622)

Jog feed : Exponential acceleration/deceleration (time constant per axis is set by parameter 1624)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

NOTE
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.
To use bell–shaped acceleration/deceleration after interpolation, set this parameter to 0 and select the acceleration/deceleration using CTBx, bit 1 of parameter No. 1610.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

CTBx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration or linear acceleration/deceleration after interpolation is applied (depending on the setting in CTLx, bit 0 of parameter No. 1610).
1 : Bell–shaped acceleration/deceleration after interpolation is applied.

> **NOTE**
> This parameter is effective only when the function of bell–shaped acceleration/deceleration after interpolation in cutting feed is provided. If the function is not provided, the setting in CTLx, bit 0 of parameter No. 1610, determines the type of acceleration/deceleration irrespective of the setting in this parameter.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.

1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used for cutting feed).

1620	Time constant used for linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse. 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 T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.

(2) When the function is not provided, specify a time constant used in linear acceleration/deceleration.

> **NOTE**
> When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used in linear acceleration/deceleration in rapid traverse.

1622	Time constant of exponential acceleration/deceleration or linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration / deceleration)
0 to 512 (For linear acceleration / deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration or linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1623	FL rate of exponential acceleration/deceleration in cutting feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in cutting feed for each axis. Except for special applications, this parameter must be set to 0 for all axes. If a value other than 0 is specified, proper straight lines and arcs cannot be obtained.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/ deceleration after interpolation or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration / deceleration)
0 to 512 (For linear acceleration / deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

1625	FL rate of exponential acceleration/deceleration in jog feed for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in jog feed for each axis.

1626	Time constant of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 4000

Set the time constant used for exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

1627	FL rate of exponential acceleration/deceleration in the thread cutting cycle for each axis

[Data type] Word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800
Rotaion axis	1 deg/min	6 – 15000	6 – 12000

Set the lower limit (FL rate) of exponential acceleration/deceleration in the thread cutting cycle (G76, G78 (G92 in G code system A)) for each axis.

1722	Rapid traverse feedrate reduction ratio for overlapping rapid traverse blocks

[Data type] Byte axis

[Unit of data] %

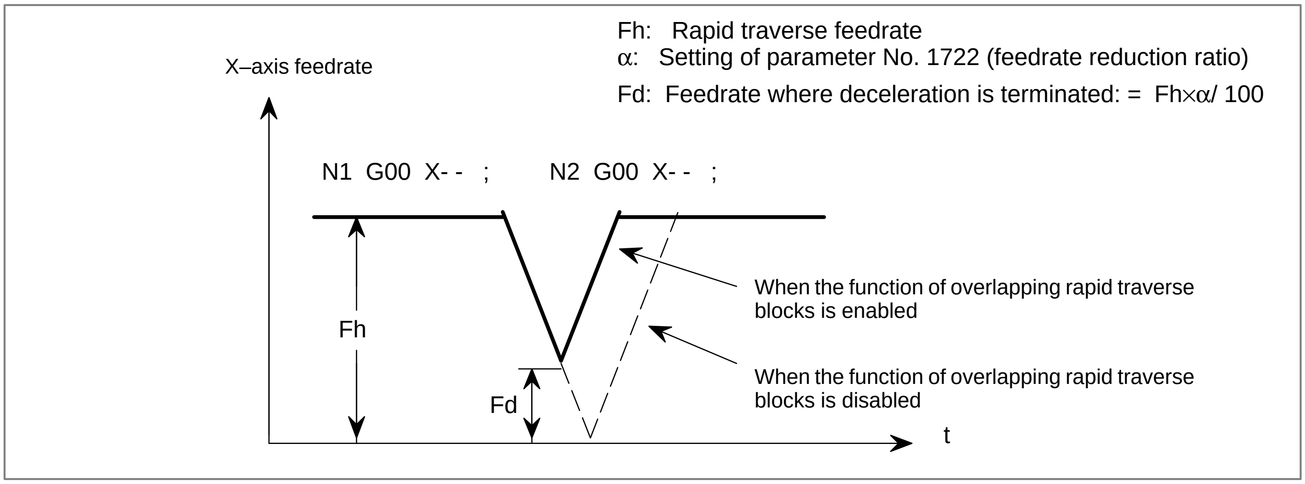
[Valid data range] 0 to 100

This parameter is used when rapid traverse blocks are arranged successively, or when a rapid traverse block is followed by a block that does not cause movement. When the feedrate for each axis of a block is reduced to the ratio set in this parameter, the execution of the next block is started.

> **NOTE**
> The parameter No. 1722 is effective when parameter No. 1601 #4 (RTO) is set to 1.

Example

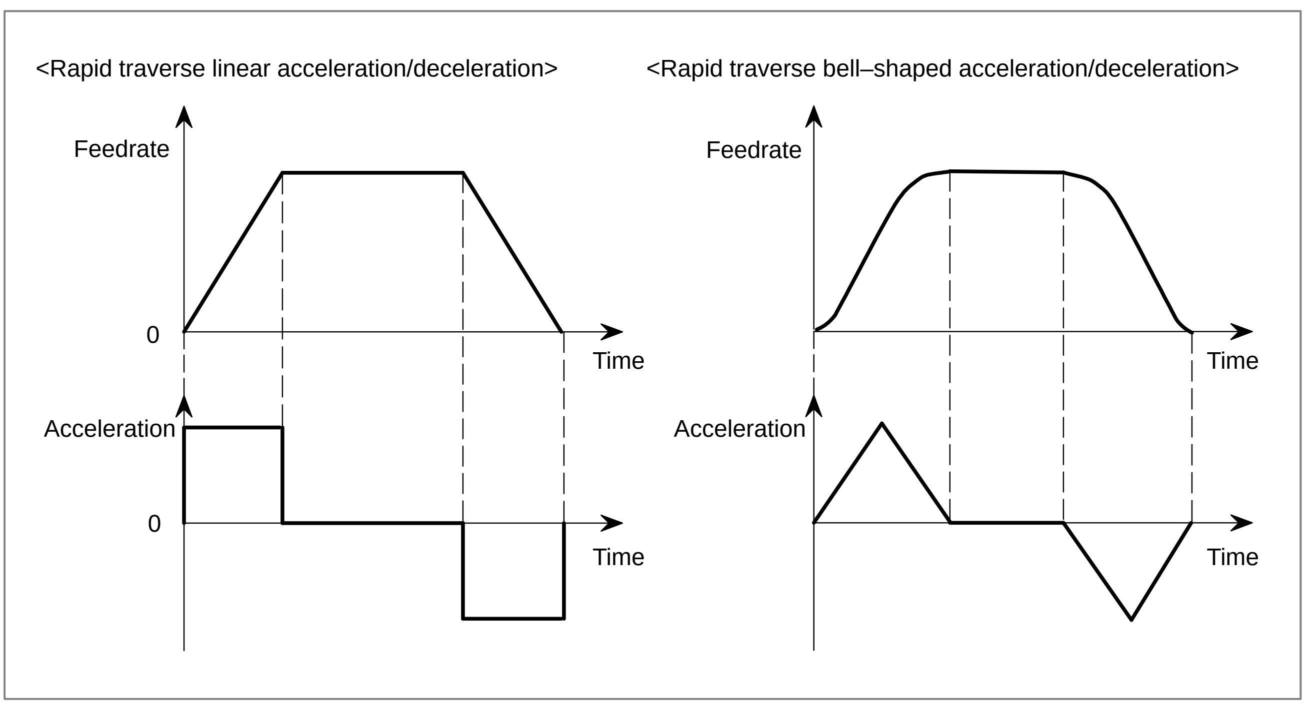
a



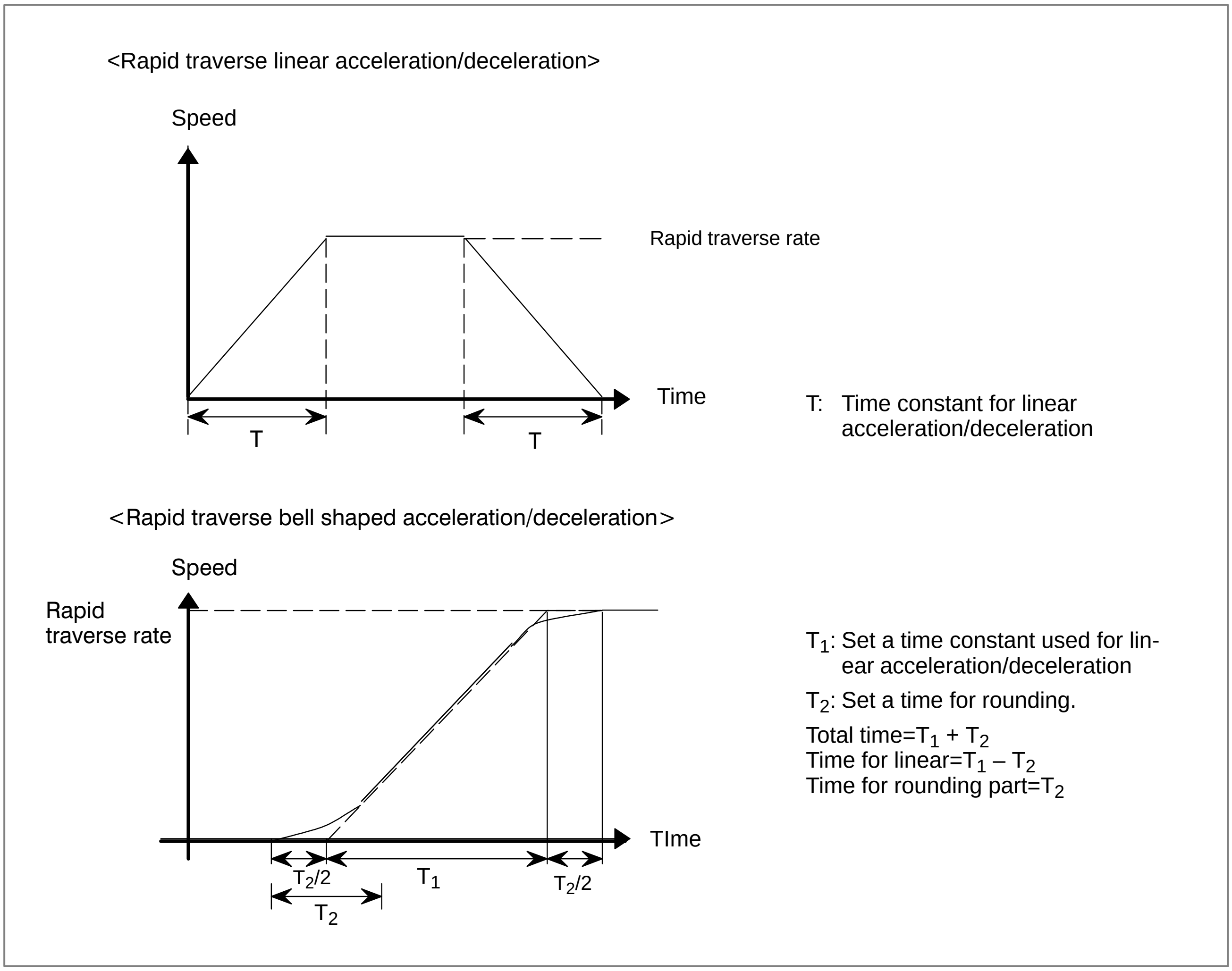
7.2.2 Rapid Traverse Bell–shaped Acceleration/ Deceleration

General

Rapid traverse bell–shaped acceleration/deceleration smoothly increases or decreases the rapid traverse rate, reducing the stress and strain imposed on the machine due to the variation in the acceleration with changes in the feedrate. As the time constant for bell–shaped acceleration/deceleration can be smaller than that for linear acceleration/deceleration, the time needed for acceleration/deceleration can be reduced.



This function is enabled when the time constants for rapid traverse bell–shaped acceleration/deceleration T1 and T2 are specified in parameter Nos. 1620 and 1621, 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

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Specify a time constant used for acceleration/deceleration in rapid traverse. When the 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 T1 used in bell–shaped acceleration/deceleration in rapid traverse, and set parameter No. 1621 to time constant T2.
- When the function is not provided, specify a time constant used for linear acceleration/deceleration.

> **NOTE**
> When parameter No. 1621 (time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse) is set to 0, linear acceleration/deceleration is applied in rapid traverse even if the function is provided. In this case, this parameter stands for a time constant used for linear acceleration/deceleration in rapid traverse.

1621	Time constant t T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 512

Specify time constant T2 used for bell–shaped acceleration/deceleration in rapid traverse for each axis.

> **NOTE**
> 1 This parameter is effective when the function of bell–shaped acceleration/deceleration in rapid traverse is provided. Set parameter No. 1620 to time constant T1 used for bell–shaped acceleration/deceleration in rapid traverse, and set this parameter to time constant T2.
> For details of time constants T1 and T2, see the 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 No. 1620 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 Cutting Feed Interpolation

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 0 of parameter No. 1610, CTL), acceleration/deceleration is performed as follows:

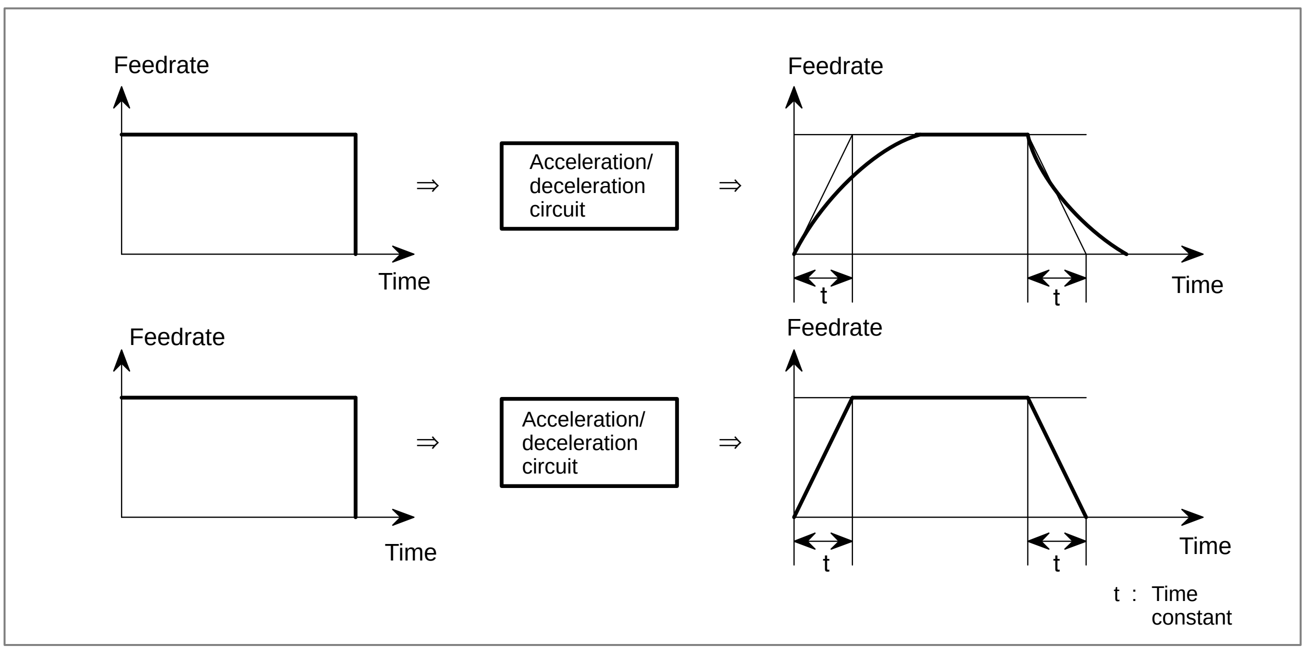
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If an identical time constant is specified, linear acceleration/deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/deceleration, thus reducing the time needed for acceleration and deceleration. If circular interpolation is performed, especially when high–speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.



Linear acceleration/deceleration after cutting feed interpolation is an optional function. This function is enabled when the CTL bit (bit 0 of parameter No. 1610) is specified. If bell–shaped acceleration/deceleration after interpolation for cutting feed is also enabled, bell–shaped acceleration/deceleration is executed. The time constants for cutting feed and jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1622	Time constant of exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/ deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, bell shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determined depending on the setting in parameter No. 1610.

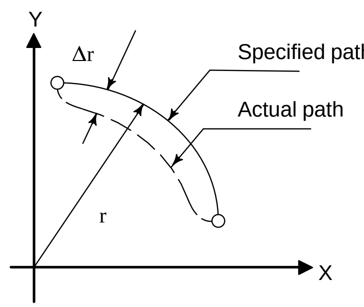
Note

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. Linear acceleration/deceleration can also be executed during jog feed if the JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

2 In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr :Maximum radius error (mm)
v :Feedrate (mm/s)
r :Radius of arc (mm)
T_1 :Acceleration/deceleration time constant (s)
T_2 :Time constant of servo loop (s)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2} T_1^2 + \frac{1}{2} T_2^2 \right) \frac{v^2}{r}$$ …Exponential acceleration/deceleration

$$\Delta r = \left(\frac{1}{24} T_1^2 + \frac{1}{2} T_2^2 \right) \frac{v^2}{r}$$ …Linear acceleration/deceleration 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 Bell–Shaped Acceleration/Deceleration after Cutting Feed Interpolation

General

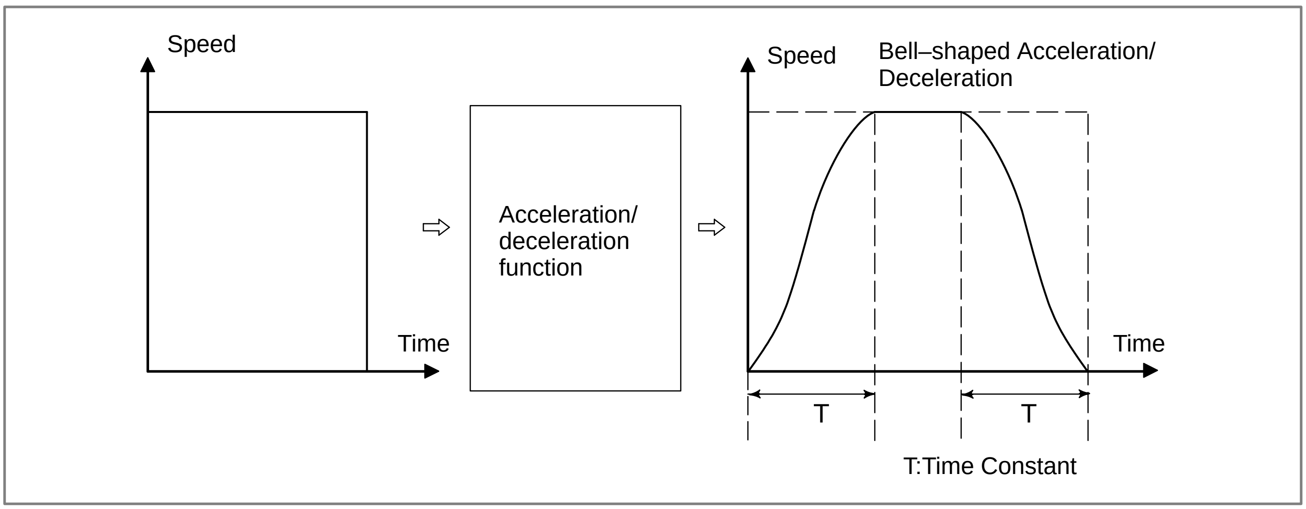
The bell–shaped acceleration/deceleration after cutting feed interpolation provides smooth acceleration and deceleration to reduce stress and strain on the machine. If this function is enabled (bit 1 of parameter No. 1610, CTB), acceleration/deceleration is performed as follows:

Cutting feed: Bell–shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or bell–shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.



Bell–shaped acceleration/deceleration after cutting feed interpolation is an optional function. This function is enabled when the CTB bit (bit 1 of parameter No. 1610) is specified. The time constants for cutting feed and for jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx			CTBx	CTLx

[Data type] Bit axis

CTLx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation is applied.

> **NOTE**
> 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.
> To use bell–shaped acceleration/deceleration after interpolation, set this parameter to 0 and select the acceleration/deceleration using CTBx, bit 1 of parameter No. 1610.

Parameter		Acceleration/deceleration
CTBx	CTLx	
0	0	Exponential acceleration/deceleration
0	1	Linear acceleration/deceleration after interpolation
1	0	Bell–shaped acceleration/deceleration after interpolation

CTBx Acceleration/deceleration in cutting feed including feed in dry run

0 : Exponential acceleration/deceleration or linear acceleration/deceleration after interpolation is applied (depending on the setting in CTLx, bit 0 of parameter No. 1610).
1 : Bell–shaped acceleration/deceleration after interpolation is applied.

> **NOTE**
> This parameter is effective only when the function of bell–shaped acceleration/deceleration after interpolation in cutting feed is provided. If the function is not provided, the setting in CTLx, bit 0 of parameter No. 1610, determines the type of acceleration/deceleration irrespective of the setting in this parameter.

JGLx Acceleration/deceleration in manual continuous feed (jog feed)

0 : Exponential acceleration/deceleration is applied.
1 : Linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation is applied (depending on which is used in cutting feed).

1622	Time constant of exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation, in cutting feed for each axis

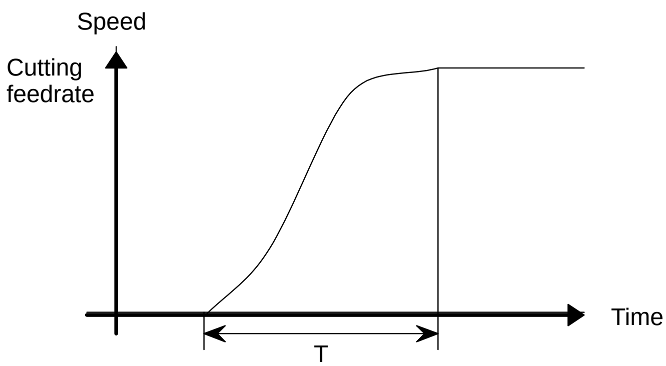
[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation in cutting feed for each axis. Except for special applications, the same time constant must be set for all axes in this parameter. If the time constants set for the axes differ from each other, proper straight lines and arcs cannot be obtained.

<Bell–shaped acceleration/deceleration after cutting feed interpolation>



T : Total time. It is constant irrespective of feedrate. (Time constant is constant).

The curve corresponds to that T1 = T/2 and T2 = T/2 set in parameter No. 1620 and 1621.
No linear part exists.

1624	Time constant of exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation, in jog feed for each axis.

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000 (For exponential acceleration/deceleration)
0 to 512 (For linear acceleration/deceleration after interpolation or bell–shaped acceleration/deceleration after interpolation)

Set the time constant used for exponential acceleration/deceleration, bell–shaped acceleration/deceleration after interpolation or linear acceleration/deceleration after interpolation in jog feed for each axis.
The type of acceleration/deceleration is determinded depending on the setting in parameter No. 1610.

Note

NOTE

1 If bell–shaped acceleration/deceleration after interpolation during cutting feed is enabled, bell–shaped acceleration/ deceleration is executed during cutting feed and during a dry run. Bell–shaped acceleration/deceleration can also be executed during jog feed if the JGL bit (bit 4 of parameter No. 1610) is specified accordingly.

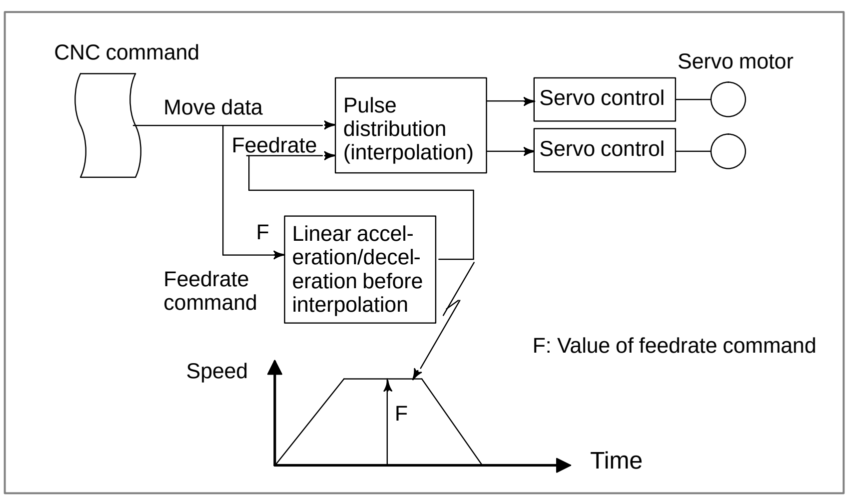
2 In circular interpolation, the actual tool path after acceleration/deceleration deviates from the programmed arc in the radial direction. To overcome this radial deviation, see the note on linear acceleration/deceleration after interpolation for cutting feed in Subsection 7.2.3.

3 Bell–shaped acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. The time constant for acceleration/deceleration for jog feed is the same as that for cutting feed. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

7.2.5 Linear Acceleration/Deceleration before Cutting Feed Interpolation

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

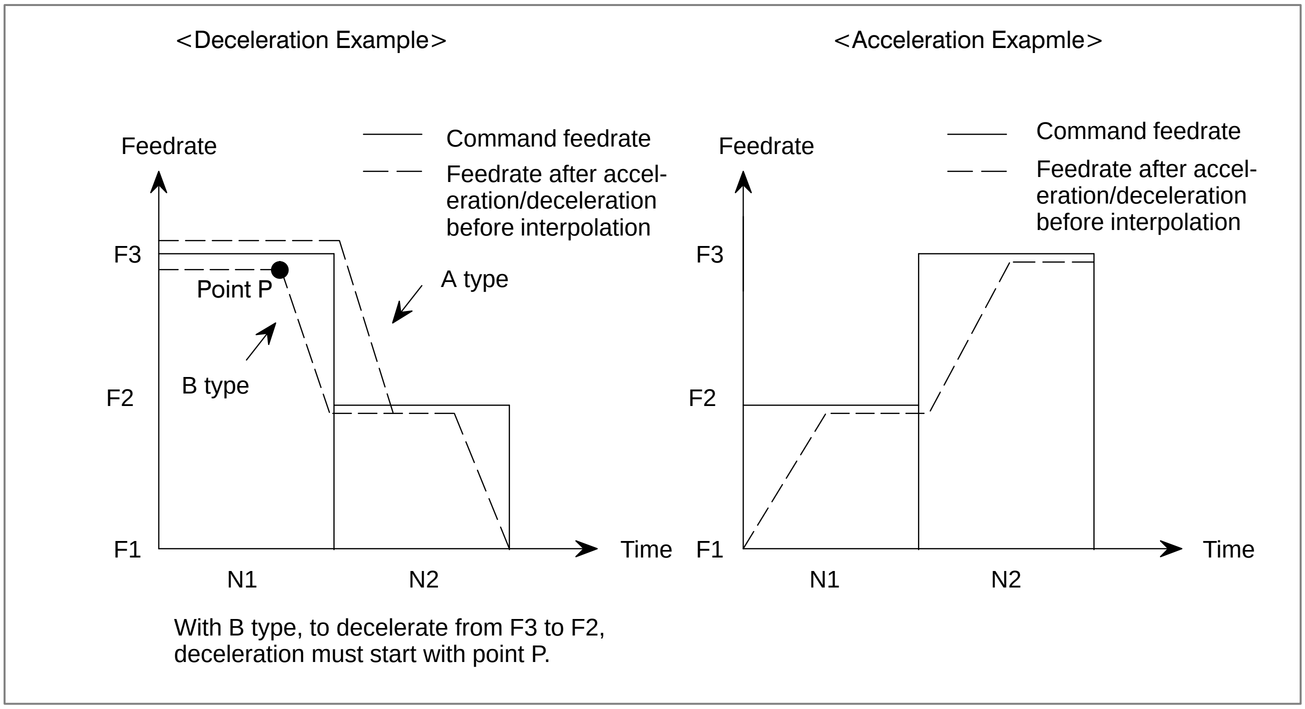
- Type A

Acceleration/deceleration is started in the block in which a new feedrate command is specified.

- Type B (Set the FWB bit (bit 0 of parameter No. 1602) to 1.)

Deceleration: Deceleration is started in a prior block such that deceleration is completed before the beginning of the block in which a new feedrate command is specified.

Acceleration: Acceleration is started in the block in which a new feedrate command is specified.



If an overtravel alarm occurs during linear acceleration/deceleration before interpolation, the movement is decelerated and stopped. As deceleration and stop are performed after the alarm occurs, the tool will overrun by an amount equal to the distance required for the deceleration. The actual overrun depends on the feedrate when the overtravel alarm occurs.

The distance can be minimized by starting deceleration in advance, such that the feedrate has fallen to the value specified in parameter No. 1784 when an overtravel alarm occurs. Because deceleration is executed such that the feedrate at the time an overtravel alarm occurs does not exceed the feedrate specified in the corresponding parameter, deceleration may be completed earlier. After deceleration is completed, the feedrate specified in the parameter is maintained.

Deceleration is performed when the following condition is satisfied:

Distance to stored stroke limit 1 for each axis	<	Distance needed to reduce the current feedrate (tangential feedrate) to that specified in parameter No. 1784

The overrun is calculated as follows:

$$\text{Overrun} \leqq \frac{[\text{FIX}\ (\frac{F_{OT}}{F} \times \frac{T}{8}) + 1.5]^2 \times \frac{F}{T}}{1875}$$

F : Maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1630)

T : Time needed to attain the maximum cutting feedrate in linear acceleration/deceleration before interpolation (parameter No. 1631)

F_{OT}: Feedrate at the time an overtravel alarm occurs during linear acceleration/deceleration before interpolation (parameter No. 1784)

FIX: Any fractional part is truncated.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1602								FWB

[Data type] Bit

FWB 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.

1630	Parameter 1 for setting an acceleration for linear acceleration/deceleration before interpolation (maximum machining feedrate during linear acceleration/deceleration before interpolation)

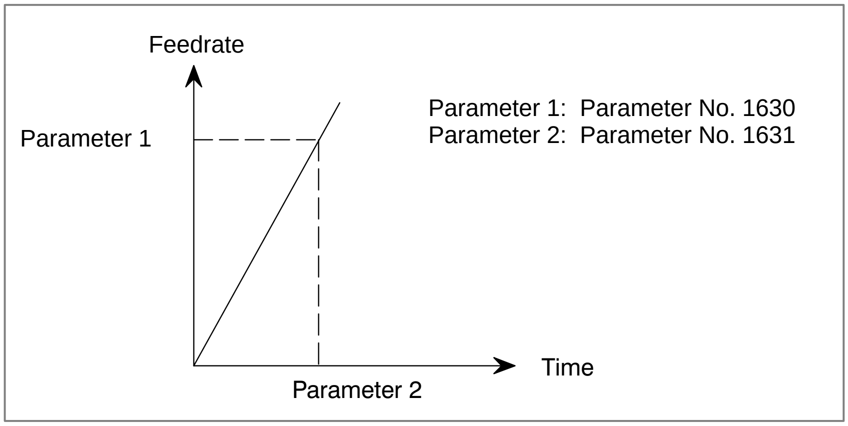
[Data type] Two–word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 240000	6 – 100000
Inch machine	0.1 inch/min	6 – 96000	6 – 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. 1631, set a time used to reach the maximum machining feedrate.



NOTE
1 When 0 is set in parameter No. 1630 or parameter No. 1631, linear acceleration/deceleration before interpolation is disabled.
2 In the advanced preview control mode, parameter No. 1770 and parameter No. 1771 are valid.

1631	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. 1630.

NOTE
1 When 0 is set in parameter No. 1630 or parameter No. 1631, linear acceleration/deceleration before interpolation is disabled.
2 In parameter Nos. 1630 and 1631, set values that satisfy the following:

$$\frac{\text{Parameter No. 1630}}{\text{Parameter No. 1631}} \geqq 5$$

3 In the advanced preview control mode, parameter No. 1770 and parameter No. 1771 are valid.

1784	Feedrate when overtravel alarm has generated during acceleration/deceleration before interpolation

[Data type] Word

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 – 15000	6 – 12000
Inch machine	0.1 inch/min	6 – 6000	6 – 4800

Deceleration is started beforehand to reach the feedrate set in the parameter when an overtravel alarm is issued (when a limit is reached) during linear acceleration/deceleration before interpolation. By using this parameter, the overrun distance that occurs when an overtravel alarm is output can be reduced.

WARNING

The control described above is applicable only to stored stroke limit 1.

NOTE

1 When 0 is set in this parameter, the control described above is not exercised.

2 Use type–B linear acceleration/deceleration before interpolation (by setting bit 0 (FWB) of parameter No. 1602 to 1).

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 If the machine lock signal (MLK1 to MLK8) for an axis is set on or off during acceleration/deceleration before interpolation, the axis for which machine lock is performed is not subjected to acceleration/deceleration.
6 During acceleration/deceleration before interpolation, automatic corner override is enabled only when the internal circular cutting feedrate is changed.
7 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.
8 In acceleration/deceleration before interpolation of type B, deceleration is started if preprocessing for the next block has not been completed before the remaining distance of the current block becomes less than that needed to decelerate and stop the movement.
9 If an F1–digit command is executed in the inch input system, avoid specifying a command for simultaneous movement on two axes, including a rotation axis during acceleration/deceleration before interpolation (M series).
10 The error detect signal (SMZ) is invalid during acceleration/deceleration before interpolation (T series).

7.2.6 Corner Control

7.2.6.1 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 No. 1826).

Signal

In–position signals INP1 to INP8 <F104>

[Classification] Output signal

[Function] These signals indicate that the control axes are set to the in-position condition.
They are provided for each control axis, and the number in the signal name corresponds to the control axis number.

I N P 1

1 . . . . The 1st axis is set to the in-position condition.

2 . . . . The 2nd axis is set to the in-position condition.

3 . . . . The 3rd axis is set to the in-position condition.

: :

: :

[Output condition] These signals turn to “1” in the following case :

· The 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
F104	INP8	INP7	INP6	INP5	INP4	INP3	INP2	INP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI Inposition check at deceleration

0 : Performed
1 : Not performed

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

The in–position width is set for each axis.

When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in–position width, the machine is assumed to have reached the specified position.

Note

NOTE
1 The in–position signals may turn to “1” even during the movement if the axis is fed at very low speed.
2 The in–position check function is enabled, at the interface between two cutting blocks, in the following cases:

M series	When the exact stop command (G09) or exact stop mode command (G61) is specified
T series	When the error detect signal is on

7.2.6.2 In–position check independently of feed/rapid traverse

General

If separate in–position check for cutting feed and rapid traverse is executed, a small in–position check range can be specified between those cutting feed blocks that require a high degree of precision. A large in–position check range can be specified between those rapid traverse blocks that require quick positioning.

Signal

See Subsection 7.2.6.1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1601			NCI					

[Data type] Bit

NCI Inposition check at deceleration

0 : Performed
1 : Not performed

	#7	#6	#5	#4	#3	#2	#1	#0
1801			CIN	CCI				

[Data type] Bit

CCI The in–position area for cutting feed is:

0 : Set in parameter No. 1826 (same as for rapid traverse).
1 : Set in bit 5 (CIN) of parameter No. 1801.

CIN When bit 4 (CCI) of parameter No. 1801 = 1, the in–position area for cutting feed is:

0 : Use value in parameter No. 1827 if the next block is also for cutting feed, or use value in parameter No. 1826 if the next block is for rapid traverse.
1 : Use value in parameter No. 1827, regardless of the next block. (The setting of parameter No. 1826 is used for rapid traverse, and the setting of parameter No. 1827 is used for cutting feed.)

		Parameter CIN (No. 1801#5)			
		0		**1**	
Parameter CCI (No. 1801#4)	0	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1826 No. 1826	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1826 No. 1826
	1	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No.1826 No.1827 No. 1826	Rapid→Rapid Rapid→Feed Feed→Feed Feed→Rapid	No. 1826 No. 1826 No. 1827 No. 1827

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

The in–position width is set for each axis.

When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in–position width, the machine is assumed to have reached the specified position.

1827	In–position width in cutting feed for each axis

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set an in–position width for each axis in cutting feed. This parameter is valid when bit 4 (CCI) of parameter No. 1801=1.

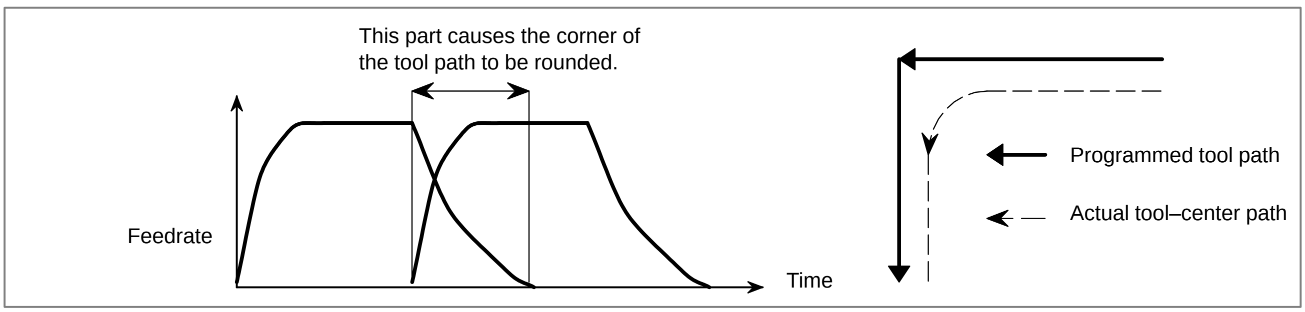
Note

NOTE
If the NCI bit (bit 5 of parameter No. 1601) is set to 1, so that position check is not performed during deceleration, this function is invalid. The system starts execution of the next block as soon as deceleration has been completed, without checking whether the servo position error is within the specified range.

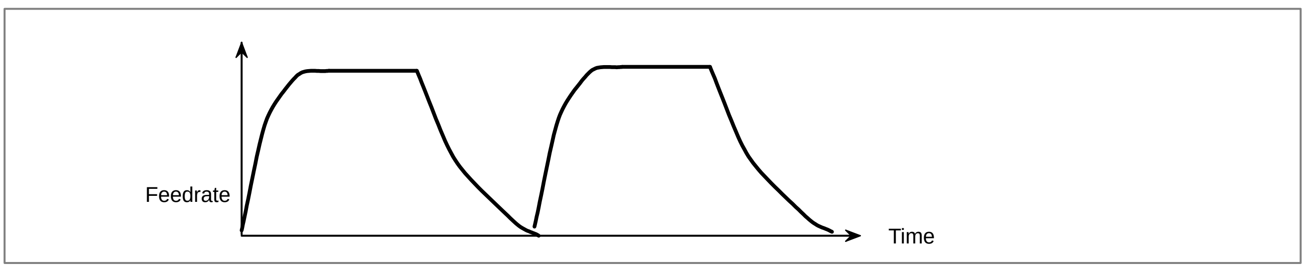
7.2.6.3 Error detect (T series)

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 <G053, #6>

[Classification] Input signal

[Function] Enables error detection.

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

- At the interface of two blocks during cutting feed, the control unit waits until the acceleration/deceleration of the first block has been completed. Only then does the control unit execute the next block.
  The setting of the SMZ signal determines whether, at the interface of two cutting blocks, the control unit waits until the acceleration/deceleration of the previous block has been completed.

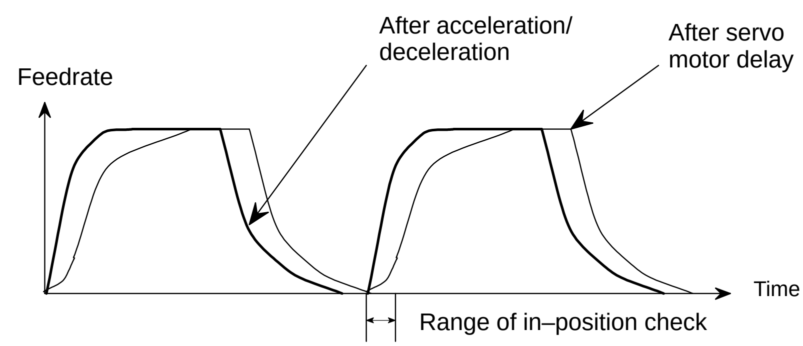
Siganl address

	#7	#6	#5	#4	#3	#2	#1	#0
G053		SMZ						

Note

NOTE

If the error detect signal is on, a cutting block is not executed until the acceleration/deceleration of the previous cutting block has been completed.
This function alone cannot prevent corner rounding due to delay caused by the servo motor, however.
To prevent corner rounding due to delay caused by the servo motor, use the in–position check function together with this function.



7.2.7 Feed Forward in Rapid Traverse

General

Feed–forward control can be performed even during rapid traverse. In this case, the servo position error is reduced, thus reducing the time required for positioning to within the specified range.

Parameter

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

[Data type] Bit

FFR Feed–forward control is enabled for

0 : Cutting feed only
1 : Cutting feed and rapid traverse

Reference item

For details of this function, refer to the "FANUC CONTROL MOTOR α series Maintenance Manual (B–65165E)."

8 AUXILIARY FUNCTION

8.1 MISCELLANEOUS FUNCTION/2ND AUXILIARYFUNCTION

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”)
Also, parameter No. 3030 can specify the maximum digits and if a specified value exceeds the maximum digits, an alarm may be issued.

• 2nd Auxiliary Function (B code)

When eight digits are specified after address B, a code signal and strobe signal are sent. These signals are used to index the rotation axis of the machine. The code signal is retained until another B code is specified.

In each block, a single B code can be specified. The maximum number of digits that can be specified after address B is specified in parameter No. 3033. If more digits than the specified value are specified, an alarm occurs.
For the M series, the address for specifying the 2nd auxiliary function can be changed from B to another address (A, C, U, V, or W) by parameter setting (parameter No. 3460).

• Command Format of 2nd Auxiliary Function

· Command range

–99999999 to +99999999

· Command method

1. For the M series, a decimal point and a negative valve can be used for input by setting AUP (bit 0 of parameter No. 3450).
   With the T series, a decimal point and a negative value are always enabled regardless of the parameter setting.

Command	Output value
B10.	10000
B10	10

2. It is possible to change over the scale factor of B output, 1000 or 1 when the decimal point input is omitted, using the parameter DPI (No.3401#0).

Command		Output value
When DPI is 1:	B1	1000
When DPI is 0:	B1	1

3. It is possible to change over the scale factor of B output 1000 or 10000 when the decimal point input is omitted in the inch input system, using the parameter AUX (No.3405#0). When DPI=1.
   Command Output value
   When AUX is 1: B1 10000
   When AUX is 0: B1 1000

Basic procedure

The following signals are used with these functions. (For details of the spindle–speed function and tool function, see Chapters 9 and 10.)

Function	Program address	Output signal			Input signal
		Code signal	Strobe signal	Distribution end signal	Completion signal
Miscellaneous function	M	M00 to M31	MF	DEN	FIN
Spindle–speed function	S	S00 to S31	SF		
Tool function	T	T00 to T31	TF		
Secondary auxiliary function	B	B00 to B31	BF		

Each function uses different program addresses and different signals, but they all input and output signals in the same way, as described below. (A sample procedure for the miscellaneous function is described below. The procedures for the spindle–speed function, tool function, and secondary auxiliary function, are obtained simply by substituting S, T, or B in place of M.)

(1) Suppose that Mxxx is specified in a program.
   For xxx, the number of specifiable digits is specified in parameter Nos. 3030 to 3033 for each function. If more digits than the specified value are specified, an alarm occurs.

(2) Code signal M00 to M31 is sent. After period TMF, specified in parameter No. 3010 (standard value: 16 msec), strobe signal MF is set to 1. The code signal is the binary representation 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. 3011 (standard value: 16 msec), the CNC sets the strobe signal to 0 and reports that the completion signal has been received.

(7) When the strobe signal is set to 0, set the completion signal to 0 in the PMC.

(8) When the completion signal is set to 0, the CNC sets all code signals to 0 and completes all sequences of the miscellaneous function.(*2)

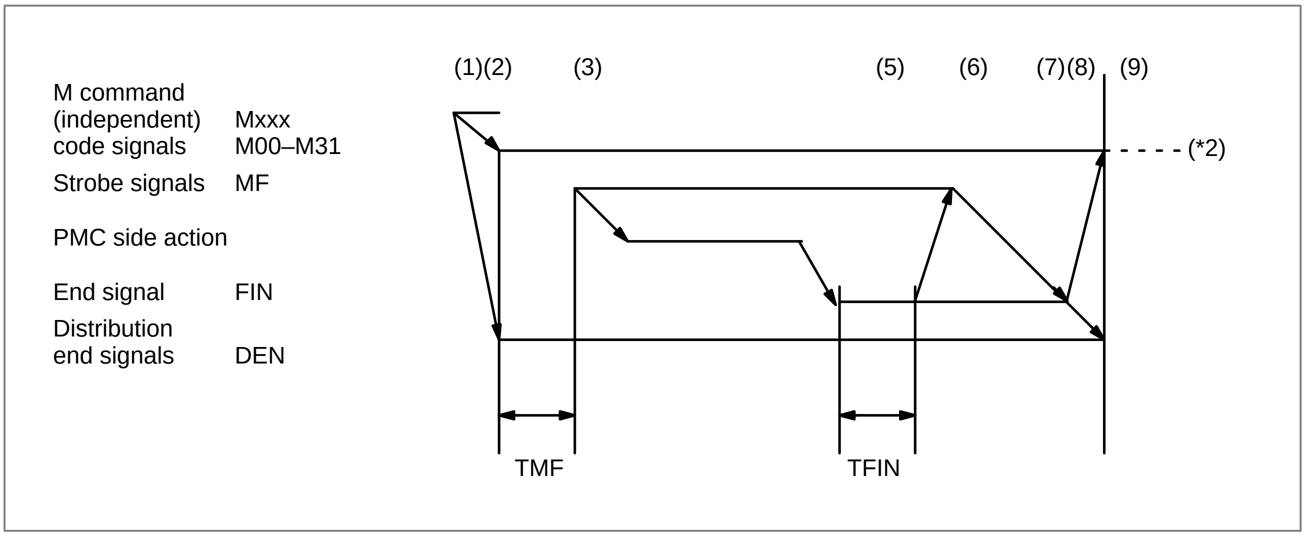
(9) Once all other commands in the same block have been completed, the CNC executes the next block.

*1 When the tool function is executed, the programmed tool number is sent as the code signal (T series).

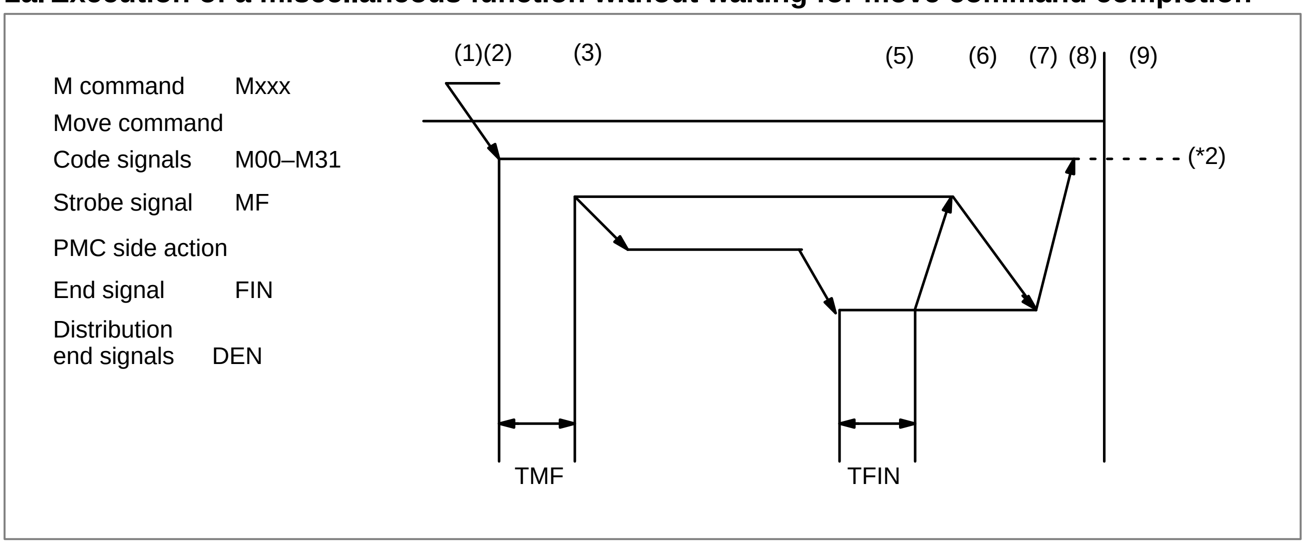
*2 When the spindle–speed function, tool function, or secondary auxiliary function is executed, the code signal is maintained until a new code for the corresponding function is specified.

The timing diagram is shown below:

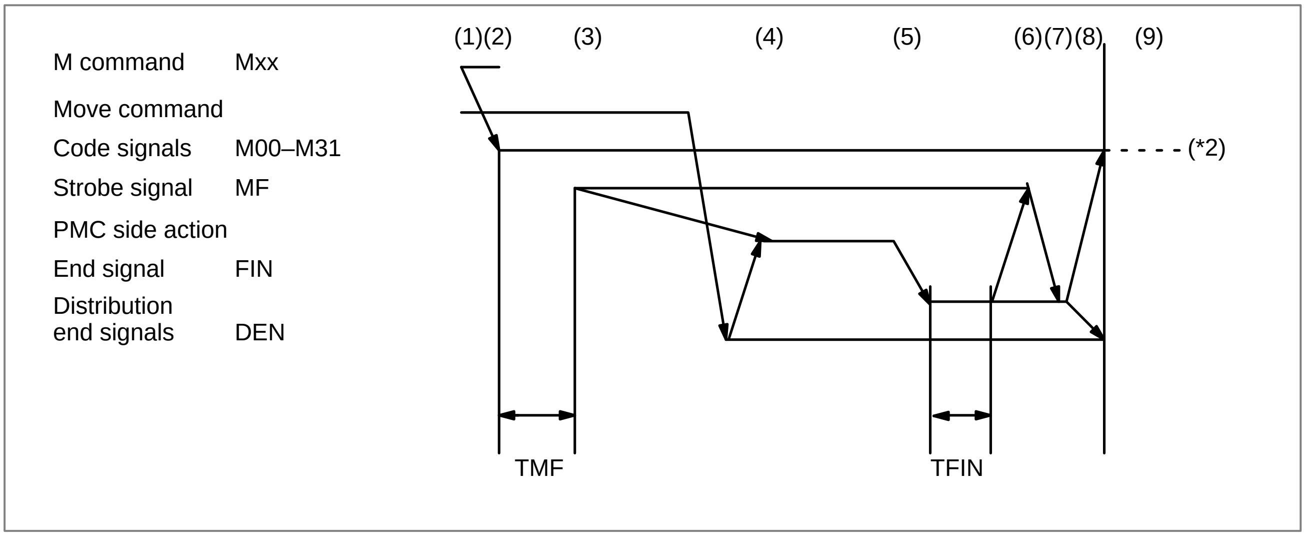
Example 1 Single miscellaneous function specified in a block



Example 2 Move command and miscellaneous function in the same block
2a. Execution of a miscellaneous function without waiting for move command completion



2b. Execution of a miscellaneous function after move command completion



Signal

Miscellaneous function code signals M00 to M31 <F010 to F013> Miscellaneous function strobe signal MF <F007#0>

[Classification] Output signal

[Function] These signals report the specification of miscellaneous functions.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

NOTE

1 The following miscellaneous functions are only processed internally by the control unit; they are not subject to output even when specified:
- M98, M99, M198
- M code that calls a sub program (parameter No. 6071 to 6079)
- M code that calls a custom macro (parameter No. 6080 to 6089)

2 Decode signals as well as the code signals and strobe signal are output for the miscellaneous functions listed below.
M00, M01, M02, M30

Decode M signals DM00 <F009#7>, DM01 <F009#6>, DM02 <F009#5>, DM30 <F009#4>

[Classification] Output signal

[Function] These signals report particular miscellaneous functions are specified. The miscellaneous functions in a command program correspond to output signals as indicated below.

Command program	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

[Output condition] A decode M signal goes “1” when:

· The corresponding miscellaneous function is specified, and any move commands and dwell commands specified in the same block are completed. These signals are not output when the end signal of the miscellaneous function is returned before completion of such move commands and dwell commands.

A decode M signal goes “0” when:

· The FIN signal goes “1”

· Reset occurs

Spindle–speed code signals S00 to S31 <F022–F025> Spindle–speed strobe signal SF <F007#2>

[Classification] Output signal

[Function] These signals report that spindle speed functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.
For S code output when the spindle serial output/spindle analog output is used refer to section 9.3.

Tool function code signals T00 to T31 <F026–F029> Tool function strobe signal TF <F007#3>

[Classification] Output signal

[Function] These signals report that tool functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

Second auxiliary function code signals B00 to B31 <F030–F033> Second auxiliary function strobe signal BF <F007#4> (For lathes) <F007#7> (For machining center)

[Classification] Output signal

[Function] These signals report that second auxiliary functions have been specified.

[Output condition] For the output conditions and procedure, see the description of “Basic procedure” above.

End signal FIN <G004#3>

[Classification] Input signal

[Function] This signal reports the completion of a miscellaneous function, spindle–speed function, tool function, second auxiliary function, or external operation function.

[Operation] For the control unit operation and procedure when this signal goes “1”, see the description of “Basic procedure” above.
The FIN signal must remain “1” for a certain time (TFIN, which is set by a parameter No. 3011) or longer. The FIN signal driven “1” is ignored if the FIN signal goes “0” before TFIN elapses.

> **WARNING**
> Only one end signal is used for all functions above. The end signal must go “1” after all functions are completed.

Distribution end signal DEN <F001#3>

[Classification] Output signal

[Function] These signals report that all commands (such as move commands and dwell) are completed except those miscellaneous functions, spindle–speed functions, 2nd auxilialy functions tool functions, and so forth that are contained in the same block and have been sent to the PMC. They also report that the end signal from the PMC is being awaited.

[Output condition] The DEN signal turns to “1” when:

· The completion of miscellaneous functions, spindle–speed functions, tool functions, 2nd auxilialy 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 NCI (No.1601#5) can specify, whether to only check if an acceleration/deceleration delay is eliminated, or to also check if a servo delay (error) has been reduced to within a certain range.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004					FIN			

	#7	#6	#5	#4	#3	#2	#1	#0
F001					DEN			
F007	BF			BF	TF	SF		MF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24

Parameter

3010	Time lag in strobe signals MF, SF, TF, and BF

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 to 32767

The time required to send strobe signals MF, SF, TF, and BF after the M, S, T, and B codes are sent, respectively.

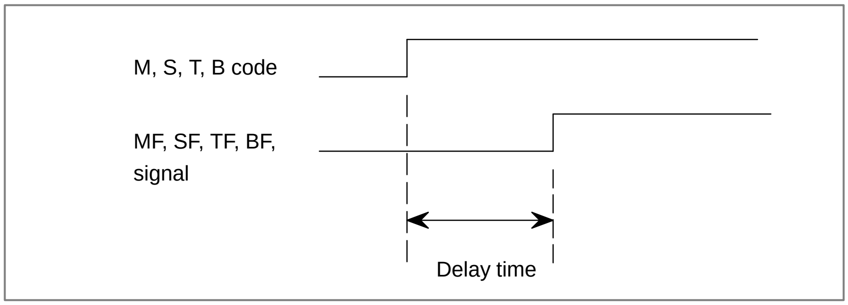


Fig. 8.1 (a) Delay time of the strobe signal

> **NOTE**
> The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.
>
> Example : When 30 is set, 32 ms is assumed.
> When 32 is set, 32 ms is assumed.
> When 100 is set, 104 ms is assumed.

3011	Acceptable width of M, S, T, and B function completion signal (FIN)

[Data type] Word

[Unit of data] 1 ms

[Valid data range] 16 to 32767

Set the minimum signal width of the valid M, S, T, and B function completion signal (FIN).

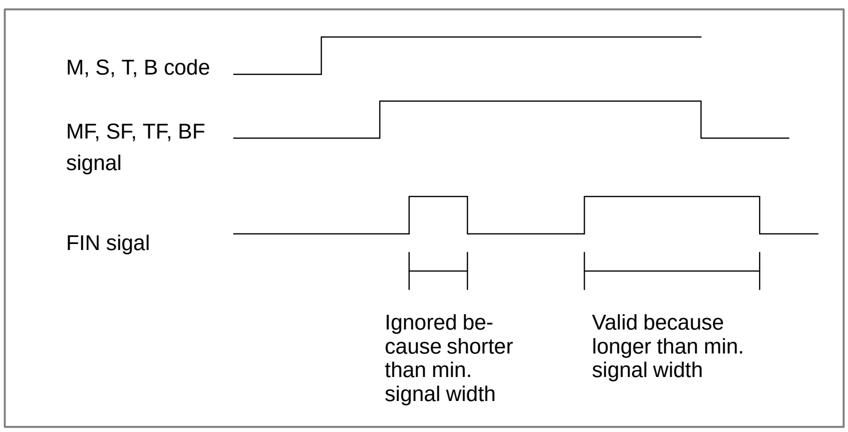


Fig. 8.1 (b) Valid Width of the FIN (M,S, T, and B Function Completion) Signal

> **NOTE**
> The time is counted in units of 8 ms. If the set value is not a multiple of eight, it is raised to the next multiple of eight.
>
> Example: When 30 is set, 32 ms is assumed.

3030	Allowable number of digits for the M code
3031	Allowable number of digits for the S code
3032	Allowable number of digits for the T code
3033	Allowable number of digits for the B code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the M, S, T, and B codes.

NOTE
Up to 5 digits can be specified in the S code

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point

0 : The least input increment is assumed.
1 : The unit of mm, inches, or second is assumed. (Pocket calculator type decimal point input)

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02	M30				

[Data type] Bit

M30 When M30 is specified in a memory operation:

0 : M30 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program is executed, starting from the beginning.
1 : M30 is sent to the machine, but the head of the program is not searched for. (The head of the program is searched for by the reset and rewind signal.)

M02 When M02 is specified in memory operation

0 : M02 is sent to the machine, and the head of the program is automatically searched for. So, when the end signal FIN is returned and a reset or reset and rewind operation is not performed, the program 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
3405									AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point

0 : Assumed to be 0.001
1 : Depending on the input increment. (For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

3411	M code preventing buffering 1
3412	M code preventing buffering 2
3413	M code preventing buffering 3
⋮	⋮
3420	M code preventing buffering 10

[Data type] Byte

[Valid data range] 0 to 255

Set M codes that prevent buffering the following blocks. If processing directed by an M code must be performed by the machine without buffering the following block, specify the M code.

M00, M01, M02, and M30 always prevent buffering even when they are not specified in these parameters.

3421	Minimum value 1 of M code preventing buffering
3422	Maximum value 1 of M code preventing buffering
3423	Minimum value 2 of M code preventing buffering
3424	Maximum value 2 of M code preventing buffering
3425	Minimum value 3 of M code preventing buffering
3426	Maximum value 3 of M code preventing buffering
3427	Minimum value 4 of M code preventing buffering
3428	Maximum value 4 of M code preventing buffering
3429	Minimum value 5 of M code preventing buffering
3430	Maximum value 5 of M code preventing buffering
3431	Minimum value 6 of M code preventing buffering
3432	Maximum value 6 of M code preventing buffering

[Data type] Word

[Valid data range] 0 to 65535

When a specified M code is within the range specified with parameter Nos. 3421 and 3422, 3433 and 3424, 3425 and 3426, 3427 and 3428, 3429 and 3430, or 3431 and 3432, buffering for the next block is not performed until the execution of the block is completed.

CAUTION
1 The specification of a minimum value that exceeds the specified maximum value is invalid.
2 When there is only one data item, set the following: minimum value =maximum value.

	#7	#6	#5	#4	#3	#2	#1	#0
3450								
								AUP

AUP A second auxiliary function command, specified with a decimal point and a negative value is:

0 : Disabled.
1 : Enabled.

NOTE
With the T series, second auxiliary function commands specified with a decimal point and a negative value are always enabled regardless of the parameter setting.

3460	
	Name of a second auxiliary function

[Data type] Byte

Set the name of a second auxiliary function according to the table given below. Note that the same address as an axis name cannot be set.

Name	A	B	C	U	V	W
Setting	65	66	67	85	86	87

If a value other than those given above is set, address B is assumed.

Note

NOTE

1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:
   i) Simultaneous execution of the move command and miscellaneous function commands.
   ii) Executing miscellaneous function commands upon completion of move command execution.
   The selection of either sequence depends on the sequence of PMC.
2 When the 2nd auxiliary function is provided, the address used for specifying the 2nd auxiliary function (B or the address specified with parameter No. 3460) cannot be used as an axis address.
3 The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (No. 3411 to 3420).
4 For M00 and M01 only, miscellaneous function code signal, auxiliary function strobe signal, and M decode signals are sent; the control of program stop and optional stop shall be designed on the PMC side.
5 When the automatic operation is stopped by M02 or M30, it is necessary to send the external reset signal from the machine side to the CNC, instead of the FIN signal, When the external reset signal is returned against the M02 or M30, the control returns to the top of the program recently executed and enters the reset state. When the FIN signal is returned, the control returns to the beginning of the program recently executed and executes it from the top.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.11.1	Miscellaneous function (M code)
		II.11.4	2nd Auxiliary function (B code)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.11.1	Miscellaneous function (M code)
		II.11.4	2nd Auxiliary function (B code)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.11.1	Miscellaneous function (M code)
		II.11.3	2nd Auxiliary function (B code)

8.2 AUXILIARY FUNCTION LOCK

General

Inhibits execution of a specified M, S, T and B function.
That is, code signals and strobe signals are not issued.
This function is used to check a program.

Signal

Auxiliary function lock signal AFL <G005#6>

[Classification] Input signal

[Function] This signal selects auxiliary function lock. That is, this signal disables the execution of specified M, S, T, and B functions.

[Operation] When this signal turns to “1”, the control unit functions as described below.

(1) The control unit does not execute M, S, T, and B functions specified for memory operation, DNC operation, or MDI operation. That is, the control unit stops the output of code signals and strobe signals (MF, SF, TF, BF).

(2) If this signal turns to “1” after code signal output, the output operation is executed in the ordinary manner until its completion (that is, until the FIN signal is received, and the strobe signal turns to “0”).

(3) Among the miscellaneous functions, M00, M01, M02, and M30 are executed even when this signal is “1”. All code signals, strobe signals, decode signals are output in the ordinary manner.

(4) Among the miscellaneous functions, even when this signal is “1”, those functions (M98 and M99) that are executed in the control unit without outputting their execution results are executed in the ordinary manner.

WARNING
Even when this signal is “1”, spindle analog output or spindle serial output is executed.

Auxiliary function lock check signal MAFL <F004#4>

[Classification] Output signal

[Function] This signal reports the state of the auxiliary function lock signal AFL.

[Output condition] This signal turns to “1” when:

· The auxiliary function lock signal AFL is “1”

This signal turns to “0” when:

· The auxiliary function lock signal AFL is “0”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G005		AFL						

	#7	#6	#5	#4	#3	#2	#1	#0
F004				MAFL				

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.5.1	Machine lock and auxiliary function lock
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.5.1	Machine lock and auxiliary function lock
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.5.1	Machine lock and auxiliary function lock
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.5.1	Machine lock and auxiliary function lock

8.3 MULTIPLE M COMMANDS IN A SINGLE BLOCK

General

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block.
Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

(Example)

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Y0Z0 ; : : :	M40M50M60 ; G28G91X0Y0Z0 ; : : : : :

Basic procedure

(1) Assume that “MaaMbbMcc;” was commanded by the program.

(2) The 1st M command (Maa) sends the code signals M00 to M31 in a manner similar to the conventional one-block single command. The strobe signal MF is set to “1” after a time TMF set by parameter No. 3010 (Standard setting: 16 msec).
The second M command (Mbb) sends the code signal M200-M215, the third M command (Mcc) sends the code signal M300-M315, and their respective strobe signals MF2 and MF3 are set to “1”.
Furthermore, the three code signals are sent simultaneously.
The strobe signals MF, MF2, and MF3 become “1” at the same time.
The code signal is a binary notation of the program command aa, bb and cc.

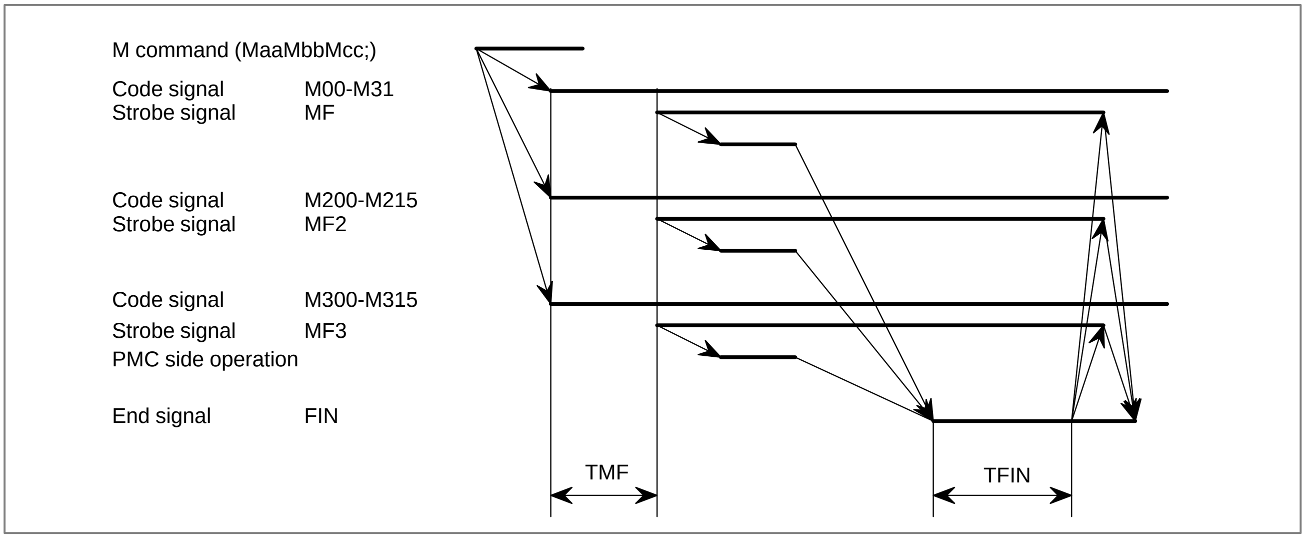
(3) On the PMC side, the code signals corresponding to the respective strobe signals are read when the strobe signals become “1”, and the appropriate operations are performed.

(4) When the operation of all M commands ends on the PMC side, the end signal (FIN) is set to “1”.

(5) When the completion signal stays “1” for a time (TFIN) set by parameter No. 3011 (Standard: 16 msec), all strobe signals (MF, MF2 and MF3) are set to “0” at the same time and the reception of completion signal is reported.

(6) On the PMC side, when MF, MF2 and MF3 are set to “0”, the completion signal is set to “0”.

A time chart for this procedure is shown below:



Signal

2nd, 3rd M function code signal M200 to M215 <F014, F015> M300 to M315 <F016, F017> 2nd, 3rd M Function strobe signal MF2 <F008#4> MF3 <F008#5>

[Classification] Output signal

[Function] Indicates that second and third auxiliary functions have been issued.

[Output condition] The output conditions and procedures are the same as that described in "Basic procedure".

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008			MF3	MF2				
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B							

[Data type] Bit

M3B The number of M codes that can be specified in one block
0 : One
1 : Up to three

Caution

CAUTION
1 M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
2 Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block. Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the PMC. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks.
The M codes which can be specified in a single block must be those which the CNC send only the M code signals to the PMC side.

Note

NOTE
1 CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For example, M42 can be specified only after the mechanical operation of M41 is completed.
2 The 1st M code can be up to 8 digits and 2nd, 3rd M codes can be the values up to 65535.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.11.2	Multiple M commands in a single block
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.11.2	Multiple M commands in a single block
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.11.2	Multiple M commands in a single block
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.11.2	Multiple M commands in a single block

8.4 HIGH–SPEED M/S/T/B INTERFACE

General

To accelerate M/S/T/B function execution, the high–speed M/S/T/B interface has simplified the transfer of the strobe and completion signals of the M/S/T/B functions.

Whether to use the usual system or high–speed system for strobe signal and completion signal handling can be specified by parameter MHI (No. 3001#7).

The description below uses the miscellaneous functions (M code commands) as an example. The same description applies to the spindle–speed function (S code), tool function (T code) and 2nd auxiliary function (B code).

Basic procedure

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the NC system sends out the code signals M00 to M31.
The NC system inverts the logical level of the strobe signal MF, that is, from “0”to “1”, or from “1” to “0”.

(3) The CNC system inverts the strobe signal, then when the logical level of the auxiliary function completion signal MFIN becomes the same as the strobe signal, the CNC assumes the completion of PMC sequence.

With the usual method, the operation is assumed to be completed when a falling edge (“1” to “0”) of the M/S/T/B completion signal FIN is received after a rising edge (“0” to “1”) of the FIN signal is detected. This new system, on the other hand, assumes the operation has been completed upon detection of only one transition of the completion signal.
In addition , the usual system uses only one completion signal (FIN) common to the M/S/T/B functions. This new system uses a different completion signal for each of the M, S, T, and B functions; the completion signals for the M, S, T, and B functions are MFIN, SFIN, TFIN, and BFIN, respectively.
The Fig. 8.4 (a) below shows the timing chart of these signals with the new system. For comparison, Fig. 8.4 (b) shows the timing chart of the conventional system.

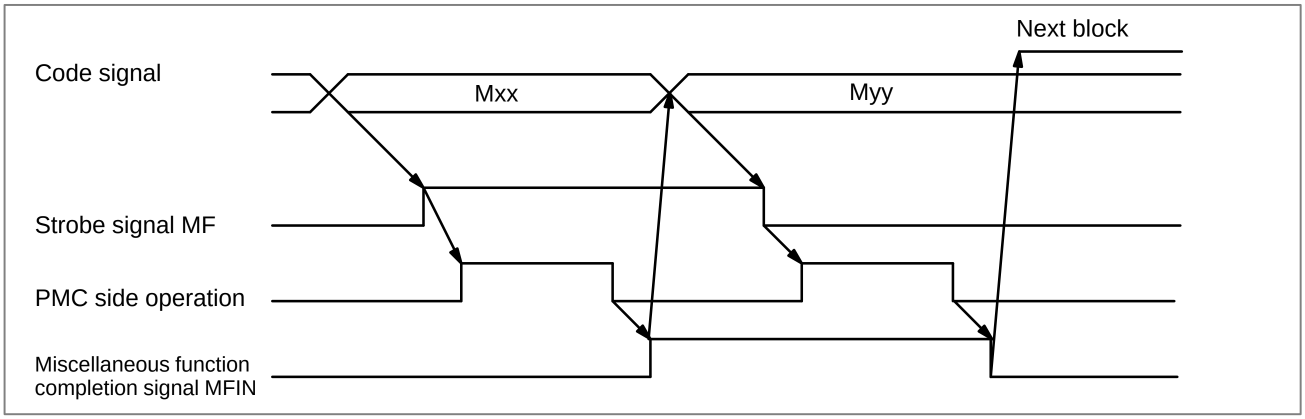


Fig. 8.4 (a) Timing chart of the high–speed system

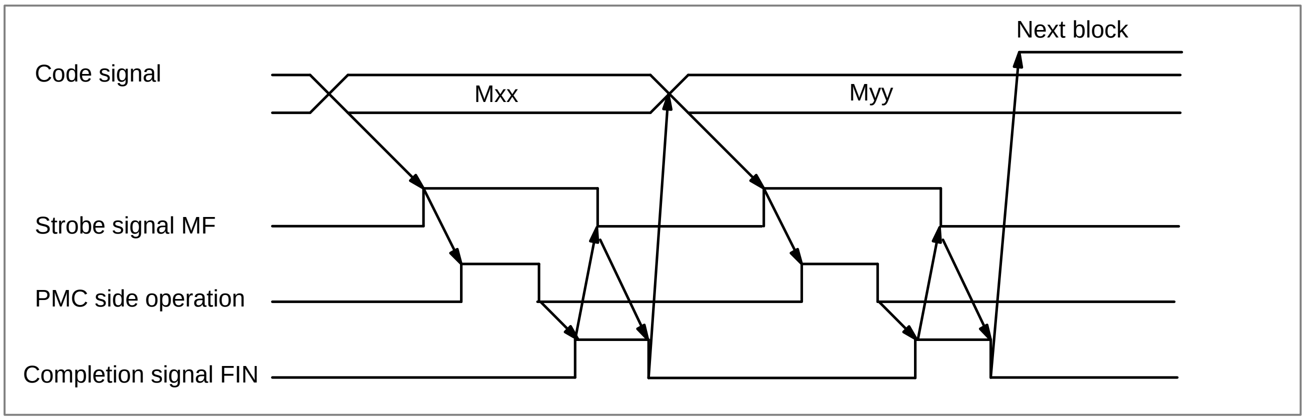


Fig. 8.4 (b) Timing chart of the usual system

A high–speed interface can also be used for multiple M commands issued for one block. This interface provides separate completion signals for each M code. They are called MFIN (the same name as for the single M command per block function), MFIN2, and MFIN3, respectively. The signal transfer sequence for multiple M codes per block is the same as that for a single M code per block.

The high–speed interface can also be used for the external operation function. In this case, special external operation signal EFD and completion signal EFIN are used. The procedure for sending and receiving these signals is identical to that for sending and receiving the strobe and completion signals of the miscellaneous function (M series).

Signal

Miscellaneous function completion signal MFIN <G005#0>

[Classification] Input signal

[Function] Reports that the execution of a miscellaneous function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and aprocedure of the contol unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

Spindle function completion signal SFIN <G005#2>

[Classification] Input signal

[Function] Reports that the execution of a spindle speed function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

Tool function completion signal TFIN <G005#3>

[Classification] Input signal

[Function] Reports that the execution of a tool function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

2nd auxiliary function completion signal BFIN <G005#4> (T series) <G005#7> (M series)

[Classification] Input signal

[Function] Reports that the execution of a second auxiliary function using the high–speed M/S/T/B interface is completed.

[Operation] For the operation and procedure of the control unit when this signal turns to “1” and “0”, see the description of “Basic procedure” above.

2nd, 3rd M function completion signals MFIN2, MFIN3 <G004#4, #5>

[Classification] Input signal

[Function] Indicate that when the high–speed interface is used for multiple M commands per block, the second to 3rd M functions have been completed.

[Operation] See “Basic procedure” for how the control unit operates and what it performs when the signal turns to “1” and “0”.

External operation signal for high–speed interface (M series) EFD <F007#1>

[Classification] Output signal

[Function] Indicates that positioning for the external operation function has been completed for the high–speed M, S, T, or B interface, and that another external operation is required.

[Operation] Refer to the description of the output conditions and procedure described in “basic procedure.”

External operation function completion signal (M series) EFIN <G005#1>

[Classification] Input signal

[Function] Indicates that the external operation function has been completed for the high–speed M, S, T, or B interface.

[Operation] The “basic procedure” describes the procedure and operation of the control unit when the signal is set to 1 or 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G004			MFIN3	MFIN2				
G005	BFIN			BFIN	TFIN	SFIN	EFIN	MFIN

	#7	#6	#5	#4	#3	#2	#1	#0
F007							EFD	

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3001	MHI							

MHI Exchange of strobe and completion signals for the M, S, T, and B codes

0 : Normal

1 : High–speed

Note

NOTE

1 The strobe signals MF, SF, TF, and BF are “0” when the power is turned on.

2 When the control unit is reset, MF, SF, TF, and BF are set to “0”.

Reference item

CONNECTION MANUAL (This manual)	8.1	Miscellaneous function/2nd auxiliary function
	8.3	Multiple M commands in a block
	11.8	External operation function

8.5 WAITING M CODE (TWO–PATH CONTROL)

General

Control based on M codes is used to cause one path to wait for the other during machining. By specifying an M code in a machining program for each path, the two paths can wait for each other at a specified block. When an M code for waiting is specified in a block for one path during automatic operation, the other path 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. 8110 and 8111) beforehand.

Signal

No–wait signal NOWT <G063#1>

[Classification] Input signal

[Function] Specifies whether to synchronize the paths 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 paths are synchronized by the M code. When the M code for waiting is specified for one path, the CNC waits for the corresponding M code of another path to be issued, then starts executing the next block.

Waiting signal WATO <F063#6>

[Classification] Output signal

[Function] Indicates that the CNC is waiting for the M code of either path 1 or 2.

[Output condition] This signal is “1” as long as:

- One path is waiting for another path. That is, the signal stays “1” for the period from when the M code for waiting is issued to one path to when the corresponding M code is issued to another path.

This signal is “0” as long as:

- Neither of the paths are waiting for the other.

Signal address

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

	#7	#6	#5	#4	#3	#2	#1	#0
F063		WATO #1						

Parameter

8110	Waiting M code range (minimum value)

[Data type] Two–word

[Valid data range] 0 and 100 to 99999999

This parameter specifies the minimum value of the waiting M code.

The waiting M code range is specified using parameter No. 8110 (minimum value) and parameter No. 8111 (maximum value).

(parameter No. 8110) ≤ (waiting M code) ≤ (parameter No. 8111)

> **NOTE**
> A value of 0 indicates that the waiting M code is not used.

8111	Waiting M code range (maximum value)

[Data type] Two–word

[Valid data range] 0 and 100 to 99999999

This parameter specifies the maximum value of the waiting M code.

Alarm and message

Number	Message	Description
160 (T series) 5096 (M series)	MISMATCH WAITING M–CODE	Different M code is commanded in heads 1 and 2 as waiting M code. Modify the program.

Caution

> **CAUTION**
> As for waiting M code, neither code signals nor strobe signal are output.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.21.2	WAITING FOR PATHS
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.20.2	WAITING FOR TOOL POSTS

8.6 M CODE GROUP CHECK FUNCTION

General

This function checks whether combinations of M codes (up to three) specified in one block are correct.

The function has two purposes. One of the purposes is to alarm if an M code which must not be combined with any other M codes is combined with another. The other purpose is to alarm if an M code in a group is combined with another M code in the same group. These errors are reflected in P/S alarm No. 5016.

• Setting the group data

Pressing the SYSTEM key, then the continuous menu key several times causes the [M CODE] soft key to appear. Pressing this soft key displays the screen shown in Fig. 8.6 (a).

```
M CODE GROUP SETTING                  O0000   N0000
NO.     MCG     NO.     MCG     NO.     MCG
0000     0      0010     0      0020     0
0001     0      0011     0      0021     0
0002     0      0012     0      0022     0
0003     0      0013     0      0023     0
0004     0      0014     0      0024     0
0005     0      0015     0      0025     0
0006     0      0016     0      0026     0
0007     0      0017     0      0027     0
0008     0      0018     0      0028     0
0009     0      0019     0      0029     0
>

 MDI   ****   ***   ***        00 : 00 : 00
[        ][        ][M CODE][        ][(OPRT)]
```

Fig. 8.6 (a)

Basically, item numbers correspond to M codes. However, there may be exceptions depending on parameter setting. (See descriptions of the relevant parameters.) The data specified on this screen remains until all parameters are cleared simultaneously. To go to the next page of the screen, use the [PAGE] key.

Pressing the [(OPRT)] soft key or the corresponding numeric key displays the soft keys shown in Fig. 8.6 (b). Searches for the desired item number and enter the data.

Semicolon “;” may be used for continuous data entry.

Data entry is enabled when PARAMETER WRITE ENABLE is set to “1” on the parameter setting screen.

```
>
MDI
      ****   ***   ***        00 : 00 : 00
[No. SRH][        ][        ][        ][ INPUT ]
```

Fig. 8.6 (b)

For M codes which must be used separately from other M codes, always set their group number to “1”. Such M codes include M00, M01, M02, M30, M98, and M99. For M codes for which the CNC performs internal processing in addition to sending them to the machine, also set their group number to “1”. See Section 8.3 for details.

For M codes you do not need to check, leave them at an initial value of “0”.

The M codes can be set with a number from 0 to 127. Neither negative values nor decimal point can be specified.

- **Parameter setting**

Basically, item numbers 0 to 99 correspond to M00 to M99. For item numbers 100 and higher, parameter No. 3441 to 3444 can specify the corresponding M codes.

The initial values for these parameters are “0”.

3441	First of the M codes assigned to item numbers 100 to 199
3442	First of the M codes assigned to item numbers 200 to 299
3443	First of the M codes assigned to item numbers 300 to 399
3444	First of the M codes assigned to item numbers 400 to 499

[Data type] Two–word

[Valid data range] 0 to 99999999

If a parameter is specified as “0”, it is set to a value specified to the previous parameter plus 100. For example, if parameter No. 3441 is specified as “0”, it is internally set to “100”. If parameter No. 3442 is specified as “0” under this condition, it is internally set to “200”.

Negative values are assumed to be “0”.

Basically, these parameters can be specified as any value. However, the following conditions should be observed to save memory space.

$X + 99 < Y$, $Y + 99 < Z$, and $Z + 99 < W$

where X = value specified for parameter No. 3441, Y = value specified for parameter No. 3442, Z = value specified for parameter No. 3443, and W = value specified for parameter No. 3444.

(Example of setting)

(i) No. 3441 = 0, No. 3442 = 0, No. 3443 = 0, No. 3444 = 0

In this case, these parameters are set to “100”, “200”, “300”, and “400” respectively. The item numbers correspond directly to the M codes. The group numbers for M500 and larger M codes are assumed to be “0” automatically.

(ii) No. 3441 = 200, No. 3442 = 0, No. 3443 = 500, No. 3444 = 800

In this case, parameter No. 3442 is set to “300”. Item numbers 100 to 299 correspond to M200 to M399, 300 to 399 correspond to M500 to M599, and 400 to 499 correspond to M800 to M899. The group numbers for M100 to M199, M400 to M499, M600 to M799, M900, and all M codes with higher numbers are automatically set to “0”.

(iii) No. 3441 = 234, No. 3442 = 345, No. 3443 = 456, No. 3444 = 567

In this case, item numbers 100 to 199 correspond to M234 to M333, 200 to 299 correspond to M345 to M444, 300 to 399 correspond to M456 to M555, and 400 to 499 correspond to M567 to M666. The group numbers for M100 to M233, M334 to M344, M446 to M455, M556 to M566, M667, and all M codes with higher numbers are automatically set to “0”.

The examples above meet the setting conditions. With these settings, up to 500 M codes can be set.

Examples that do not meet the setting conditions follow.

(iv) No. 3441 = 200, No. 3442 = 50, No. 3443 = 100, No. 3444 = 600

In this case, item numbers 0 to 99 correspond to M00 to M99, 100 to 199 correspond to M200 to M299, and 400 to 499 correspond to M600 to M699. Item numbers 200 to 399 are meaningless. With these settings, only up to 300 M codes can be set.

(v) No. 3441 = 50, No. 3442 = 100, No. 3443 = 150, No. 3444 = 200

In this case, the correspondence between the item numbers and M codes is set up as listed below. With these settings, up to 300 M codes can be set. The group number for M300 and all M codes with higher numbers are automatically set to “0”.

Item number	M code
0 – 49	0 – 49
50 – 99	Meaningless
100 –199	50 – 149
200 –249	Meaningless
250 –299	150 – 199
300 –349	Meaningless
350 –399	200 – 249
400 –449	Meaningless
450 –499	250 – 299

- **Input/output with floppy cassettes**
- **Input**

The file you want to read out should be set on the read station. First locate the file in the program screen in the EDIT mode. In this mode, display the M code group setting screen. See Section 8.2, Part III of the operator’s manual for how to locate the file.

On the soft key screen shown in Fig. 8.6 (b), pressing the continuous menu key several times displays the soft keys shown in Fig. 8.6 (c).

```
>
MDI    ****  ***  ***       00 : 00 : 00
(        )( READ )( PUNCH )(        )(        )
```

Fig. 8.6 (c)

Now pressing the [READ] key displays the soft keys shown in Fig. 8.6 (d).

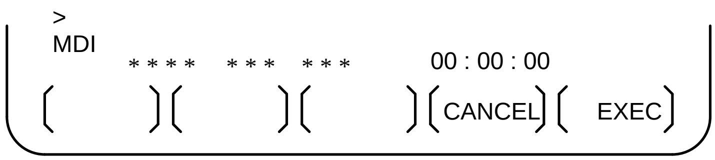


Fig. 8.6 (d)

To execute the read operation, just press the [EXEC] key.

- **Output**

Pressing the [PUNCH] key on the screen shown in Fig. 8.6 (c) displays the soft keys shown in Fig. 8.6 (d). To execute the punch operation, just press the [EXEC] key.

After this operation, you can confirm that the file M CODE GROUP is output, by searching through floppy cassette files. The output data has a format with 60000 added to the item number:

N60xxxPyyy (where xxx = 0 to 499, yyy = 0 to 127)

Parameter

3441	First of the M codes assigned to item numbers 100 to 199
3442	First of the M codes assigned to item numbers 200 to 299
3443	First of the M codes assigned to item numbers 300 to 399
3444	First of the M codes assigned to item numbers 400 to 499

[Data type] Two–word

[Valid data range] 0 to 99999999

Alarm and message

Number	Message	Description
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.11.3	M CODE GROUP CHECK FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.11.3	M CODE GROUP CHECK FUNCTION

9 SPINDLE SPEED FUNCTION

9.1 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

General

When up to five digits are specified after address S, code and strobe signals are sent out and used to control the spindle speed. The code signals are retained until another S code is issued.

One S code is used for each block. Parameter No. 3031 can be used to specify the maximum number of digits. If a number greater than the maximum number is specified, an alarm can be raised.

Signal

Refer to section 8.1.

Parameter

3031	Allowable number of digits for the S code

[Data type] Byte

[Valid data range] 1 to 5

Set the allowable numbers of digits for the S codes.

Note

NOTE

1 When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:
   i) Simultaneous execution of the move command and miscellaneous function commands.
   ii) Executing miscellaneous function commands upon completion of move command execution.

   The selection of either sequence depends on the PMC processing sequence.

2 For S code output when the spindle serial output/spindle analog output is used, refer to section 9.3.

9.2 SPINDLE SERIAL OUTPUT/SPINDLE ANALOG OUTPUT

General

There are two types of spindle motor control interfaces, spindle serial output and spindle analog output.

The spindle serial output interface can control two serial spindles. The spindle analog output interface can control one analog spindle.

The table below lists the relationships between the spindle control interfaces and the configuration of the spindle.

Spindle serial output	Spindle analog output	First spindle	Second spindle	Third spindle
○	○	First serial spindle The PC can be used.	Second serial spindle The PC can be used. (*)	Analog spindle The PC cannot be used.
○	×	First serial spindle The PC can be used.	Second serial spindle The PC can be used. (*)	—
×	○	Analog spindle The PC can be used.	—	—
×	×	See Section 9.1. ⇒ Controlled by the PMC using an external interface.		

· PC = position coder

· (*) The multispindle function is necessary to use the position coder of the second spindle.

· 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 revolution (synchronous feed)	○	○ (*1)	○	×
Constant surface speed control	○	○ (*1)	○	×
Spindle speed fluctuation detection	○	○ (*1)	○	×
Actual spindle speed output (T series only)	○	○ (*1)	○	×
Spindle positioning (T series only)	○	×	○	×
Cs contour control	○	×	×	×
Multi–spindle (*2)	○ (First spindle)	○ (Second spindle)	×	○ (Third spindle)
Rigid tapping	○	○ (*1)	○	×
Spindle synchronous control	○ Master (*3)	○ Slave (*3)	×	×
Spindle control unit functions (*4), such as spindle orientation, spindle output switching, spindle switching, and etc.	○	○	○	○
Polygon turning (T series only) (using the servo motor axis and spindle)	○	○ (*1)	○	×
Polygon turning between spindles (T series only) (using two spindles)	○ Master (*5)	○ Slave (*5)	×	×
Spindle output control by the PMC	○	○	○	○

NOTE

1 The multispindle function is necessary. The function cannot be used for the first and second spindles simultaneously.
2. The multispindle function can control the speed of three spindles and switch the feedback signal between two position coders. It also can work without the second or third spindle.
3. For a two–path lathe application, 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. These functions belong to the spindle control unit. They cannot be used unless the spindle control unit supports those functions.
5. No spindle polygon turning is available for a combination of the spindle of tool post 1 and the spindle of tool post 2 for the two–path lathe application.

The signals and parameters for spindle speed control are common to both spindle serial output and spindle analog output. (See Section 9.3.)

The table below lists the differences related to direct control of the spindle control unit.

	Spindle control unit for spindle serial output interface	Spindle control unit for spindle analog output interface
Parameters for the spindle control unit	Specified as CNC parameters (4000 to 4351/S1, S2) Used after being transferred to the spindle control unit	Directly specified for the spindle control unit
Control signal for the spindle control unit	Connected to the PMC via the CNC G0070 to G0073 and F0045 to F0048: Addresses for the first spindle G0074 to G0077 and F0049 to F0052: Addresses for the second spindle	Connected to the PMC via an external contact
Spindle speed command interface	Digital data in a range from 0 to ± maximum spindle motor speed	Analog voltage from 0 to ± 10 V (excluding portion for offset voltage adjustment)
Position coder interface	Connected to the CNC via the spindle control unit	Connected directly to the CNC

Signal

· Spindle control unit signals for the serial spindle

<G0070 to G0073> (input), <F0045 to F0048> (output)
→ for the first serial spindle
<G0074 to G0077> (input), <F0049 to F0052> (output)
→ for the second serial spindle

These addresses are on the CNC. Actually, however, they are input/output signals for the spindle control unit for the serial spindle.

For details of the signals belonging to these addresses, refer to the manuals for the serial spindle:

FANUC CONTROL MOTOR AMPLIFIER α series Descriptions (B–65162E)
FANUC AC SPINDLE MOTOR α series Descriptions (B–65152E)
FANUC CONTROL MOTOR α series Maintenance Manual (B–65165E)

Signal address

- **For 1st SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA		SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072	RCHHGA	MFNHGA	INCMDA	OVRIDA	DEFMDA	NRROA	ROTAA	INDXA
G073						MPOFA	SLVA	MORCMA

	#7	#6	#5	#4	#3	#2	#1	#0
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
F047							INCSTA	PC1DEA
F048								

- **For 2ND SERIAL SPINDLE**

	#7	#6	#5	#4	#3	#2	#1	#0
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1A	CTH2B	TLMHB	TLMLB
G075	RCHB	RSLB		SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076	RCHHGB	MFNHGB	INCMDB	OVRIDB	DEFMDB	NRROB	ROTAB	INDXB
G077						MPOFB	SLVB	MORCMB

	#7	#6	#5	#4	#3	#2	#1	#0
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB
F050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
F051							INCSTB	PC1DEB
F052								

Parameter

- **Connection of serial spindle control unit**

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2			ISI	

[Data type] Bit

ISI Specifieds whether the serial spindle interface is used.

0 : Used
1 : Not used

> **NOTE**
> 1 This parameter is enabled only when the serial spindle interface option is provided. The parameter is used when the CNC is started after serial spindle interface control is temporarily disabled during startup adjustment of the CNC. This bit should normally set to be 0.
> 2 When Serial spindle is used with Analog spindle and this parameter is set to “1”, Analog spindle becomes 1st spindle.

SS2 The number of connections in serial spindle control

0 : 1
1 : 2

> **NOTE**
> To connect two serial spindles, set jumper S1 on the 1st serial spindle control unit to B.
> (For S series SERIAL SPINDLE AMPLIFIER)

- **Parameters of serial spindle control unit**

No. 4000 to 4351: S1 → For 1st serieal spindle
S2 → For 2nd serial spindle

The above parameters are on the CNC, but actually they are used for the spindle control unit of serial spindle.
For details of these parameters, refer to the following manual:

FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)

FANUC CONTROL MOTOR α series MAINTENANCE MANUAL (B–65165E)

Alarm and message

Number	Message	Description
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be noises, disconnection of an optical cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated.
750	SPINDLE SERIAL LINK ERROR	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 SP2, bit 4 of parameter No. 3701, 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. See diagnostic display No. 409 for details.
751	SPINDLE–1 ALARM DETECT (AL–XX)	This alarm indicates the NC that an alarm is generated in the spindle control unit of the serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to the manuals for serial spindle. The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.

Number	Message	Description
752	SPINDLE–1 MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the CNC.
761	SPINDLE–2 ALARM DETECTION (AL–XX)	Refer to alarm No. 751. (For 2nd spindle)
762	SPINDLE–2 MODE CHANGE ERROR	Refer to alarm No. 752.(For 2nd spindle)

DIAGNOSIS SCREEN

- **Information on spindle control**

	#7	#6	#5	#4	#3	#2	#1	#0
400				SAI	SS2	SSR	POS	SIC

SIC 0: No module is available for spindle serial output.
1: A module for spindle serial output is available.

POS 0: No module is available for spindle analog output.
1: A module for spindle analog output is available.

SSR 0: Spindle serial output is not used.
1: Spindle serial output is used.

SS2 0: The second spindle is not used with spindle serial output.
1: The second spindle is used with spindle serial output.

SAI 0: Spindle analog output is not used.
1: Spindle analog output is used.

401	Alarm condition for the serial spindle unit for the first spindle (AL–??)
402	Alarm condition for the serial spindle unit for the second spindle (AL–??)

- **Communication error on spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
408	SSA		SCA	CME	CER	SNE	FRE	CRE

CRE 1 : CRC error (warning)

FRE 1 : Framing error (warning)

SNE 1 : Mismatch between sending and receiving sections

CER 1 : Abnormal reception

CME 1 : No answer during auto scanning

SCA 1 : Communication error in the spindle amplifier

SSA 1 : System error in the spindle amplifier

(These errors are reflected in spindle alarm 749. They are caused by noise, disconnection, or instantaneous power interruption.)

- **Information related to the activation of the spindle serial output interface**

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SHE 1 : Abnormal operation in the serial spindle communication module of the CNC

S1E 1 : Abnormal operation on the first spindle during activation

S2E 1 : Abnormal operation on the second spindle during activation

SPE 1 : Serial spindle parameter not meeting activation conditions

(These errors are reflected in spindle alarm 750.)

- **Load and speed meter readings for the serial spindle**

410	First serial spindle: Load meter reading (%)
411	First serial spindle: Speed meter reading (rpm)
412	Second serial spindle: Load meter reading (%)
413	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 No. 4020 (main) and 4196 (sub)

Load meter reading at maximum output:
Parameter No. 4127 (main) and 4276 (sub)

> **NOTE**
> The spindle switch function is used for main/sub switching. Select main if the spindle switch function is not used.

- **Position error display during spindle synchronization**

414	Master spindle motion error during spindle synchronization
415	Slave spindle motion error during spindle synchronization
416	Absolute value of synchronization error during spindle synchronization

The display for diagnosis No. 414 to 416 are in pulse units (one pulse = 360/4096 degrees)

- **Position error display during spindle synchronization**

417	First serial spindle: Position coder feedback information
418	First serial spindle: Position error
419	Second serial spindle: Position coder feedback information
420	Second serial spindle: Position error

The above display data is the information obtained directly from the serial spindle control unit.

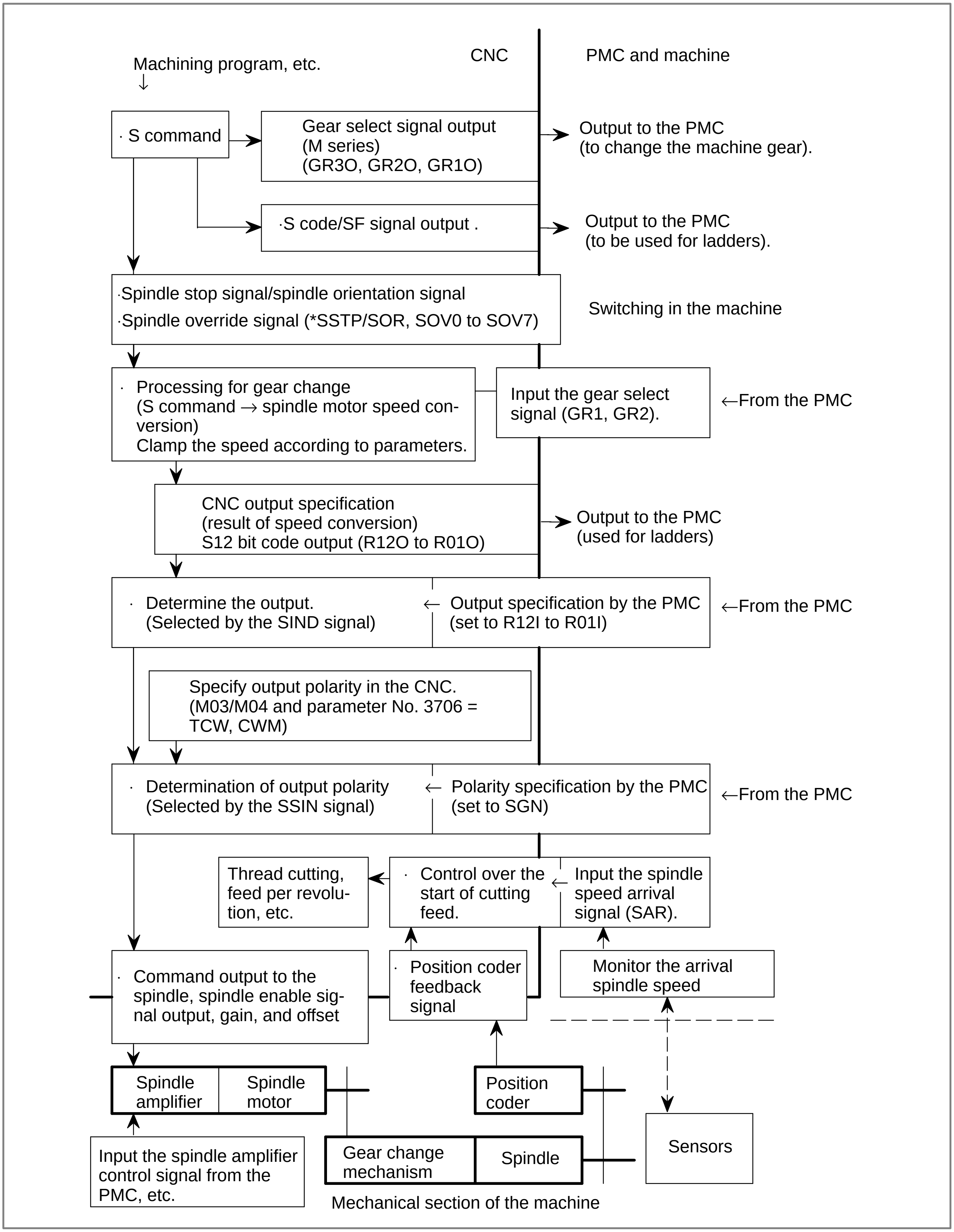
9.3 SPINDLE SPEED CONTROL

General

This section describes spindle speed control. It also explains the position coder and the spindle speed arrival signal (SAR).

Command flow of spindle speed control

The following chart summarizes spindle speed control.



- **S command**

The S command specifies the spindle speed entered from machining programs, etc. for the CNC.

For constant surface speed control (during G96 mode), the CNC converts the specified surface speed to the spindle speed.

In the M series with bit 4 (GTT) of parameter No. 3706 = 0 without the constant surface speed control option, the CNC specifies the gear stage for the desired spindle speed to the PMC according to parameter No. 3741, 3742, and 3743, and the S command.

(GR3O, GR2O, GR1O <F034#2, #1, #0>)

- **S code/SF signal output**

With the spindle serial output or spindle analog output option, the spindle control function in the CNC converts the S command value to the output value for the spindle motor. For correspondence to gear change and constant surface speed control, the S code/SF signals output is different as follows in case of the spindle serial output and spindle analog output are not used.

M series → Outputs the S code.
The SF signal is output only when the CNC directs the PMC to change the gear.

T series → Outputs neither S code nor SF signal.
(This is because the S code is not always the spindle speed when the constant surface speed control option is used.)

If you use the S code for processing in the PMC ladder, you must specify parameters related to parameter No. 3705.

- **Spindle stop signal (*SSTP)**

This signal sets the S command value in the CNC to 0. If the CNC 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 (ORM) of parameter No. 3706 at a constant speed specified by parameter No. 3732.

Because the spindle rotates at a constant speed regardless of the gear stage, this signal can be used to rotate the spindle to drive the stopper or pin during mechanical spindle positioning.

In the M series, specifying bit 1 (GST) of parameter No. 3705 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 (SOV00 to SOV07)**

This signal specifies an override of 0% to 254% for the specified S value for spindle control.

However, the spindle speed override function is disabled when the CNC is in the following state:

Tapping cycle (M series : G84, G74 T series : G84, G88)
Thread cutting (M series : G33 T series : G32, G92, and G76)

When the spindle speed control is performed but the spindle speed override is not used, set the override value to 100%.

- **Processing for gear changing**

Although the S command contains the spindle speed, the object that is actually controlled is the spindle motor. Therefore, the CNC must have some provision to detect the gear stage between the speed and spindle motor.

There are two types of gear selection methods:

<u>M type</u>

The CNC selects a gear stage according to the range of speed for each gear stage previously specified in a parameter, as directed by the S command, and informs the PMC of the selected gear stage (one of the three gear stages) using the gear select signal output (GR3O, GR2O, GR1O).

Also, the CNC outputs the spindle speed based on the selected gear stage (output as the gear select signal).

<u>T type</u>

The gear stage (one of the four gear stages) being currently used by the machine is 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, and bit 4 (GTT) of parameter No. 3706 = 0

T type ← With constant surface speed control, or bit 4 (GTT) of parameter No. 3706 = 1

The T series system can use only T type.

- **Details of M type (Output of GR1O, GR2O, GR3O)**

By specifying from S0 to S99999 in memory or MDI operation, the CNC outputs a command corresponding to the spindle speed. There is a two-speed (GR1O and GR2O) or three-speed range (GR1O, GR2O, GR3O), set by parameter nos. 3741-3743, and the gear selection signal is output simultaneously. When the gear selection signal is changed, the SF signal is output at the same time (parameter SFA no. 3705#6).
The meaning of the gear signals is shown below:

	Gear 2–stage	Gear 3–state	Remarks
GR1O	Low	Low	Low: Low Gear Middle: Middle Gear High: High Gear
GR2O	High	Middle	
GR3O		High	

The speed commands output to the spindle motor are as follows:

· For the serial spindle, the speed commands are processed as values 0 to 16383 between the CNC and spindle control unit.

· For the analog spindle, the speed commands are output to the analog voltage signal SVC as analog voltages 0 to 10 V.

The following descriptions exemplify the analog spindle. However, they can be applied to the serial spindle on the assumption that spindle motor speed with analog voltage 10 V corresponds to the maximum spindle motor speed.

- M type gear change method A (Fig. 9.3 (a))

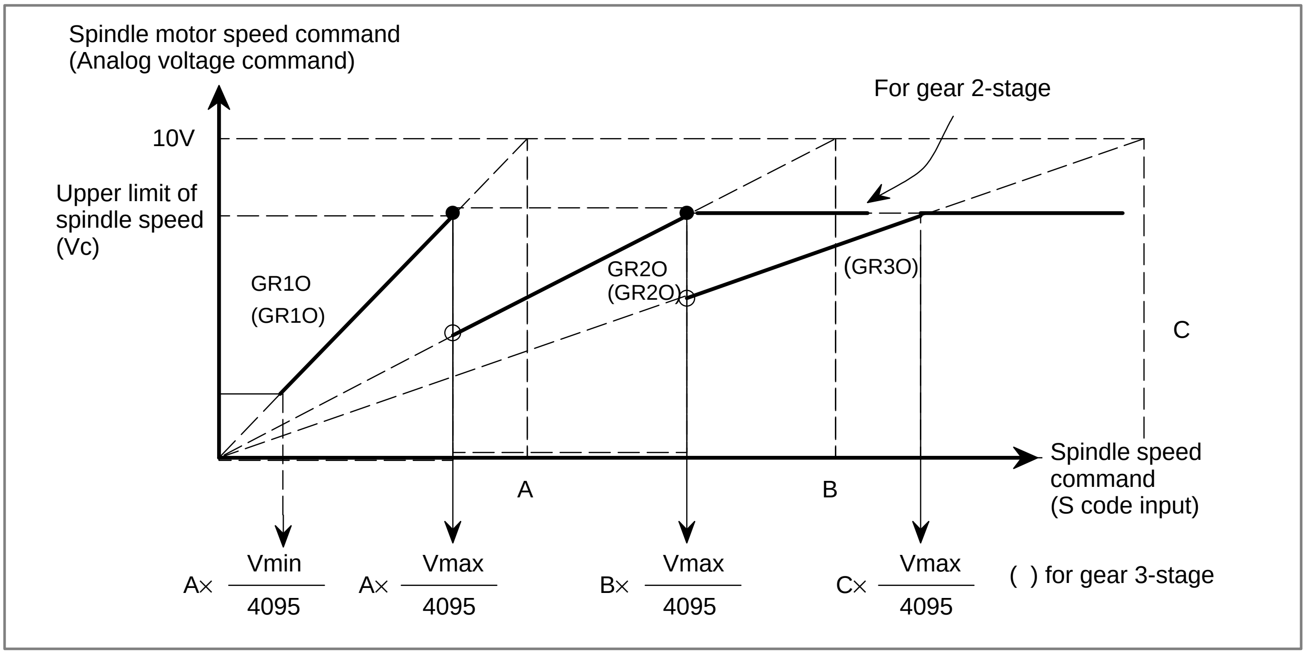


Fig. 9.3 (a) S code input and output voltage

Set the following values as parameters:

· Constant Vmax: for upper limit of spindle speed (parameter No. 3736)

$$Vmax=4095\times\frac{\text{Upper limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Constant Vmin; for lower limit of spindle speed (parameter No. 3735)

$$Vmin=4095\times\frac{\text{Lower limit of spindle speed}}{\text{Spindle speed at command voltage 10V}}$$

· Spindle speed A (rpm) ; at command voltage 10V and low gear (parameter no. 3741)

· Spindle speed B (rpm) ; at command voltage 10V and high gear (or middle-high gear) (parameter no. 3742)

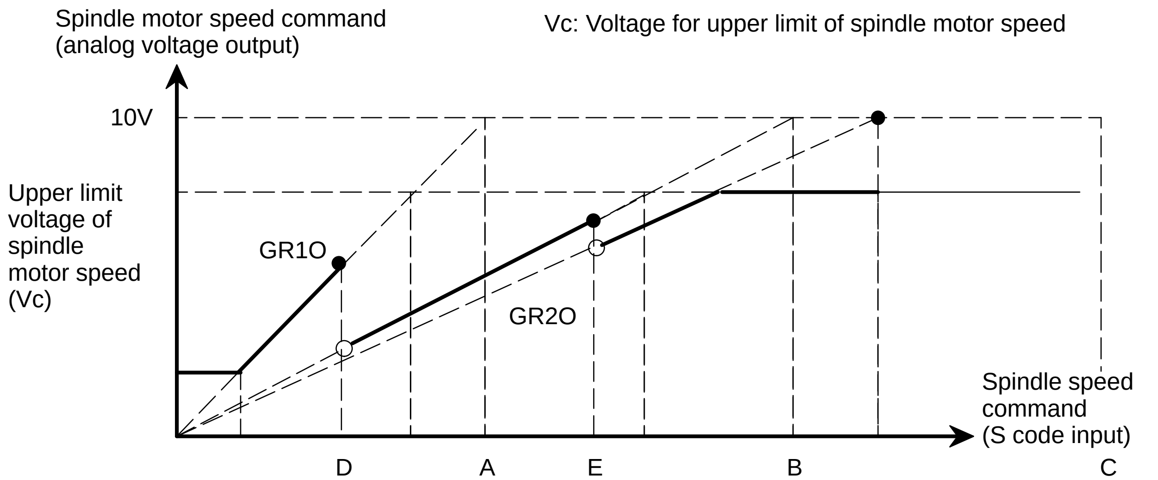
· Spindle speed C (rpm) ; at command voltage 10V and high gear (parameter no. 3743)

NOTE

If a specified voltage of 10 V is already higher than the acceptable input voltage for the spindle drive system, calculate the spindle speed that corresponds to 10 V using a proportional calculation method and use it instead. Now, in response to the specified S code, the speed command and gear select commands (G3O, G2O, G1O) are output to the spindle motor as shown in Fig. 9.3. (a).

- Gear change point during tapping cycle mode (G84, G74)

  In case of G84 (tapping cycle) or G74 (counter tapping cycle) the gear shift speed is changed by parameter SGT(No. 3705#3). In this case, gear shift is performed at the speed set by parameter nos. 3761 and 3762 (Fig. 9.3 (b)).



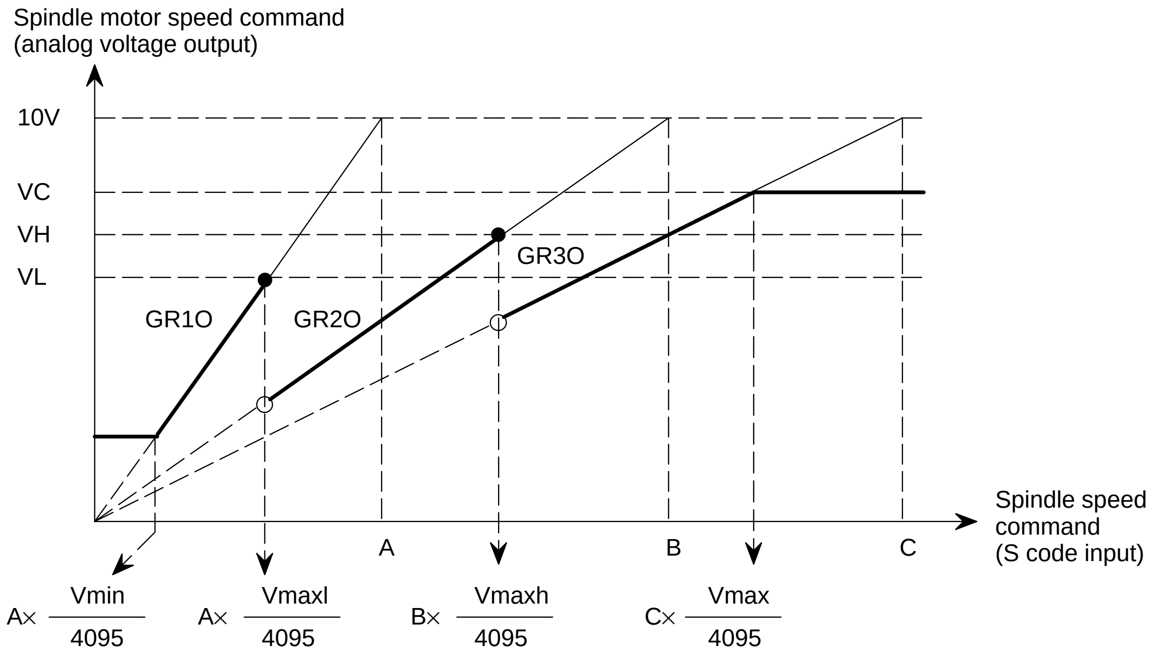
D : Gear shift point (parameter No. 3761) for low/high speed change (or low/medium speed change) (set by parameter as rpm value) during tapping.

E : Gear shift point (parameter No. 3762) (3-stage gear) for medium/high speed change (set by parameter as rpm value) during tapping.

Fig. 9.3 (b)   S code input and output voltage (in tapping)

- M type gear change method B (M series) (Fig. 9.3 (c))

The speed (rpm) at which the low-speed and the high-speed gears are changed over can be set as a parameter (No.3751, 3752) by setting parameter SGB (No. 3705# 2). When a 3-stage gear is used, it is possible to set the speeds (rpm) for switching low-speed and medium-speed gears, and medium-speed and high-speed gears, using parameters No. 3751, 3752.



VC: Voltage corresponding to the upper limit of output value to spindle motor.

VH: Voltage corresponding to the upper limit of output value to spindle motor with high-speed gears (medium speed gear for 3-stage gear)

VL: Voltage corresponding to the upper limit of output value to spindle motor at low-speed gear

Fig. 9.3 (c) M type gear change B

When using this function, set the following parameters:

· Constant Vmax (Parameter No.3736) related to the upper limit of spindle motor speed (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. 3735) 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. 3751) 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. 3752) related to the upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-stage gear)

$$\text{Vmaxh} = 4095\times\frac{\text{Upper limit of spindle motor speed (rpm) with high-speed gears (medium-speed gear for 3-stage gear)}}{\text{Spindle motor speed (rpm) when the command voltage is 10V}}$$

· Spindle speed A (Parameter No.3741) (rpm) with low-speed gears when the command voltage is 10V

· Spindle speed B (Parameter No.3742) (rpm) with high-speed gears when the command voltage is 10V (medium-speed gear for 3-stage)

· Spindle speed C (Parameter No.3743) (rpm) with high-speed gears when the command voltage is 10V (3-stage gear)

Spindle motor speed commands (0 to 10V) and gear selecting signals (GR1O, GR2O, GR3O) are issued on each S code command as shown in the figure:

CAUTION
1 In a tapping cycle when parameter SGT (No. 3705 #3) is set, the gears are changed over at the gear changing point for tapping.
2 For this function (parameter SGB=1 (No. 3705#2)), when only one-stage gear is used, the voltage corresponding to the upper limit value to the spindle motor is calculated using Vmaxl, and when 2-stage gear is used, it is calculated according to Vmaxh. Therefore, when SGB is 1, set Vmaxl when only one-stage gear is used, Vmaxl and Vmaxh when 2-stage gear is used.

- **Time chart**

When S code is commanded, the I/O signal time chart is :

- When Gear select signal does not change

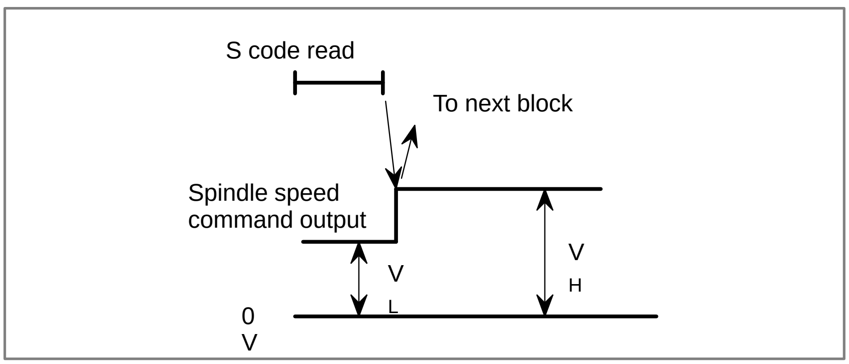


Fig. 9.3 (d) Time chart when gear select signal does not change

In this case, the SF signal is not output and the CNC advances to the next block automatically after the next spindle speed command is output.

- When Gear select signal change

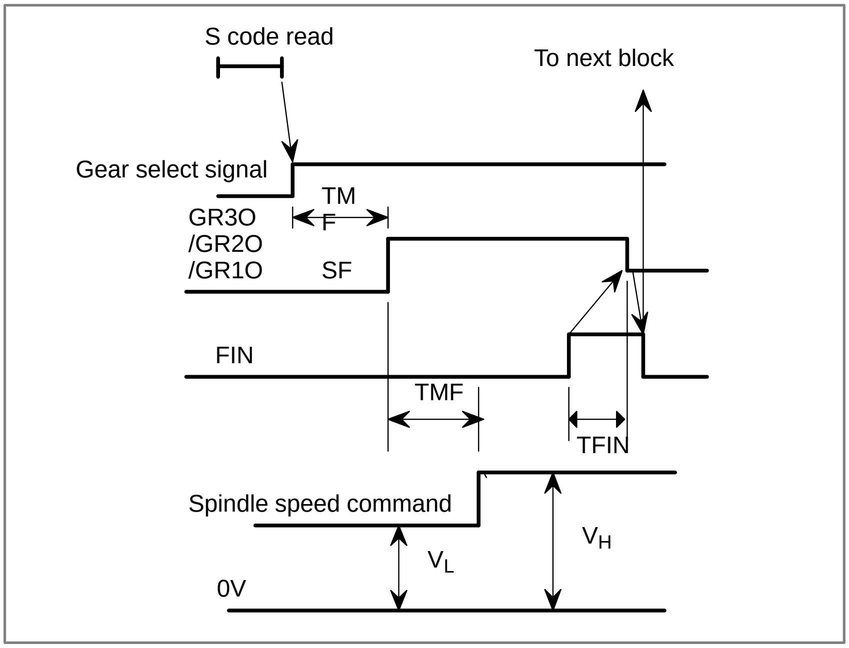


Fig. 9.3 (e) Time chart when gear select signal changes

In this case, the gear select signal is output; after elapse of the time constant set by parameter (TMF), the SF signal is output. After another TMF elapse, the spindle speed command is output. On the PMC side, change the gears by this signal, and return the FIN signal after the end of gear change. The time chart for SF and FIN signals is the same as in S code output.TMF, set by parameter No. 3010, is common to M, S and T functions.

Moreover, specifying bit 6 (SFA) of parameter No. 3705 can specify that the SF signal be output even if no gear change is used.

- **Details of T type (Input of GR1, GR2)**

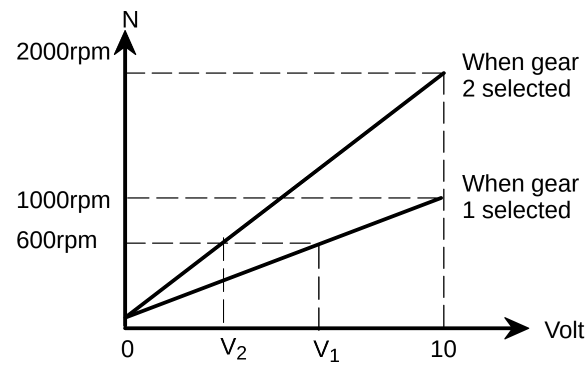
To perform the T type gear changing, the maximum spindle speed for each gear select signal issued from the PMC side must be set by parameter No. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear No.	Parameter No. for max. spindle speed
0	0	1	No. 3741
1	0	2	No. 3742
0	1	3	No. 3743
1	1	4	No. 3744

The following descriptions 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 No. 3741, 3742. In this case, the analog voltage has the linear relationship shown below.



When spindle speed S=600 is given, V_1 (for G1) or V_2 (for G2) is calculated inside the CNC and output to the machine side.

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

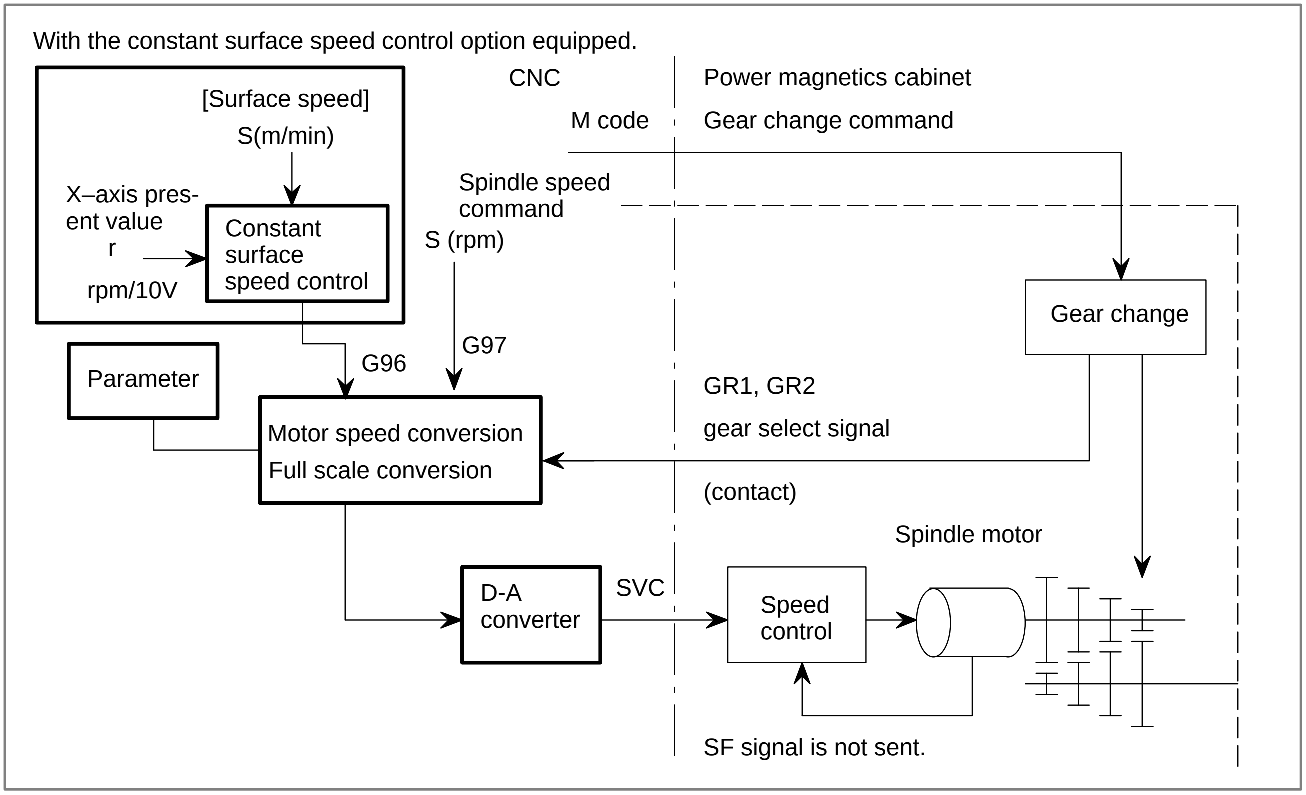
N: Spindle speed given by S5-digits

This is equivalent to the G97 mode for constant surface speed control.

See Section 9.5 for operations during the constant surface speed control mode (G96).

In addition, parameter No. 3772 (upper limit to the spindle speed) can specify speed clamping for all gear positions.

Reference→ Block Diagram for Analog Voltage Output



• Determination of output R12O–R01O (Output) R12I–R01I (Input) SIND (Input)

Using the above processing for gear change, the CNC calculates the speed command output to the spindle motor that is necessary to obtain the specified spindle speed with the gear.

For either serial spindle or analog spindle control, the calculation results are output as the S12 bit code signal from 0 to 4095 to the PMC.

(R12O to R01O<F037#3 to F036#0>)

After the calculation results are received, the SIND signal <G033#7> determines which is to be used, the speed command output calculated by the CNC or the data specified in the PMC. Thus 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. 3706 = 0
→ Determined according to bit 6 (CWM) of parameter No. 3706

· If bit 7 (TCW) of parameter No. 3706 = 1
→ Determined according to bit 6 (CWM) of parameter No. 3706 and M03/M04 given to the CNC

After that, the SSIN signal <G033#6> determines which is to be used, the output polarity calculated by the CNC or the polarity specified in the PMC. In this way, the 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. 3706 = 1, the CNC cannot determine the output polarity if it has not issued M03/M04, and therefore, actual output does not work even if the speed command 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. 3706 = 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 <F001#4>**

Another output related to spindle control is the spindle enable signal ENB.

The ENB signal is logical 1 when a nonzero command output is sent to the spindle. If the command is logical 0, the ENB signal becomes logical 0.

When the analog spindle is being used, an offset voltage in the spindle motor speed amplifier may cause the spindle motor to rotate at low speed even if the command output (in this case, analog voltage) to the spindle is zero. The ENB signal can be used to stop the motor in such a case.

- **Gain and offset**

The analog spindle may require gain and offset voltage adjustment depending on the spindle motor speed amplifier being used.

The following parameters are available for such adjustment.

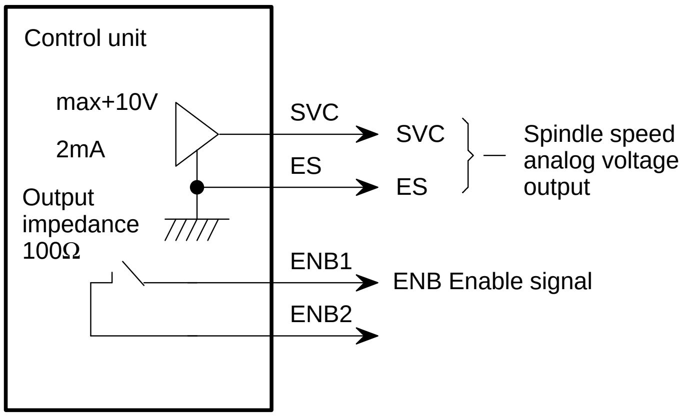
· Analog spindle as the first spindle
  → Gain adjustment data: Parameter No. 3730
    Offset voltage compensation: Parameter No. 3731

· Analog spindle as the third spindle
  → Gain adjustment data: Parameter No. 3820
    (valid for multispindle control)
    Offset voltage compensation: Parameter No. 3821

- **Electrical specification of analog spindle interface**

The signals related to analog spindle interface are described below.

The ENB1 and ENB2 signals are turned on and off under the same condition as for the ENB signal <F001#4>. They can be used also for the serial spindle.



Name	Connector/Pin
SVC	JA40/7
ES	JA40/5
ENB1	JA40/8
ENB2	JA40/9

> **WARNING**
> Since the output voltage is a weak signal, do not relay it through contacts.

- **Position coder feedback signal**

The position coder is necessary for thread cutting or feed per revolution. (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 1 and 0 (PG2, PG1) of parameter No. 3706.

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. 3740 after the S command or cutting feed command was issued.

When using the SAR signal, set bit 0 (SAR) of parameter No. 3708 to 1.

Item No. 06 (SPINDLE SPEED ARRIVAL CHECK) on the diagnosis screen is kept at 1 while this function is keeping the cutting feed block at a halt.

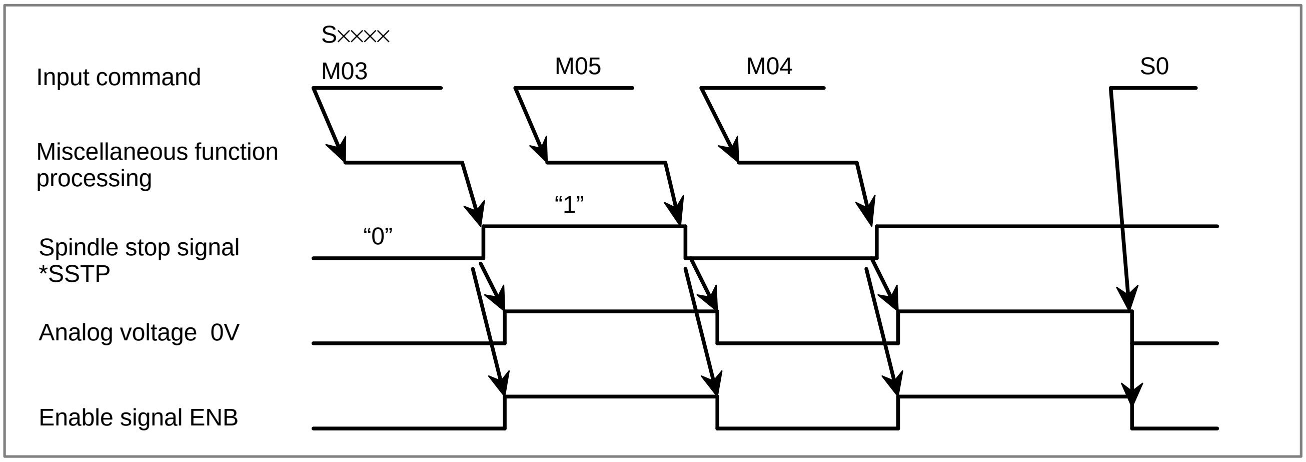
Signal

Spindle stop signal *SSTP<G029#6>

[Classification] Input signal

[Function] The command output to the spindle is 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 <G029#5>

[Classification] Input signal

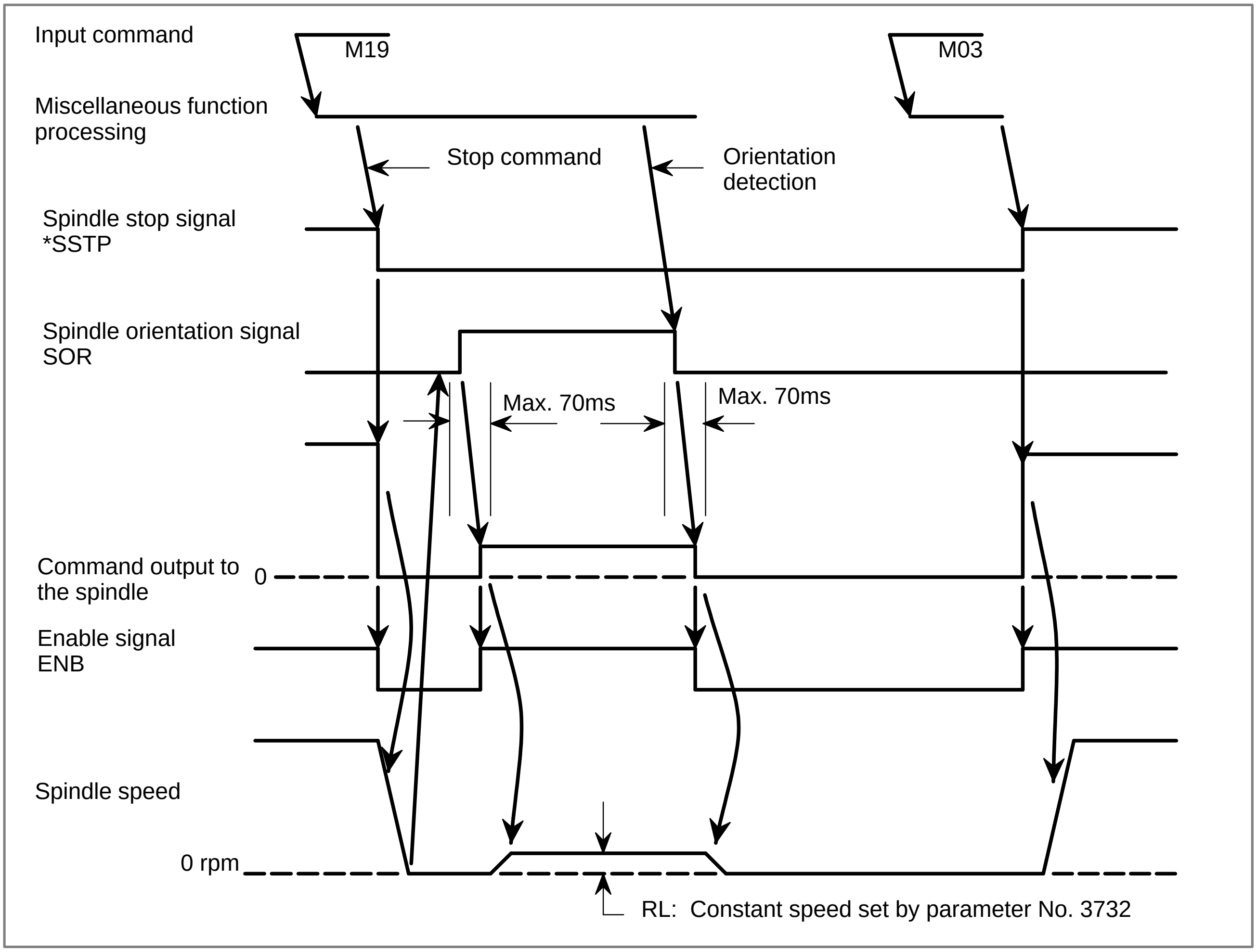
[Function] The spindle or the spindle motor is rotated at a constant speed.

[Operation] When the spindle orientation signal turns to “1” and the spindle stop signal *SSTP turns to “0”, a spindle speed command which lets the spindle rotate at the constant speed set by parameter No. 3732 is output. The enable signal ENB also turns to “1”. This signal is disabled when the spindle stop signal is “1”.

When the spindle speed for orientation is set by parameter GST No. 3705#1 and the SOR signal is input, the CNC outputs the spindle speed command corresponding to the speed set to parameter 3732 with an output polarity set by parameter ORM (No. 3706#5), but the gear select signal does not change. For example, if the SOR signal is turned to “1” with high gear selected, and the speed set to parameter No. 3732 is in the low gear range, the gear select signal does not change and the command output is calculated and output to obtain the set speed at high gear.

When the spindle motor speed is set by parameter GST (No. 3705#1)=1, the command output is output regardless of gear select signal. When the spindle motor speed is set, it is used for gear shift.

Example of usage is shown below:



Spindle speed override signal SOV0 to SOV7 <G030>

[Classification] Input signal

[Function] The spindle speed override signal specifies an override from 0% to 254% in 1% units for the S command sent to the CNC.

[Operation] An override value in binary must be set in 8 bits from SOV7 to SOV0.

The spindle speed override function is disabled (an override of 100% is applied) under the following conditions:

· Tapping cycle (M series : G84, G74 T series : G84, G88)
· Thread cutting (M series : G33 T series : G32, G92, G76)

→ When this function is not in use, specify an override of 100%; otherwise, an override of 0% becomes effective, thus disabling the spindle from rotating.

Spindle speed arrival signal SAR <G029#4>

[Classification] Input signal

[Function] The SAR signal initiates cutting feed. In other words, if the signal is logical 0, cutting feed will not start.

[Operation] Generally, this signal is used to inform the CNC that the spindle has reached the specified speed.

For this purpose, the signal must be set to 1 only after the actual speed of the spindle has reached the specified speed.

Setting parameter No. 3740 with a wait time before the start of checking the SAR signal inhibits cutting feed from starting under a condition of SAR = 1 specified before the change of the spindle command.

To use the SAR signal, it is necessary to set bit 0 (SAR) of parameter No. 3708 to 1.

The CNC checks the SAR signal under the following conditions:

a. Bit 0 (SAR) of parameter No. 3708 is set to 1.

b. Before starting distribution of the first feed (move command) block after shifting from the rapid traverse mode to the cutting feed mode. This checking is performed after the time set by parameter No. 3740 has elapsed after the feed block is read.

c. Before starting distribution of the first feed command block after an S code is commanded. The wait time for checking is the same as in item b.

d. When an S code and feed are programmed in the same block, the S code (or command output to the spindle) is output, and the SAR signal is checked after a fixed time elapses. If the SAR signal is set to “1”, feed begins.

CAUTION

According to the conditions of item d above, note that if the circuit is so designed that SAR is turned to “0” simultaneously with the output of an S code and the change of spindle speed is 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 <F001#4>

[Classification] Output signal

[Function] Informs absence or presence of spindle output command.

[Output condition] The ENB signal becomes logical 0 when the command output to the spindle becomes logical 0. Otherwise, the signal is logical 1.

During analog spindle control, S0 may not be able to stop the spindle from rotating at low speed because of an offset voltage in the spindle motor speed control amplifier. In such a case, the ENB signal can be used to provide a condition to determine whether to stop the motor.

The analog spindle interface (JA40) has electric signals (ENB1 and ENB2) similar to the ENB. These signals work under the same conditions as with the ENB signal.

The ENB signal can be used also for serial spindle control.

Gear selection signal GR1O, GR2O, GR3O <F034#0 to #2>

[Classification] Output signal

[Function] The gear select signal specifies a gear stage to the PMC.

[Output condition] For details of this signal, see descriptions on the M type gear selection method in General.

Gear selection signal GR1, GR2 <G028#1, #2>

[Classification] Input signal

[Function] This signal informs the CNC of the gear stage currently selected.

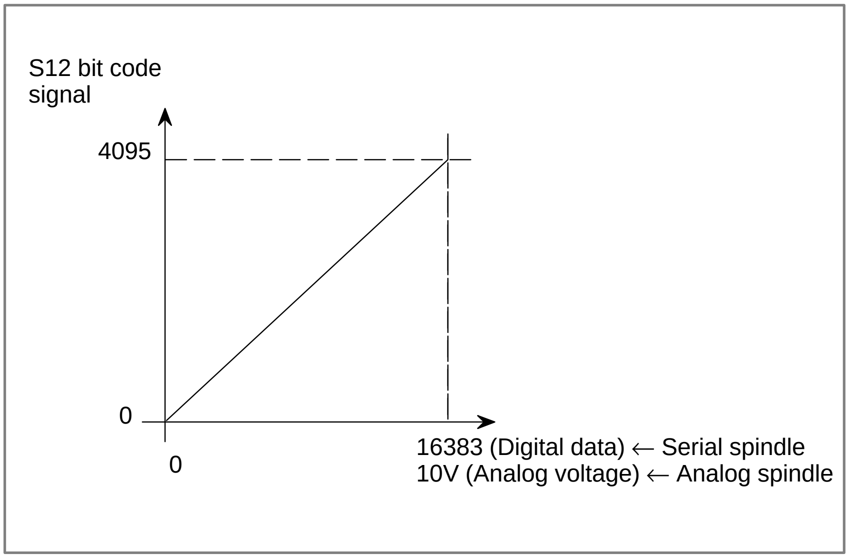
[Output condition] For details of this signal, see descriptions on the T type gear selection method in General.

S12–bit code signal R01O to R12O <F036#0 to F037#3>

[Classification] Output signal

[Function] This signal converts the spindle speed command value calculated by the CNC to code signals 0 to 4095.

[Output condition] The relationship between the spindle speed command value (calculated by the CNC) and the value output by this signal is as shown below.



This signal converts the spindle speed command value calculated by the spindle control function of the CNC to data from 0 to 4095 (for both serial and analog spindle control) and outputs the result. Note that the conversion result is not the actual output value. (See Section 15.4.)

Other signals

Spindle speed function code signal S00 to S31 <F025 to F022> (Output) Spindle speed function strobe signal SF<F007#2> (Output)	See Sections 9.1 and 15.4 for these signals.
Spindle speed output control signal by PMC SIND<G033#7> (Input) R01I to R12I <G032#0 to G033#3> (Input) SSIN <G033#6> (Input) SGN <G033#5> (Input)	See Section 15.4 for these signals.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028						GR2	GR1	
G029		*SSTP	SOR	SAR				
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I

	#7	#6	#5	#4	#3	#2	#1	#0
F001				ENB				
F007						SF		
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16
F025	S31	S30	S29	S28	S27	S26	S25	S24
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT	SGB	GST	ESF

[Data type] Bit

ESF When the spindle control function (S analog outpu or S serial output) is used, and the constant surface speed control function is used or bit 4 (GTT) of parameter No. 3706 is set to 1:

0 : S codes and SF are output for all S commands.

1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S––;).

NOTE

For the T series, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.

For the M series, SF is not output:

(1) For an S command used to specify maximum spindle speed clamping (G92S––;) in constant surface speed control mode

(2) When bit 5 (NSF) of parameter No. 3705 is set to 1

GST: The SOR signal is used for:

0 : Spindle orientation
1 : Gear shift

SGB: Gear switching method

0 : Method A (Parameters No. 3741 to 3743 for the maximum spindle speed at each gear are used for gear selection.)
1 : Method B (Parameters No. 3751 and 3752 for the spindle speed at the gear switching point are used for gear selection.)

SGT: Gear switching method during tapping cycle (G84 and G74)

0 : Method A (Same as the normal gear switching method)
1 : Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters No. 3761 and 3762).

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.
1 : Output for an S command.

> **NOTE**
> The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S——;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface speed control,

0 : SF is output.
1 : SF is not output.

SFA: The SF signal is output:

0 : When gears are switched
1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3706	TCW	CWM	ORM				PG2	PG1
	TCW	CWM	ORM	GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Spindle speed}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method

0 : Type M
1 : Type T

> **NOTE**
> 1 Type M:
> The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in parameters. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
> Type T:
> The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.
> 2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.
> 3 When type T spindle gear switching is selected, the following parameters have no effect:
> No. 3705#2 SGB, No. 3751, No. 3752, No. 3705#3 SGT, No. 3761, No. 3762, No. 3705#6 SFA, No. 3735, No. 3736
> However, parameter No. 3744 is valid.

ORM Voltage polarity during spindle orientation

0 : Positive
1 : Negative

TCW, CWM Voltage polarity when the spindle speed voltage is output

TCW	CWM	Voltage polarity
0	0	Both M03 and M04 positive
0	1	Both M03 and M04 negative
1	0	M03 positive, M04 negative
1	1	M03 negative, M04 positive

	#7	#6	#5	#4	#3	#2	#1	#0
3709								SAM

[Data type] Bit

SAM The sampling frequency to obtain the average spindle speed

0 : 4 (Normally, set to 0.)
1 : 1

3730	Data used for adjusting the gain of the analog output of spindle speed

[Data type] Word

[Unit of data] 0.1 %

[Valid data range] 700 to 1250

Set data used for adjusting the gain of the analog output of spindle speed.

[Adjustment method] (1) Assign standard value 1000 to the parameter.

(2) 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. 3730.

$$\text{Set value} = \frac{10\ (\text{V})}{\text{Measured data (V)}} \times 1000$$

(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.

3731	Compensation value for the offset voltage of the analog output of the spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to +1024

Set compesation value for the offset voltage of the analog output of the spindle speed.
Set value =–8191 × Offset voltage (V)/12.5

[Adjustment method] (1) Assign standard value 0 to the parameter.

(2) 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. 3731.

$$\text{Set value} = \frac{-8191 \times \text{Offset voltage (V)}}{12.5}$$

(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.

3732	The spindle speed during spindle orientation or the spindle motor speed during spindle gear shift

[Data type] Two–word

[Valid data range] 0 to 20000

Set the spindle speed during spindle orientation or the spindle motor speed during gear shift.

When GST, #1 of parameter 3705, is set to 0, set the spindle speed during spindle orientation in rpm.

When GST, #1 of parameter 3705, is set to 1, set the spindle motor speed during spindle gear shift calculated from the following formula.

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 16383 \text{ (For a serial spindle)}$$

$$\text{Set value} = \frac{\text{Spindle motor speed during spindle gear shift}}{\text{Maximum spindle motor speed}} \times 4095 \text{ (For an analog spindle)}$$

3735	Minimum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the minimum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Minimum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$

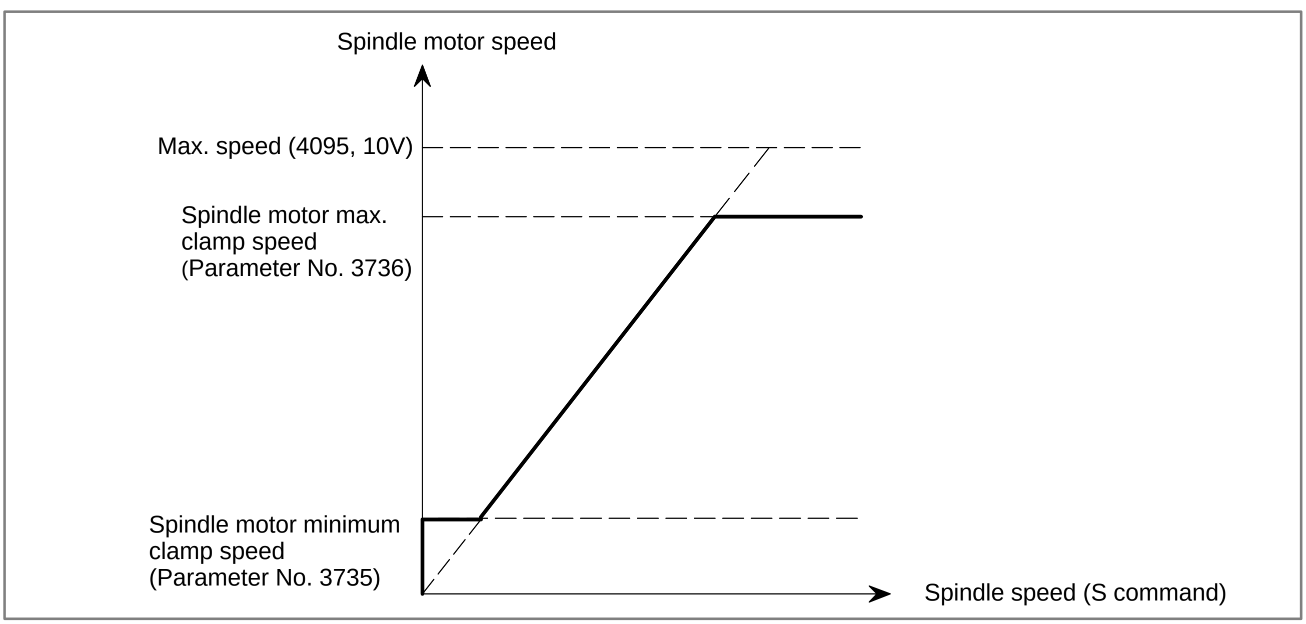
3736	Maximum clamp speed of the spindle motor

[Data type] Word

[Valid data range] 0 to 4095

Set the maximum clamp speed of the spindle motor.

$$\text{Set value} = \frac{\text{Maximum clamp speed of the spindle motor}}{\text{Maximum spindle motor speed}} \times 4095$$



3740	Time elapsed prior to checking the spindle speed arrival signal

[Data type] Byte

[Unit of data] msec

[Valid data range] 0 to 225

Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

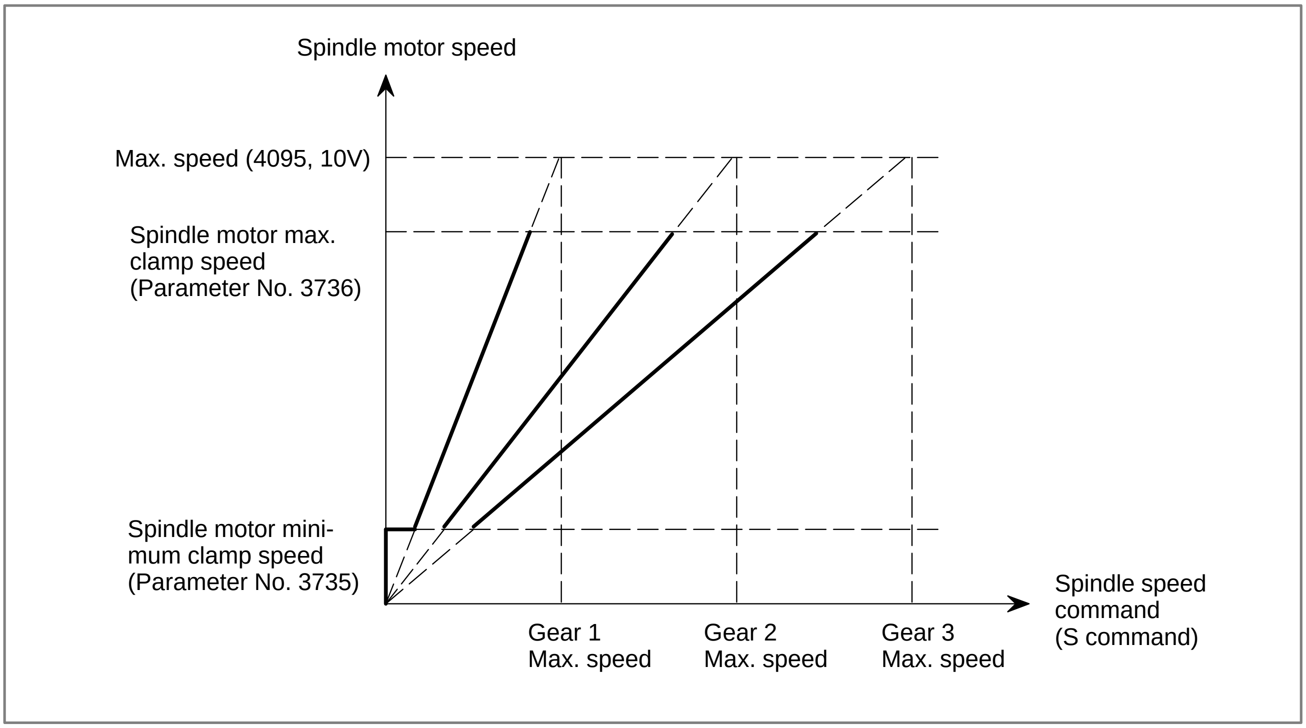
3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the maximum spindle speed corresponding to each gear.



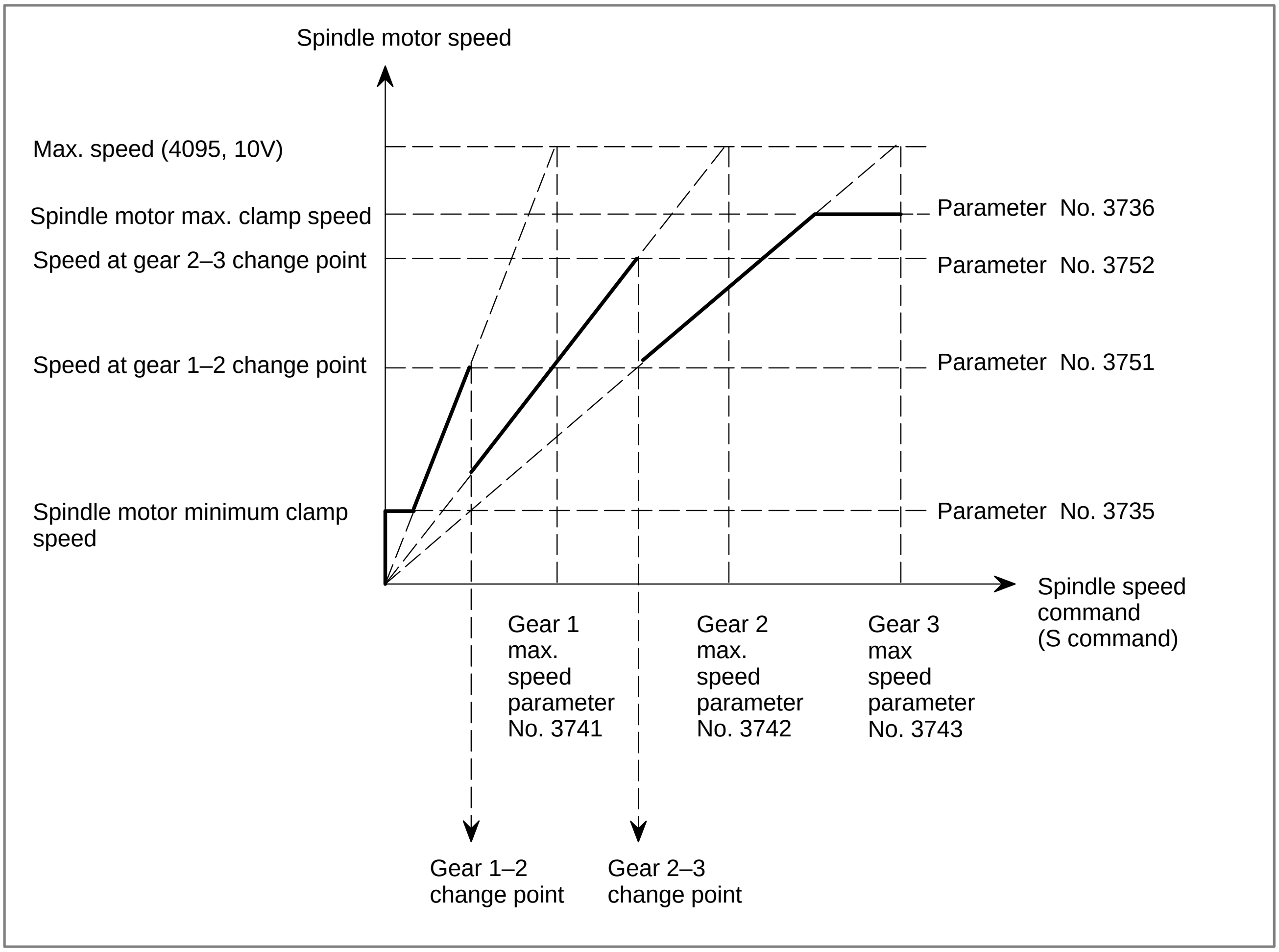
3751	Spindle motor speed when switching from gear 1 to gear 2
3752	Spindle motor speed when switching from gear 1 to gear 3

[Data type] Word

[Valid data range] 0 to 4095

For gear switching method B, set the spindle motor speed when the gears are switched.

$$\text{Set value} = \frac{\text{Spindle motor speed when the gears are switched}}{\text{Maximum spindle motor speed}} \times 4095$$



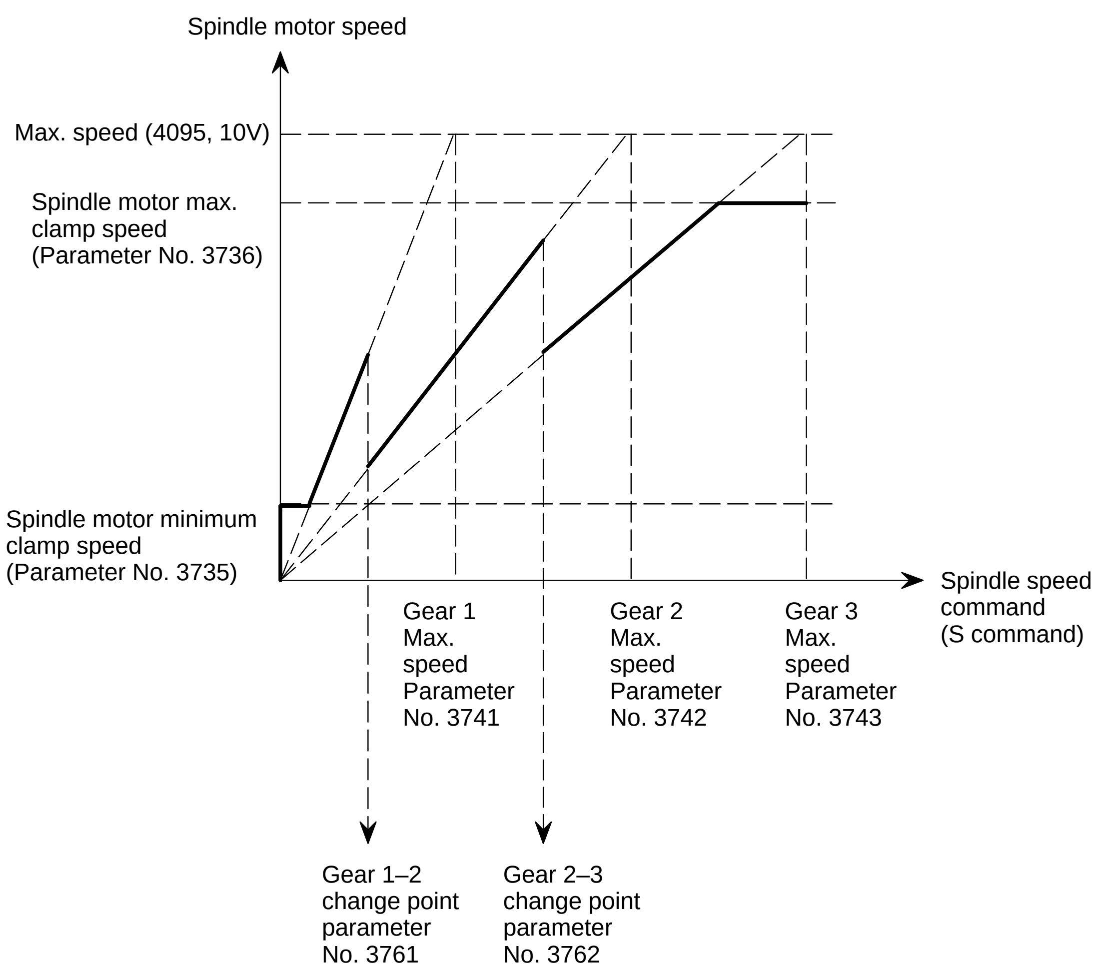
3761	Spindle speed when switching from gear 1 to gear 2 during tapping
3762	Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



3772	Maximum spindle speed

[Data type] Word

[Unit of data] 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.

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.
3 When the multi–spindle control option is selected, set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

3820	Data used 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 data used for adjusting the gain of the analog output of the third–spindle speed.

3821	Offset–voltage compensation value of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset–voltage compensation value of the analog output of the third–spindle speed.

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 TWO–PATH LATHE

General

In a two–path lathe application, the additional path section (path No. 2) can have the same spindle interface as a one–path 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 a two–path lathe application and how it is controlled.

In the following description, the term tool post 1 refers to path No. 1, and the term tool post 2 refers to path No. 2.

One–spindle control and two–spindle control

In a two–path lathe application, 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. 3703#0 (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<G063#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, the position coder feedback signal is supplied to both paths in the NC. Either tool post can be used for thread cutting and feed per rotation.

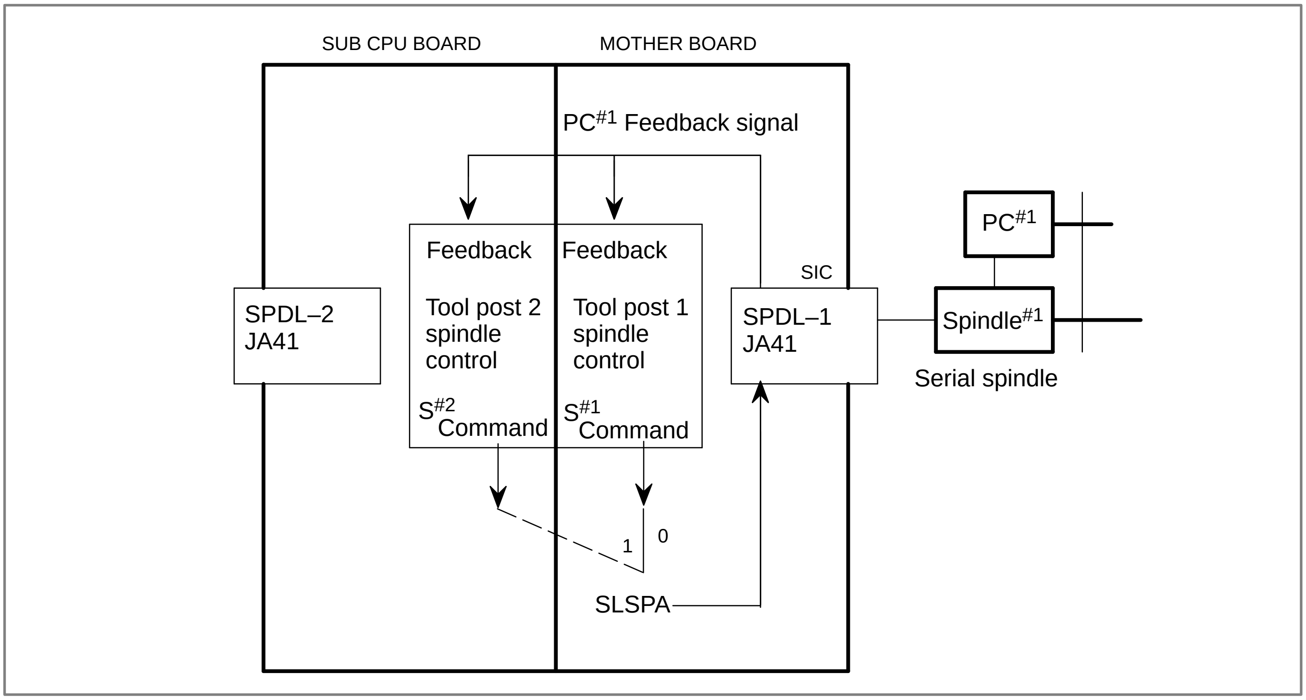


Fig. 9.4 (a) One spindle control (serial spindle)

- **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.

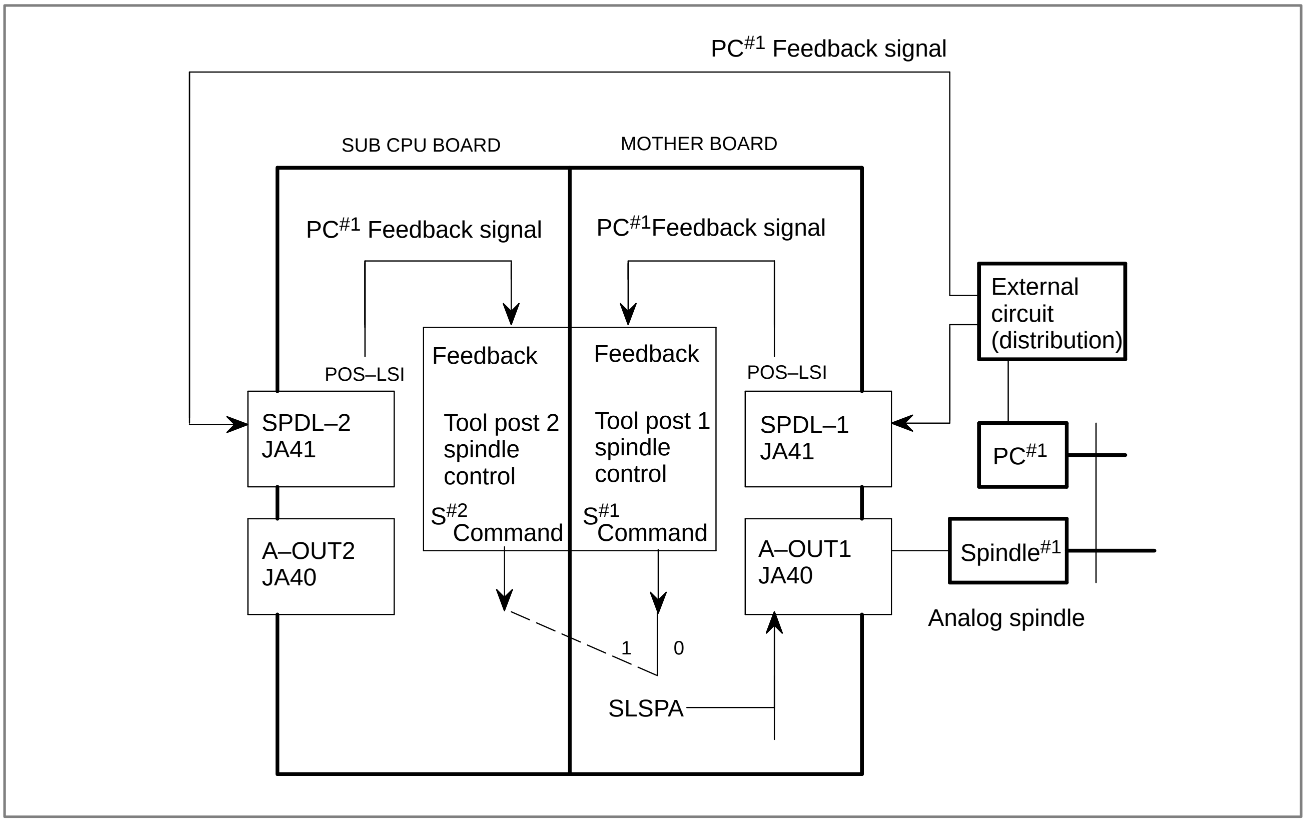


Fig. 9.4 (b) One spindle control (analog spindle)

Two–spindle control

The spindle interface for either tool post is used.

- **Selection of spindle command**

The spindle command select signals SLSPA <G063#2> and SLSPB <G063#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<G064#2> and SLPCB<G064#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.

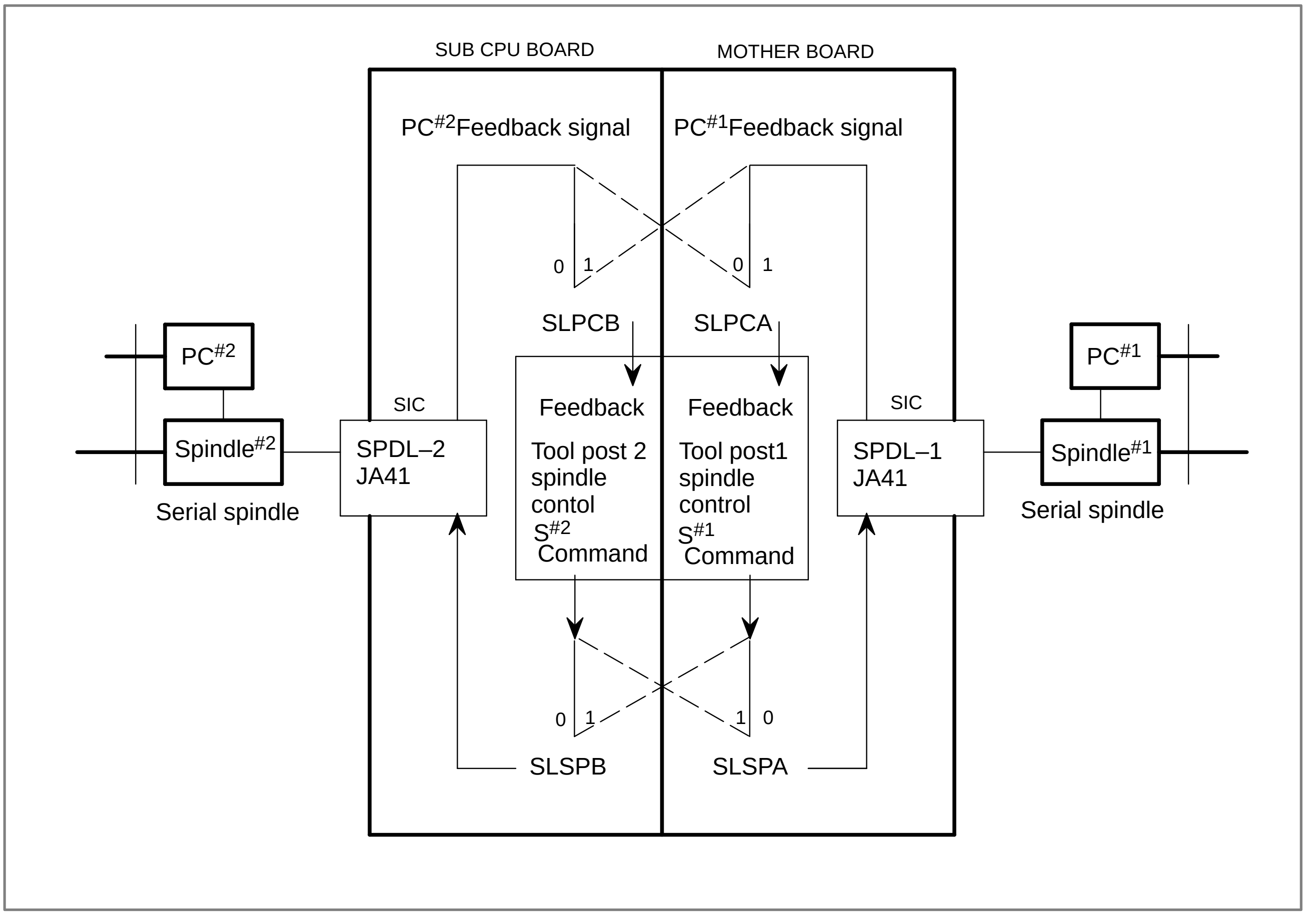


Fig. 9.4 (c) Two–spindle control (serial spindle)

- **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<G064#2> and SLPCB<G064#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.

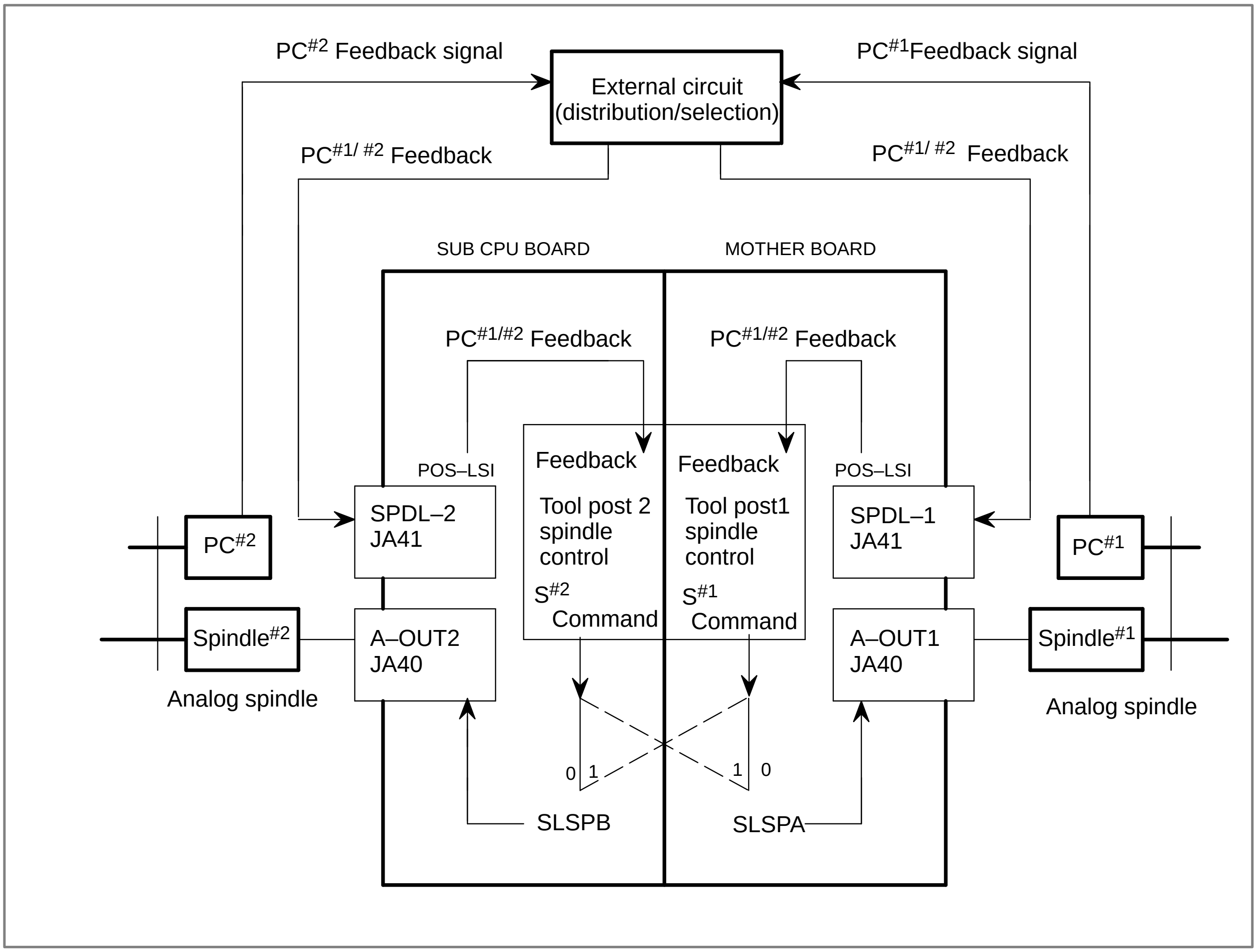


Fig. 9.4 (d) Two–spindle control (analog spindle)

2nd and 3rd spindles

If the first spindle is a serial spindle, the second and third spindles can also be used in a two–path lathe application. (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.

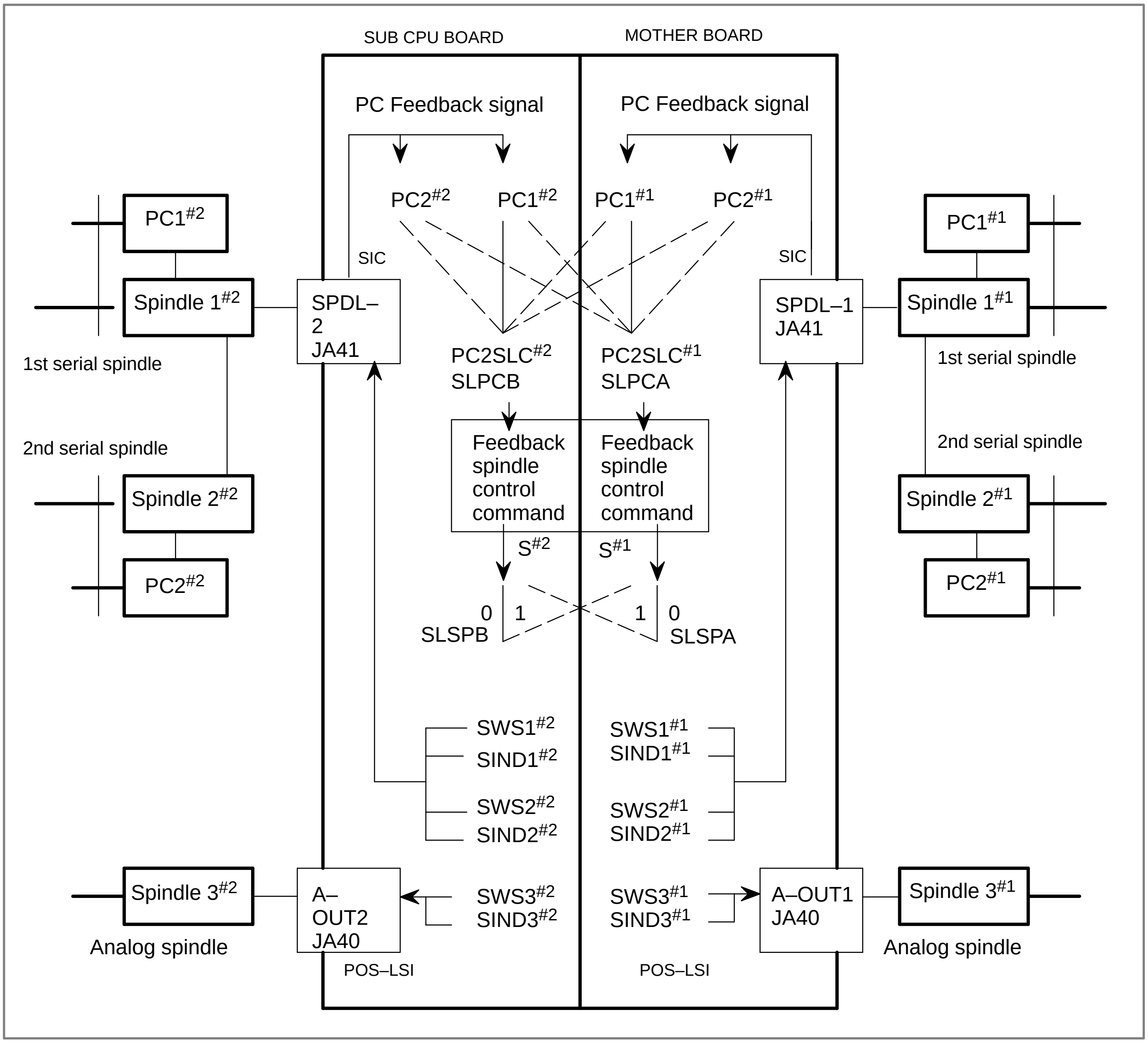


Fig. 9.4 (e) Spindle configuration and flow of commands and feedback signals in an two–path lathe application (with all spindles under two–spindle control)

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 multi–spindle control is applied to both tool posts, the position coder feedback signal for the second spindle of each tool post also becomes usable.

See descriptions of bit 3 (PCS) of parameter No. 3706.

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

· Cs contour control

· Spindle positioning

· Multi–spindle control

Refer to parameter No. 3702.

Signal

Spindle command select signals SLSPA <G063#2>, SLSPB <G063#3>

[Classification] Input signal

[Function] Selects each tool post's spindle receives spindle command of which tool posts.

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 <G064#2> SLPCB <G064#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 with the spindle connected to tool post 1.

PC#2=Position coder feedback signal with 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 <F064#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
G063					SLSPB	SLSPA		
G064					SLPCB	SLPCA		

	#7	#6	#5	#4	#3	#2	#1	#0
F064			COSP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702	ECS	ESS	EAS	ESI			EMS	

[Data type] Bit

EMS Multi–spindle control function

0 : Used
1 : Not used

> **NOTE**
> If the multi–spindle control function is not required for one tool post in two–path control, specify this parameter for the tool post to which the multi–spindle control function need not be applied.

ESI The spindle positioning function is

0 : Used
1 : Not used

> **NOTE**
> This parameter is used when the spindle positioning option specified with two–path control, and the spindle positioning function is not required for either path. Set ESI to 1 for a system that does not require the spindle positioning function.

EAS For tool post 1 (or tool post 2), the S analog output function is:

0 : Used.
1 : Not used.

ESS For tool post 1 (or tool post 2), the S serial output function is:

0 : Used.
1 : Not used.

ECS For tool post 1 (or tool post 2), the Cs contour control function is:

0 : Used.
1 : Not used.

> **NOTE**
> Parameter EAS, ESS, and ECS are used for T series 2–path control.
> These parameters are used to determine whether the optional function, S analog output function, S serial output function, and Cs contour control function, are used for each tool post.

	#7	#6	#5	#4	#3	#2	#1	#0
3703								2SP

[Data type] Bit

2SP Specifies whether one or two spindles are controlled (T series 2–path control).

0 : One spindle (two tool posts)
1 : Two spindle (two tool posts)

	#7	#6	#5	#4	#3	#2	#1	#0
3706					PCS			

[Data type] Bit

PCS When multi–spindle control is applied to two tool posts in two–path control, this parameter specifies whether a position coder feedback signal from the other tool post is selectable, regardless of the state of the PC2SLC signal (bit 7 of G028/bit 7 of G1028) of the other tool post:

0 : Not selectable.
1 : Selectable. (To select a position coder for the other tool post, the SLPCA signal (bit 2 of G064) and the SLPCB signal (bit 3 of G064) are used.)

NOTE
Multi–spindle control based on the same serial spindle must be applied to both tool posts.

- Selecting position coder feedback signals for both tool posts in a two–path lathe application under multi–spindle control.

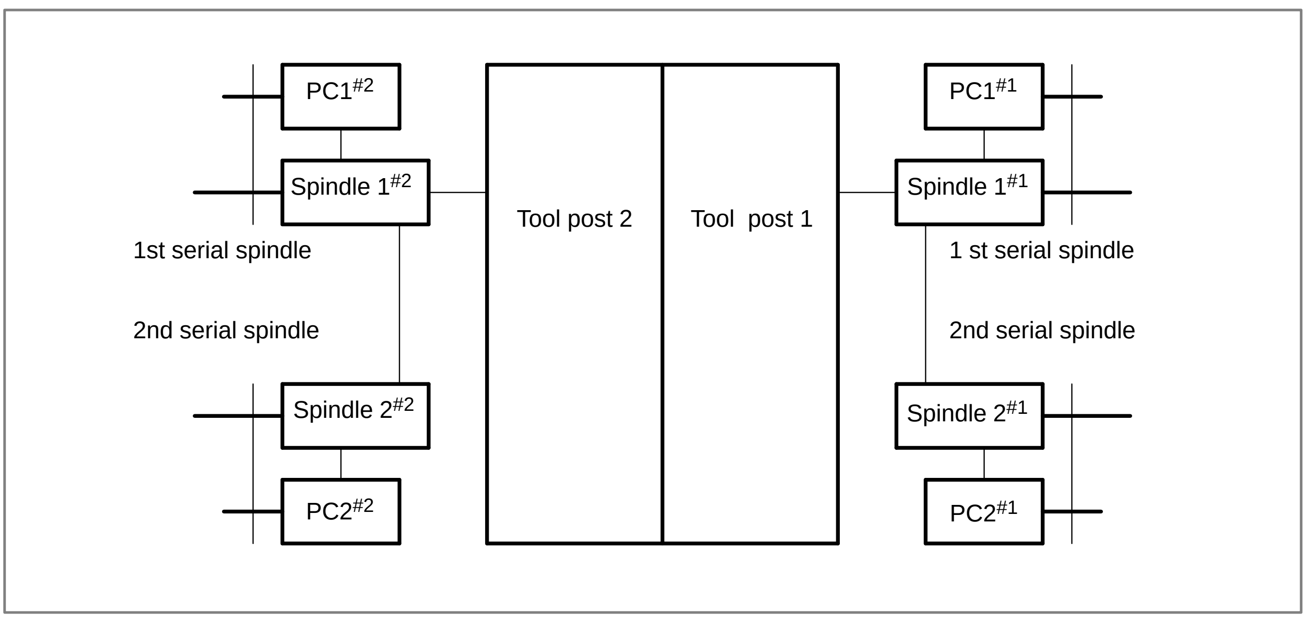


Table 9.4 lists the position coder feedback signals used for each tool post in the above configuration. These position coder feedback signals are selected according to the following:

· Bit 3 (PCS) of parameter No. 3706

· Spindle feedback select signals SLPCA <G064#2> and SLPCB <G064#3>

· Multi–spindle control
Second position coder select signals PC2SLC#1 <G028#7> and PC2SLC#2 <G1028#7>

Table 9.4 Selection of position coder feedback signal in two–path lathe
(— means position coder selection is indifferent on the tool post side)

When parameter No. 3706#3=1

	Position coder select	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”

When parameter No. 3706#3=0

	Position coder select	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)

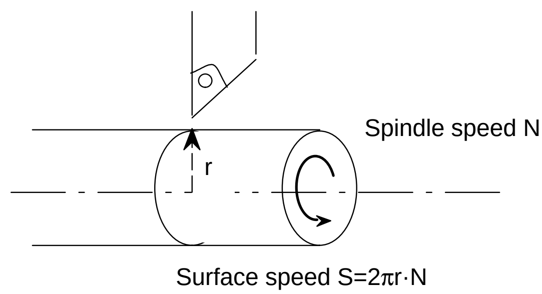
2 Signals to operate the spindle control unit are not affected by the spindle command select signals SLSPA<G063#2> or SLSPB<G063#3>. They may be processed in the PMC ladder, as required.
(Example: SFRA <G070#5>is always a forward rotation command for the first spindle control amplifier of tool post 1.)

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.

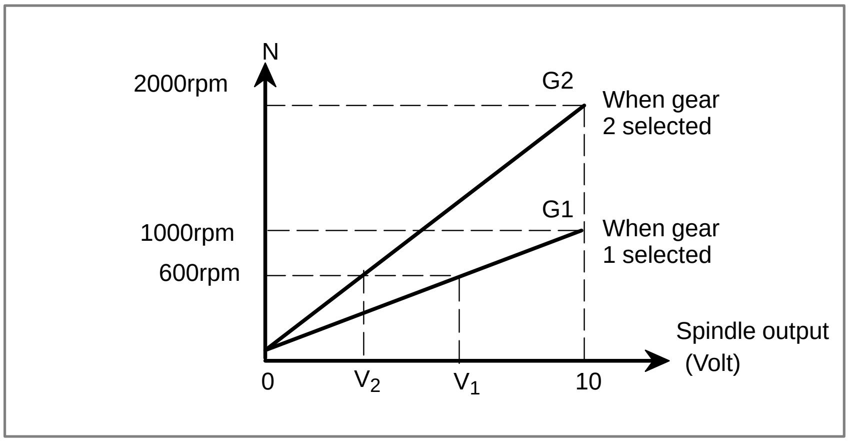
G97: Constant surface speed control not performed. S in the G97 mode is rev/min.

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 No. 3741-3744.
The gear select signal is a 2 bit code signal (GR1, GR2). The relationship between the signal and gear number is :

GR1	GR2	Gear number
0	0	1
1	0	2
0	1	3
1	1	4

Example of Spindle Analog Output

Assume that gear switching is two stage switching. If the spindle speed with the output 10 V is 1000 rpm for the low speed gear (G1) and 2000 rpm for the high speed gear (G2), set these speeds to the parameter No. 3741, 3742, respectively. In this case, the spindle output has the linear relationship shown below:



Here, S = 60 m/min is given as the surface speed; if the position of the present X-axis cutter is 16 mm from the center, the spindle speed N becomes 600 rpm (S = 2πr 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 No. 3741 to No. 3744)

S: Surface speed (m/min) specified by S command

r: Radius value in the X-axis direction (m)

(ii) G97

$$V= \frac{10N}{R}$$

R: Spindle speed at 10V output voltage (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 No. 3741 to No. 3744)

S: Surface speed (m/min) specified by S

r: Radius value in the X-axis direction (m)

(ii) G97

$$D=\frac{4095N}{R}$$

R: Spindle speed at maximum spindle motor speed (rpm)

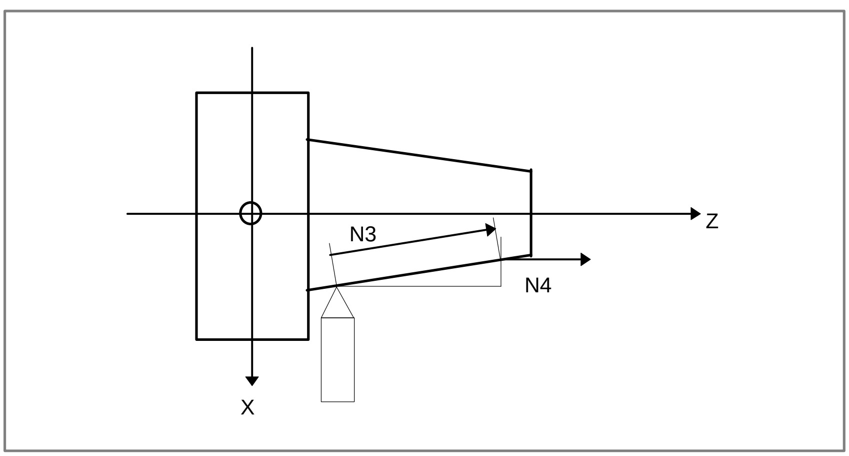
N: Spindle speed given by S command (rpm)

Constant surface speed control with no position coder (T series)

In a machine with no position coder (or without using a position coder), it is usually impossible to use feed per rotation during constant surface speed control. Setting parameter FPR (bit 2 of parameter No. 1405) to 1, however, makes it possible to use feed per revolution in a machine with no position coder (or without using a position coder) even when it is under constant surface speed control. In this type of feed per revolution, it is assumed that the spindle speed is specified by the 12–bit S code signal R01O to R12O <F036#0 to F037#3>.

For example, the following program is executed with G code system B and diameters specified.

```
N1 G00 X50.  Z10.  ;
N2 G96 G95 S12  ;           ← Constant surface speed control and
                              feed per revolution begin.
N3 G01 X20.  Z30.  F10.  ;
N4 Z50.  ;
N5 G97 S200  ;              ← Constant surface speed control ends.
N6 G00 Z200.  ;
N7 M30  ;
```



In this program, block N2 issues a constant surface speed control command (G96), a surface speed command (S12 m/min), and a feed–per–revolution command (G95). Block N3 causes the CNC to change the spindle speed specification from 76.4 rpm to 191 rpm so as to maintain a constant surface speed during movement to X = 20.

Meanwhile, the feed–per–revolution speed is changed according to the changed spindle speed specification, and used for movement along the feed axis. However, the specified spindle speed is clamped to the upper limit to the spindle speed specification (for the first spindle, using parameter No. 3772). In the above program, the feed–per–revolution speed command F10 in block N3 corresponds to an actual speed of 764 (mm/min) to 1910 (mm/min).

Signal

Gear selection signal (Input) GR2, GR1 <G028#2, #1>

Refer to section 9.3 “Spindle Speed Control”.

Constant surface speed signal CSS <F002#2>

[Classification] Output signal

[Function] This signal indicates that constant surface speed control is in progress.

[Output condition] “1” indicates that constant surface speed control mode (G96) is in progress, while “0” indicates it is not.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002						CSS		

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
1405						FPR		

[Data type] Bit

FPR Specifies whether to use a function that converts the specified spindle rotation speed to the actual spindle rotation speed in feed–per–revolution mode, that is, the feed–per–revolution function with no position coder.

0 : Not used.
1 : Used.

3741	Maximum spindle speed for gear 1
3742	Maximum spindle speed for gear 2
3743	Maximum spindle speed for gear 3
3744	Maximum spindle speed for gear 4

[Data type] Word

[Unit of data] rpm

3770	Axis as the calculation reference in constant surface speed contorol

[Data type] Byte

[Valid data range] 0, 1, 2, 3, ..., number of control axes
set the axis as the calculation reference in constant surface speed contorol.

> **NOTE**
> When 0 is set, constant surface speed control is always applied to the X–axis. In this case, specifying P in a G96 block has no effect on the constant surface speed control.

3771	Minimum spindle speed in constant surface speed control mode (G96)

[Data type] Word

[Unit of data] 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 3771.

3772	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	In the constant surface speed control, the axis specification is wrong. (See parameter No. 3770.) The specified axis command (P) contains an illegal value. Correct the program.

Caution

CAUTION

1 If the spindle speed corresponding to the calculated surface speed exceeds the speed specified in the spindle speed clamp command (G50S_ for T series and G92S_ for M series) during the G96 mode, the actual spindle speed is clamped at the value specified in the spindle speed clamp command.
If the specified spindle speed is lower than the value specified in parameter No. 3771, the actual spindle speed is clamped at the 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 enables constant surface speed control for spindles other than the first spindle. (See Section 9.10.)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.9.3	CONSTANT SURFACE SPEED CONTROL (G96, G97)

9.6 SPINDLE SPEED FLUCTUATION DETECTION

General

With this function, an overheat alarm (No. 704) is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.
G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

Detection of Spindle Speed Fluctuation

The function for detecting spindle speed fluctuation checks whether the actual speed varies for the specified speed or not. Sd or Sr, whichever is greater, is taken as the allowable fluctuation speed (Sm). An alarm is activated when the actual spindle speed varies for the commanded speed (Sc) under the condition that the variation width exceeds the allowable variation width (Sm).

Sd: The allowable constant variation width which is independent of the specified spindle speed (Sd is set with parameter No. 4913.)

Sr: The allowable variation width which is obtained by multiplying Sc (commanded spindle speed) by r (constant ratio). (r is set with parameter No. 4912.)

Sm: Sd or Sr, whichever is greater

Conditions to start spindle speed fluctuation detection

If the specified spindle speed Sc changes, spindle speed fluctuation detection starts when one of the conditions below is met:

· The actual spindle speed falls in a range of (Sc – Sq) to (Sc + Sq)

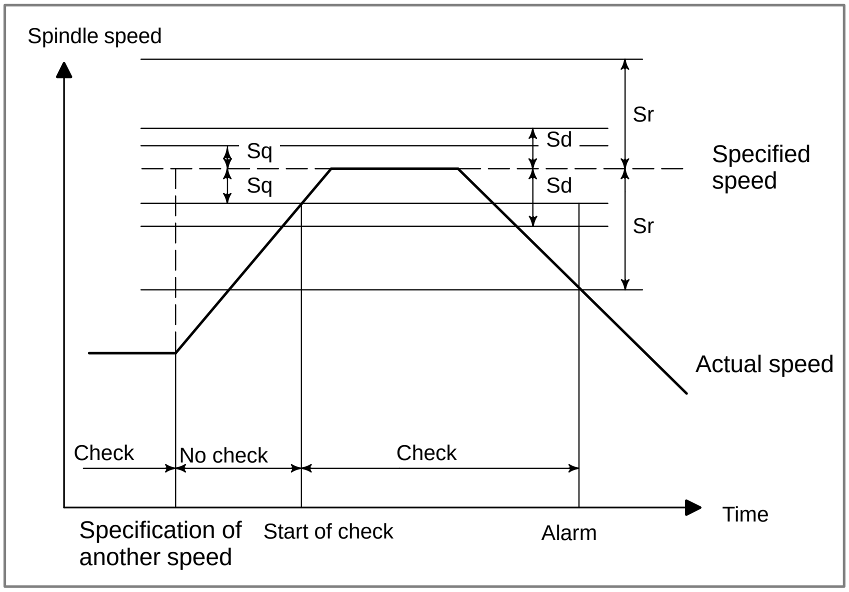
where Sq = Sc × q/100

q: Percent tolerance of the target spindle speed, specified in parameter No. 4911. If the actual spindle speed is in a range of the specified speed ± q, it is assumed that the actual speed has reached the specified speed.

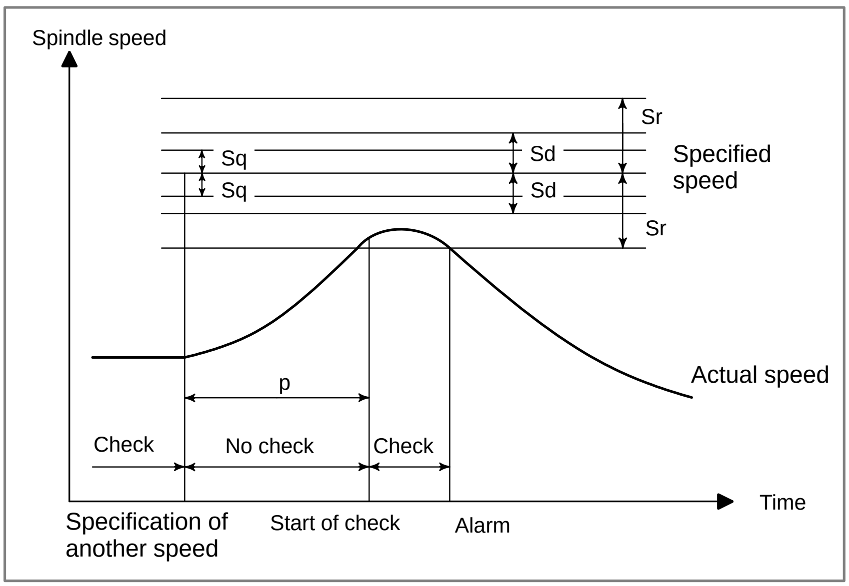
· When time p specified in parameter No. 4914 elapses after the specified speed Sc changes.

Parameter No. 4914, 4911, and 4912 can be rewritten also by program (G26 PpQqRr).

1. When an alarm is issued after a specified spindle speed is reached



2. When an alarm is issued before a specified spindle speed is reached



Specified speed :
(Speed specified by address S and five–digit value)×(spindle override)
Actual speed : Speed detected with a position coder

Signal

Spindle fluctuation detection alarm signal SPAL <F035#0>

[Classification] Output signal

[Function] This signal indicates that the actual spindle speed is not within a tolerance to the specified speed.

[Output condition] The signal becomes logical “1” when:

· The actual spindle speed goes out of tolerance to the specified speed.

The signal becomes logical “0” when:

· No alarm condition has been detected for spindle speed fluctuation.

· An alarm condition is cleared by resetting the NC when the signal is logical “1”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F035								SPAL

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3708				SVD				

[Data type] Bit

SVD When the SIND signal is on, the detection of spindle speed fluctuation is:

0 : Disabled
1 : Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
4900								FLR

[Data type] Bit

FLR When the spindle speed fluctuation detection function is used, the rates of allowance (q) and fluctuation (r) those are set in parameter No. 4911 and No. 4912, respectively are set in steps of:

0 : 1%
1 : 0.1%

4911	Ratio (q) of the fluctuation of spindle speed which is assumed to be the specified spindle speed

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

NOTE
Unit of data depends on parameter No. 4900#0 FLR (T series only)

Set the ratio (q) of the spindle speed which is assumed to be the specified spindle speed in the spindle speed fluctuation detection function.

4912	Spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data]

[Valid data range]

Unit of data	1%	0. 1% (T series)
Data range	1 – 100	1 – 1000

> **NOTE**
> Unit of data depends on parameter No. 4900#0 FLR (T series only).

Set the spindle speed fluctuation ratio (r) for which no alarm is activated in the spindle speed fluctuation detection function.

4913	Spindle speed fluctuation value (d) for which no alarm is activated in the spindle speed fluctuation detection function

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Set the allowable fluctuation speed (d) for which no alarm is activated in the spindle speed fluctuation detection function.

4914	Time (p) elapsed from when the commanded spindle speed is changed to the start of spindle speed fluctuation detection

[Data type] Two–word

[Unit of data] msec

[Valid data range] 0 to 999999

Set the time (p) elapsed from when the specified spindle speed is changed to the start of spindle speed fluctuation detection 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	OVER HEAT : SPINDLE	Spindle overheat in the spindle fluctuation detection (1) If the cutting load is heavy, relieve the cutting condition. (2) Check whether the cutting tool is share. (3) Another possible cause is a faulty spindle amp.

Note

NOTE

1 When an alarm is issued in automatic operation, a single block stop occurs.
2 No check is made during spindle stop state (*SSTP = 0).
3 An alarm is issued one second later if the actual spindle speed is found to be 0 rpm.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.9.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.9.4	SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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 <F040, F041>

[Classification] Output signal

[Function] These 16-bit binary code signals output from the CNC to the PMC the actual spindle speed obtained by feedback pulses from the position coder mounted on the spindle.

[Operation] $$\text{Spindle speed} = \sum_{i=0}^{15}\{2^i \times V_i\}\text{rpm}$$

where Vi = 0 when ARi is “0” and Vi = 1 when ARi is “1”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08

Note

NOTE

1 The AR0 - AR15 signals are always output. Their values change every 64 msec.

2 An absolute error of about 0.5 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.

Also, relationship between M codes and these operations are set by parameters (refer to No. 4950#2 (ISZ), #7 (IMB)).

· Least command increment

$$\frac{360}{4096} \fallingdotseq 0.088 \text{ deg}$$

· Minimum input increment

0.001 deg

· Maximum command value

±9999.999 deg

Selecting a spindle positioning axis

Any axis in the control axis group can be used as the C axis (parameter no. 1020). Specify -1 as its servo axis number (parameter no. 1023).

Only one set of this setting can be used for each control path. The spindle subjected to spindle positioning is the first spindle.

Switching to spindle positioning mode (Spindle orientation)

Orientation is required in advance if spindle positioning is first performed after the spindle motor is used as a normal spindle, or when spindle positioning is interrupted. The orientation stops the spindle in a constant position. The orientation position can be sifted in the range of ±180deg for analog spindle and in the range from 0 to 360° for serial spindle.

To specify orientation, use the M code whose parameter No. 4960.
The orientation direction is specified by using parameters ZMIx No. 1006 #5 for analog or RETURN No. 4000#5 for serial spindles.

- **Orientation speed**

The spindle moves at rapid traverse set by parameter No. 1420 until it reaches the orientation enable speed (shown below). After the spindle crosses that speed point, it performs orientation at the speed set by parameter No. 1425. When a serial spindle is used, orientation speed depends on the spindle.

· Orientation enable speed
RPD>9 × (loop gain) KPPM
Loop gain: Parameter No. 4970 (unit: 1/sec)
Set rapid traverse speed at above value.

(Example)

When the loop gain parameter No. 4970 is set to 20 [1/sec], the orientation speed is:

RPD>9 × 20 × 1000 × (360/4096)=15820 [deg /min]

The serial spindle stops at the orientation position as soon as the command is issued. The lower limit to the rapid traverse speed value does not need to be specified for the serial spindle to reach the orientation enable speed.

The analog spindle stops after the spindle speed is changed from rapid traverse to the FL speed. The rapid traverse speed lower limit must be specified for the analog spindle, or obtaining the orientation enable speed need not be specified for the serial spindle. However, it must be specified for the analog spindle.

- **Program origin**

The orientation position is regarded as a program origin. It is possible to modify the program origin using the coordinates system or automatic coordinates system settings (parameter ZPR No. 1201#0)

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 ($M\alpha$ to $M(\alpha + 5)$), where α is set by parameter No. 4962. Indexing angle β also requires prior parameter setting data No. 4963. Rotation can be done in any direction, using parameter IDM data No. 4950#1.
Also, when extended specification is used (parameter No. 4950#6 ESI=1), max. 256 kinds of values ($M\alpha$ to $M(\alpha+255)$) can be specified.

M–code	Indexing angle	eg) when β=30°
Mα	β	30°
M (α+1)	2β	60°
M (α+2)	3β	90°
M (α+3)	4β	120°
M (α+4)	5β	150°
M (α+5)	6β	180°

- **Optional angle positioning by C or H address**

Numerical value following either the C or H address is used to command the position. C and H addresses are commanded in G00 mode.

(Example) C-1000

H4500

(i) Minimum setting unit :

0.001deg

(ii) Maximum command value:

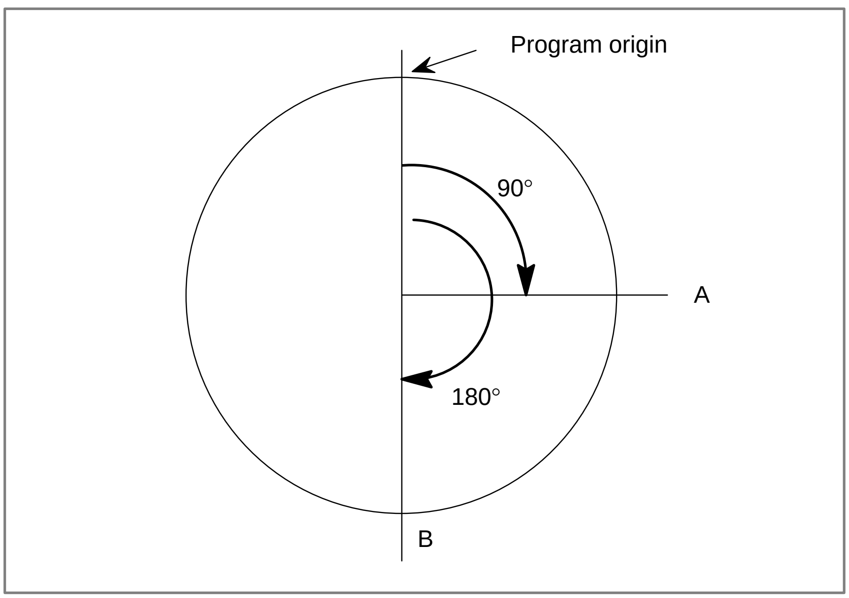
±9999.999 deg

(iii) Decimal point input: A numerical value with decimal point can be entered. The decimal point location is in "degrees", for instance:

C35.0=C35 degrees

- **Absolute and incremental commands**

The semi–fixed angle is always incremental. The optional angle positioning using address C or H differs as follows:



Command method		G code system A		G code system B, C	
		Address used	Command of A→B on the above Fig.	Address used and G–code	Command of A→B on the above Fig.
Absolute command	Direct the end point position by the distance from the program origin.	C	C180.0 ;	G90,C	G90C180.0 ;
Incremental command	Command by the distance between the start and end points.	H	H90.0 ;	G91,C	G91C90.0 ;

- **Spindle positioning feedrate**

Spindle positioning is done at the rapid traverse rate specified by parameter No. 1420, to which linear acceleration deceleration are applied. Overrides of 100%, 50%, 25% and F0 (parameter No. 1421) are also applied.

- **Spindle positioning reset**

A specific M code (parameter no. 4961) must be set when the mode is changed from spindle positioning to normal spindle rotation.

Signal

Spindle stop complete signal SPSTP <G028#6>

[Classification] Input signal

[Function] When this signal is 1, the CNC orients and positions the spindle.

Spindle unclamp signal SUCLP <F038#1>

[Classification] Output signal

[Function] This signal specifies that spindle mechanical clamping be released in a spindle positioning sequence.

When this signal is output, unclamp the spindle on the machine (release the brakes or extract the pin).

[Output condition] Refer to the sequence (time chart) in this section.

Spindle unclamp completion signal *SUCPF <G028#4>

[Classification] Input signal

[Function] This signal indicates that unclamping the spindle is complete in response to the spindle unclamp signal SUCLP.

Spindle clamp signal SCLP <F038#0>

[Classification] Output signal

[Function] This signal specifies that the spindle be clamped mechanically in a spindle positioning sequence.

When this signal turns to 1, clamp the spindle on the machine (apply the brakes or insert the pin).

[Output condition] Refer to the sequence (time chart) in this section.

Spindle clamp completion signal *SCPF <G028#5>

[Classification] Input signal

[Function] This signal indicates that clamping the spindle is complete in response to the spindle clamp signal SCLP.

Other signals

Gear selection signal GR1, GR2, <G028#1, #2>

Refer to 9.3 “Spindle Speed Control.”

CTH1A, CTH2A <G070#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 ZPx<F094>

[Classification] Output signal

[Function] This signal indicates that the spindle orientation for the spindle positioning has been completed.

[Output condition] When spindle orientation is complete, this signal turns to 1. When spindle positioning is performed or cleared, it turns to 0.

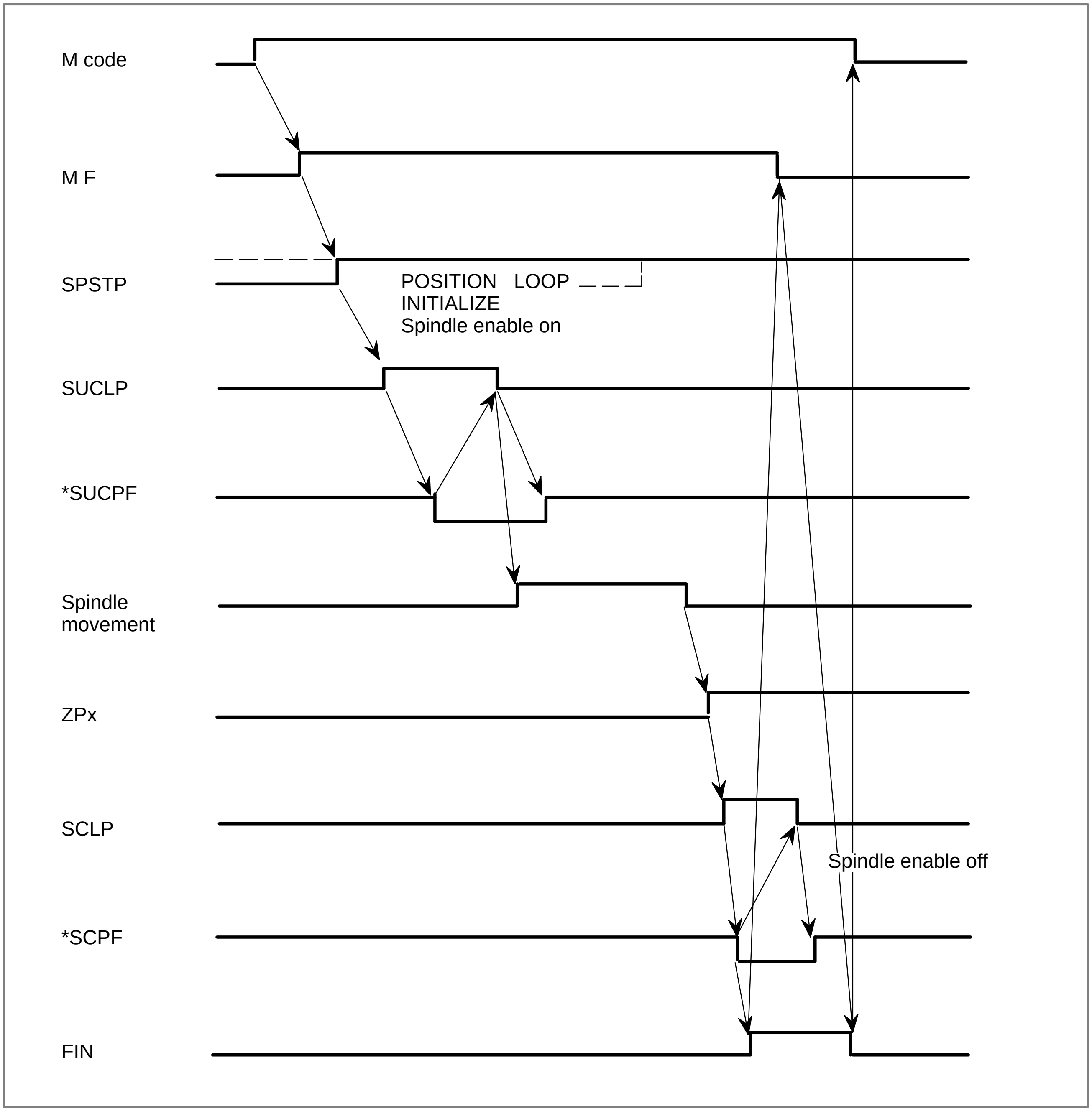
Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G028		SPSTP	*SCPF	*SUCPF		GR2	GR1	
G070					CTH1A	CTH2A		

	#7	#6	#5	#4	#3	#2	#1	#0
F038							SUCLP	SCLP
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1

Sequence (Time chart)

❑ Spindle Orientation



⇒ POSITION LOOP INITIALIZE is performed within the CNC.

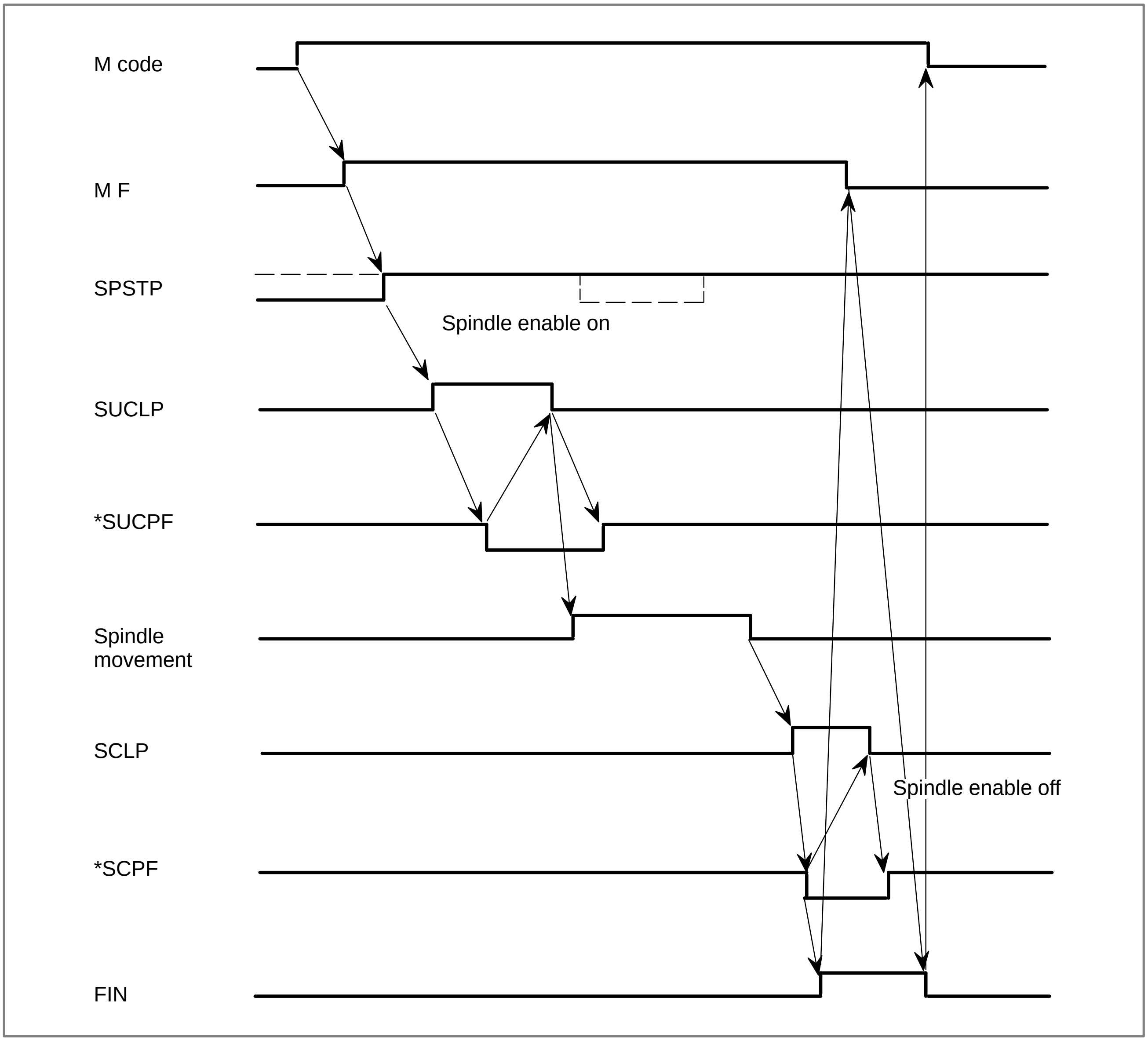
⇒ Spindle ENABLE ON/OFF specifies that the PMC ladder direct the spindle control unit to run or stop the spindle motor.

(Example) For serial spindles, the ladder should contain the following command or something like that:

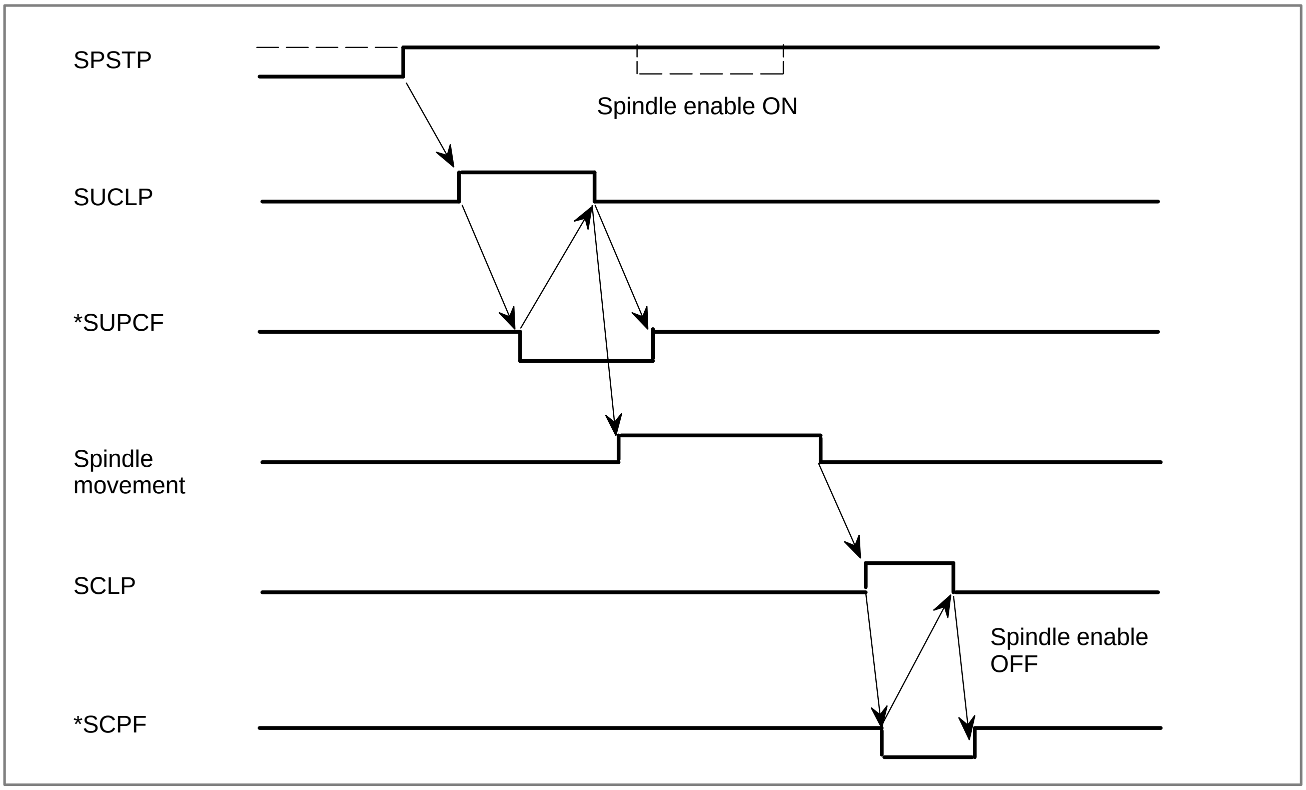
ENABLE ON, and SFRA<G070#5> ⇒ 1
ENABLE OFF, and SFRA<G070#5> ⇒ 0

For details, refer to the manual for the spindle control unit you actually use.

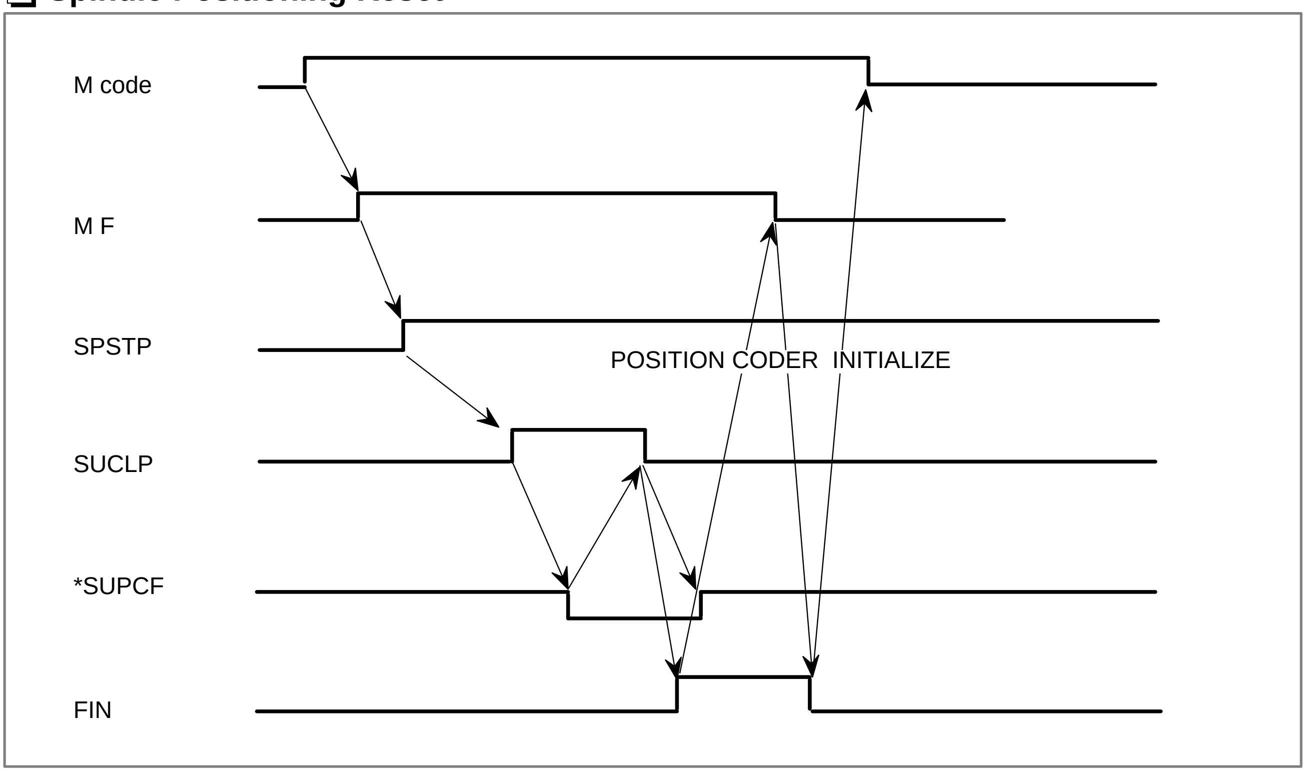
❑ Spindle Positioning by M code



❑ Spindle Positioning by Address C,H



❑ Spindle Positioning Reset



⇒ POSITION CODER INITIALIZE is performed only in the CNC.

Parameter

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

[Data type] Bit axis

ZMIx The direction of reference position return and the direction of initial backlash at power–on

0 : Positive direction

1 : Negative direction

NOTE
When the serial spindle is being used, this parameter is invalid for the spindle positioning axis.

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

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

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

NOTE
1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
2 The same axis name cannot be assigned to more than one axis.

<u>The axis name of spindle positioning is C axis.</u>

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

<u>Set –1 to the C axis when spindle positioning function is used.</u>

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed

0 : Not set automatically
1 : Set automatically

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

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

1420	Rapid traverse rate for each axis

[Data type] Word axis

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

For spindle positioning.

[Unit of data] 10 deg/min

[Valid data range] 30 to 12000

1421	F0 rate of rapid traverse override for each axis

[Data type] Word axis

Set the F0 rate of the rapid traverse override for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

1425	FL rate of the reference position return for each axis

[Data type] Word axis

Set feedrate (FL rate) after deceleration when the reference position return is performed for each axis.

For spindle positioning.

[Unit of data] deg/min

[Valid data range] 600 to 15000

NOTE
When serial spindle is used, this parameter becomes invalid.

1620	Time constant of rapid traverse linear acceleration/deceleration for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

Set time constant of rapid traverse linear acceleration/deceleration for each axis.

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

[Data type] Bit axis

DM1x to DM3x Setting of detection multiply

<u>Set this parameter to “111” (=4) for spindle positioning.</u>

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value}=\frac{1}{(\text{Command multiply})}+100 \quad [\text{Valid data range: } 102 \text{ to } 127]$$

- 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>

1821	Reference counter size for each axis

[Data type] Two–word axis

[Unit of data]

[Valid data range] 0 to 99999999

Set the size of the reference counter.

<u>Set this parameter to 10000 for spindle positioning.</u>

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

1850	Grid shift for each axis

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to ±99999999

Set a grid shift for each axis.

> **NOTE**
> Set this parameter when the analog spindle is used. When the serial spindle is used, set the value to No. 4073.

1851	Backlash compensating value for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] –9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3405				CCR				

[Data type] Bit

CCR Addresses used for chamfering and corner rounding

0 : Address used for chamfering and corner rounding is I or K, not C. In direct drawing dimension programming, addresses ’C, ’R, and ’A (with comma) are used in stead of C, R, and A.

1 : Addresses used for chamfering, corner rounding, and direct drawing dimension programming are C and R without comma. Thus, addresses C cannot be used as the names of axes.

<u>Always set this parameter to “0” for spindle positioning.</u>

	#7	#6	#5	#4	#3	#2	#1	#0
4000					RETRN			

[Data type] Bit

RETRN Reference position return direction of spindle.

0 : CCW (Counter clockwise)

1 : CW (Clockwise)

> **NOTE**
> The direction for spindle orientation (or reference position return) in spindle positioning using a serial spindle is determined by this parameter.

4044	Velocity loop proportion gain in servo mode (High gear)
4045	Velocity loop proportion gain in servo mode (Low gear)

[Data type] Word

[Unit of data]

[Valid data range] 0 to 32767

This parameter sets a velocity loop proportional gain in servo mode (spindle positioning, rigid tapping, etc.)

> **NOTE**
> Set this parameter when serial spindle is used.

4052	Velocity loop integral gain in servo mode (High gear)
4053	Velocity loop integral gain in servo mode (Low gear)

[Data type] Word

[Unit of data]

[Valid data range] 0 to 32767

This parameter sets a velocity loop integral gain in servo mode (spindle positioning, rigid tapping, etc.)

> **NOTE**
> Set this parameter when serial spindle is used.

4056	Gear ratio (HIGH)
4057	Gear ration (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word

[Unit of data] Motor speed per spindle rotation × 100

[Valid data range] 0 to 32767

These parameters set the gear ration between the spindle and AC spindle motor.

> **NOTE**
> Set the gear ration between spindle and AC spindle motor when the spindle positioning is performed with serial spindle. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH1B.

4065	Position gain in servo mode (HIGH)
4066	Position gain in servo mode (MEDIUM HIGH)
4067	Position gain in servo mode (MEDIUM LOW)
4068	Position gain in servo mode (LOW)

[Data type] Word

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

[Valid data range] 0 to 32767

This parameter sets a servo loop gain in servo mode.
(spindle positioning, rigid tapping, etc.)

> **NOTE**
> When the spindle positioning by a serial spindle is performed, set the position control loop gain in place of parameter No. 4970. For which gear is used, it depends on the clutch/gear signal (serial spindle) CTH1A, CTH1B.

	#7	#6	#5	#4	#3	#2	#1	#0
4950	IMB	ESI	TRV			ISZ	IDM	IOR

[Data type] Bit

IOR Resetting the system in the spindle positioning mode

0 : Does not releases the mode.
1 : Releases the mode

IDM The positioning direction for the spindle using a M code is

0 : The positive direction
1 : The negative direction

ISZ When an M code for spindle orientation is specified in spindle positioning:

0 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode, and spindle orientation operation is performed.
1 : The spindle rotation mode is cleared and the mode is switched to the spindle positioning mode but spindle orientation operation is not performed.

TRV Direction of rotation of spindle positioning

0 : Normal
1 : Reverse

ESI Selection of a spindle positioning specification

0 : The conventional specificaion is used.
1 : The extended specificaion is used.

> **NOTE**
> The extended specification includes the following two extensions:
> - With the conventional specification, the number of M codes for specifying a spindle positioning angle is always 6. With the extended specification, an arbitrary number of such M codes from 1 to 256 can be selected by parameter setting (See parameter No. 4964.)
> - The maximum feedrate for spindle positioning (setting of parameter No. 1420) can be extended from 240000 to 269000 (unit: 10 deg/min).

IMB When the spindle positioning function is used, half–fixed angle positioning based on M codes uses:

0 : Specification A
1 : Specification B

> **NOTE**
> In the case of half–fixed angle positioning based on M codes, three types of spindle positioning operations can occur:
> (1) The spindle rotation mode is cleared, then the mode is switched to the spindle positioning mode.
> (2) Spindle positioning is performed in the spindle positioning mode.
> (3) The spindle positioning mode is cleared, then the mode is switched to the spindle rotation mode.
> In the case of specifiection A:
> Operations (1) to (3) are specified using separate M codes.
> (1)–Specified using M codes for performing spindle orientation.
> (See parameter No. 4960)
> (2)–Specified using M codes for specifying a spindle positioning angle. (See parameter No. 4962)
> (3)–Specified using M codes for clearing spindle positioning operation. (See parameter No. 4961.)
> In the case of specification B:
> When M codes for specifying a spindle positioning angle are specified, operations (1) to (3) are performed successively. (See parameter No. 4962.)

4960	M code specifying 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.

4961	M code releasing the spindle positioning mode

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 97

Set the M code to release the spindle positioning mode and to change the mode to the spindle rotating mode.

4962	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] Integer

[Valid data range] 6 to 92

Two methods are 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.

- When bit 6 (ESI) of parameter No. 4950=0

  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.
- When bit 6(ESI) of parameter No. 4950=1

  Set the start M code in this parameter, and set the number of M codes in parameter No. 4964. Then β M codes from Mα to M(α+β–1) are used for half fixed angle positioning.

The table below indicates the relationship between the M codes and positioning angles.

M code	Positioning angle	Example: Positioning angle when θ = 30°
Mα	θ	30°
M (α+1)	2θ	60°
M (α+2)	3θ	90°
M (α+3)	4θ	120°
M (α+4)	5θ	150°
M (α+5)	6θ	180°
⋮	⋮	⋮
M (α+n)	(n+1)θ	

> **NOTE**
> θ represents the basic angular diplacement set in pamrameter No. 4963.

4963	M code for specifying a spindle positioning angle

[Data type] Word

[Unit of data] deg

[Valid data range] 1 to 60

This parameter sets a basic angular displacement used for half–fixed angle positioning using M codes.

4964	Number of M codes for specifying a spindle positioning angle

[Data type] Byte

[Unit of data] Integer

[Valid data range] 0, 1 to 255

This parameter sets the number of M codes used for Half–fixed angle positioning using M codes.
As many M codes as the number specified in this parameter, starting with the M code specified in parameter No. 4962, are used to specify half–fixed angle positioning.
Let α be the value of parameter No. 4962, and let β be the value of parameter No. 4964. That is, M codes from Mα to M (α+β–1) are used for half–fixed angle positioning.

> **WARNING**
> Make sure that M codes from Mα to M (α+β–1) do not duplicate other M codes.

> **NOTE**
> 1 This parameter is valid when bit 6 (ESI) of parameter No. 4950=1.
> 2 Setting this parameter to 0 has the same effect as setting 6. That is, M code from Mα to M (α+5) are used for half–fixed angle positioning.

4970	Servo loop gain of the spindle

[Data type] Word

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

[Valid data range] 1 to 9999

Set the servo loop gain of the spindle in the spindle positioning mode.

4971	Servo loop gain multiplier of the spindle for gear 1
4972	Servo loop gain multiplier of the spindle for gear 2
4973	Servo loop gain multiplier of the spindle for gear 3
4974	Servo loop gain multiplier of the spindle for gear 4

[Data type] Word

Set the servo loop gain multipliers of the spindle for gears 1 to 4.

The multipliers are used to convert the amount of the position deviation to the voltage used in the velocity command. Assign the data obtained from the following equation to the parameters.

Loop gain multiplier = $2048000 \times E \times A/L$

where;

E: Voltage required to rotate the spindle motor at 1000 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. 4970 to No. 4974 are for analog spindle.

Alarm and message

Number	Message	Description
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Or comma (",") was specified with parameter No. 3405#4=1. Modify the program.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering or corner R command, I(K) is commanded for the X(Z) axis. Modify the program.
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attept was made for spindle indexing. Perform spindle orientation.
136	C/H–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program.
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
751	SPINDLE–1 ALARM DETECT (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	SPINDLE–1 MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Caution

CAUTION
1 Feed hold is invalid during spindle positioning.
2 Spindle positioning stops when emergency stop is applied; restart with orientation operation.
3 Dry run, machine lock, and auxiliary function lock are not available during spindle positioning.
4 The spindle positioning function and the serial spindle Cs contour control function cannot be used together. If both functions are specified, positioning has priority.
5 Specify parameter No. 4962 even if semi-fixed angle positioning is not used; otherwise M codes (M00 to M05) do not work.

Note

NOTE
1 Command spindle positioning with an independent block. X- and Y-axis positioning cannot be commanded to the same block.
2 Spindle positioning cannot be done by manual operation.
3 Automatic drift compensation is not effective for spindle positioning. To adjust the amount of drift compensation for each axis, set values manually and adjust the spindle amplifier to minimize the spindle motor rotation at a voltage of 0V. (parameter No. 3731). Insufficient adjustment causes poor positioning accuracy. Drift compensation is not needed with a serial spindle.
4 The machine coordinates for the spindle positioning axis are displayed in pulses units.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.9.5	SPINDLE POSITIONING FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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.1420)

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)

Explanations

The speed of the serial spindle is controlled by the spindle speed control function, while the spindle positioning is controlled by the Cs contouring control function ("spindle contour control"). Spindle speed control rotates the spindle using the velocity command, while the spindle contour control rotates the spindle using the move command.

Switching between spindle speed control and spindle contour control is performed by the DI signal from the PMC.

In the Cs contour control mode, the Cs contour control axis can be operated either manually or automatically, in the same way as normal servo axes.

(For a reference position return, see the relevant description in this section.)

Setting the Cs contour control axis

The axis used for Cs contour control must be set as an axis of the axes controlled by the CNC. Using parameter no. 1023, assign "−1" in the field corresponding to the chosen servo axis. Also set the spindle contour control axis as a rotation axis by setting ROTx of parameter No. 1006#0 and No. 1022.

Only one set of this setting can be used 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.1020. 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 A or C is used for the name of the contour axis, clear CCR (parameter no. 3405#4) to “0”.

Setting Axes that interpolate with Cs contour axis

Up to five servo axes can be specified for linear interpolation against the Cs contour control axis, by setting defined parameters :

- When no servo axis is used for interpolation, specify “0” in parameter nos. 3900, 3910, 3920, 3930, 3940.
- When one or more servo axes are used for interpolation, set the parameter for each as follows :

(1) Assign the axis number (1 to 8) to each of the servo axes used for interpolation in parameter nos. 39n0 (n=0, 1, 2, 3, or 4).

(2) Set the loop gain for each of the servo axes specified in (1) in parameter nos. 39n1, 39n2, 39n3, 39n4. The loop gain must be the position loop gain for the Cs contour control axis or a desired value. Four parameters are provided to correspond to the four gears of the spindle. Use those parameters according to the inputs of the serial spindle clutch /gear signal CTH1A, CTH2A <G70#3, #2>.

(3) When the number of servo axes to be used for interpolation is smaller than five, set “0” in remaining parameter nos. 39n0.

Switching spindle speed control/Cs contour control

❑ Switching from spindle speed control to Cs contour control

The serial spindle is put in the Cs contour control mode by setting the DI signal CON (G027#7) to “1”. When the mode is switched while the spindle is rotating, the spindle stops immediately to perform the change.

❑ Switching from Cs contour control to spindle speed control

Turning the DI signal CON (G027#7) to “0” puts the serial spindle in spindle speed control mode. Confirm that the move command for the spindle has been completed, then specify the switch. If it is switched while the spindle is moving, the machine enters interlock, or excessive position deviation alarm occurs.

Reference Position Return of Cs Contour Control Axis

After the serial spindle is switched from spindle speed control to Cs contour control mode, the current position is undefined. Return the spindle to the reference position.

The reference position return of the Cs contour control axis is as follows:

❑ In manual mode

After the serial spindle enters the Cs contour control mode, move the spindle in the direction of the reference position by turning on the feed axis and direction select signal (+Jn (G100) or –Jn (G102)). The spindle starts the movement to the reference position; when it reaches that position, the reference position return completion signal (ZPn (F094)) turns to “1”.
Turning any feed axis and direction select signal to “1” moves the tool in the reference position direction.

❑ In the automatic mode

After the serial spindle enters the Cs contour control mode, the spindle returns to the reference position when G28 is specified. Under certain conditions, the G00 command returns the spindle to the reference position, depending upon the setting of parameter NRF no. 3700#1:

(i) G00 command

Returning to the reference position using the G00 command differs from using the G28 command or the manual method. The serial spindle can be positioned at any point using the G00 command, while the latter two methods always return the serial spindle to the reference position.

When parameter NRF no. 3700#1 is “0” and the serial spindle is put in the Cs contour control mode, if the G00 command is given before returning the spindle to the reference position, the serial spindle returns to the reference position and indexes it before moving to the commanded position. After positioning at the reference position, the reference position return completion signal (ZPn(F094)) turns to “1”. When the G00 command is issued after the serial spindle has returned to the reference position at least once, normal positioning operation is executed.

(ii) G28 command

After the serial spindle is put in the Cs contour control mode, issuing the G28 command stops the spindle motor, then moves the spindle to the midpoint. The spindle then returns to the reference position. At this point, the reference position return completion signal (ZPn F094) turns to “1”. When the serial spindle has returned to the reference position once while in the Cs contour control mode, the G28 command positions the spindle at the reference position without moving to the midpoint and ZPn comes on.

❑ Interruption of reference position return

(i) Manual operation

Return to the reference position can be interrupted by resetting, emergency stop, or turning off the feed axis and direction select signal. When the interrupted return operation is resumed, start from the beginning.

(ii) Automatic operation

Return to the reference position can be interrupted by resetting, emergency stop, or feed hold. When the interrupted return operation is resumed, start from the beginning.

Operation of Cs contour control axis (Manual/Automatic)

If a reference position return is performed on the Cs contour control axis, the axis can be operated in the same way as a normal NC axis.

In the spindle speed control mode, on the other hand, it does not operate as the Cs contour control axis, and P/S alarm 197 occurs during automatic operation.

In the spindle speed mode, inhibit manual operation of the Cs contour control axis using the PMC ladder.

Display of Position Error of Cs Contour Control Axis

DGN No.

418	Position deviation amount of 1st spindle

Position deviation amount of the position loop for the 1st spindle.

This diagnostic display shows information obtained from the serial spindle control unit. This diagnosis displays position error of the spindle contour axis during spindle contour control.

The position error can also be checked using a servo error display (DGN of No. 300x) for an axis under Cs contour control.

Signal

Spindle contour control change signal CON <G027#7>

[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.

When this signal turns to ”1”, the spindle mode changes from speed control to Cs contour control.

If the spindle is moving at the time of the change, it stops immediately. Turning the signal to “0” changes the spindle mode from Cs contour control back to speed control.

Spindle contour control change completion signal FSCSL <F004#1>

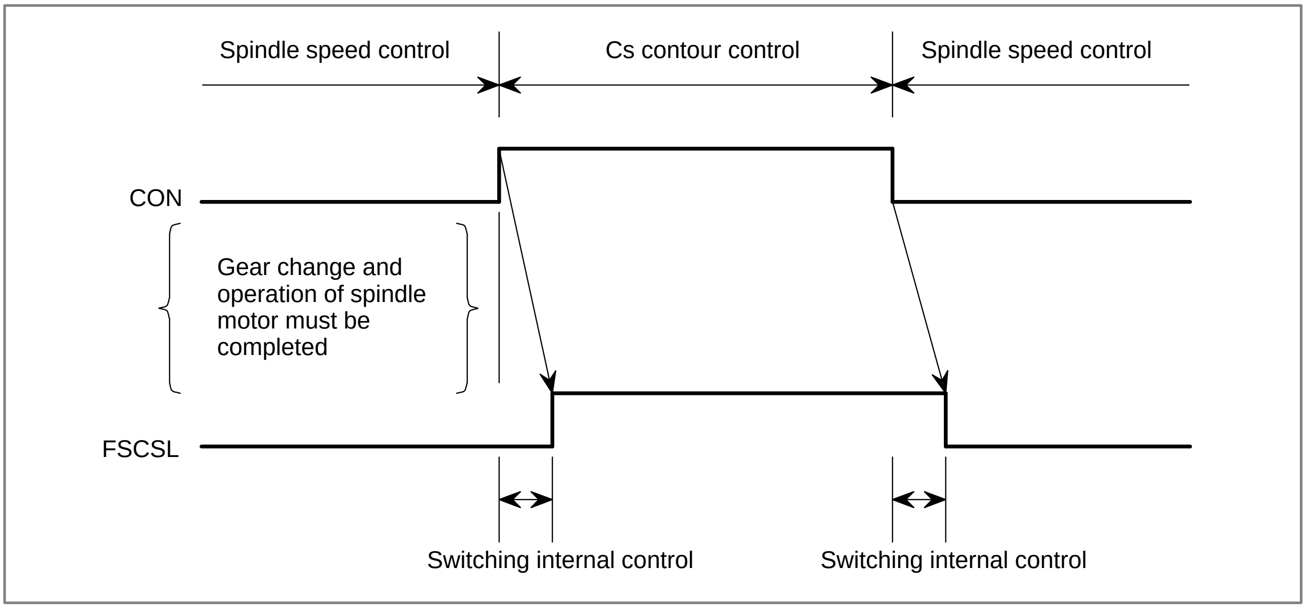
[Classification] Output signal

[Function] This signal indicates the axis is under Cs contour control.

[Output condition] Spindle speed control mode → 0

Cs contour control mode → 1

Time Chart



> **NOTE**
> Any mechanical gear change needed and inputs for GR1, GR2, CTH1A, and CTH2A must be completed before the CON signal selects Cs contour control mode.
> A servo excessive error may be generated if the spindle motor is not ready for operation. (Signal SRVA, SFRA <G070#4, #5> or other required signals must be appropriately processed on the machine side).

Other signals

Gear select signal (Input)
GR1, GR2, <G028#1, #2>
Gear select signal (Output)
GR3O, GR2O, GR1O
<F034#2, #1, #0>
(M series)

Refer to 9.3 “Spindle speed Control”.

Clutch/Gear signal (Serial spindle)
CTH1A, CTH2A
<G070#3, #2>

Refer to the manual of serial spindle.

These signals determine what parameter (loop gain, etc.) to be used for each gear position.

CTH1A and CTH2A are the gear select signals for the serial spindle, but GR1 and GR2 must also be set. Do not change these signals while in the Cs contour control mode.

Relationship between gears selected and spindle gear select signals

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, and parameter No. 3706#4 GTT=0, GR1 and GR2 do not need to be input. Input CTH1A and CTH2A when gears are changed using GR1O, GR2O and GR3O.

Cs contour control axis reference position return completion signal ZPx <F094>

[Classification] Output signal

[Function] This signal indicates that a reference position return has been made for the Cs contour control axis.

[Output condition] If a manual reference position return or automatic reference position return by G28 is performed during the Cs contour control mode, this signal becomes logical 1 when the Cs contour control axis reaches the reference position.

Signals on manual operation

Feed axis and direction select signal +Jn, –Jn <G100, G102> (Input)
Manual handle feed axis select signal HSnA, HSnB, HSnC, HSnD <G018, G019> (Input) (Refer to respective items in this manual)

The Cs contour control axis can be manually operated in the same way as normal servo axes, except for a manual reference position return. In the spindle speed control mode, however, manual operations for the Cs contour control axis must be inhibited using the PMC ladder, etc.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027	CON							
G028						GR2	GR1	
G070	MRDYA		SFRA	SRVA	CTH1A	CTH2A		

	#7	#6	#5	#4	#3	#2	#1	#0
F034						GR3O	GR2O	GR1O
F044							FSCSL	
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1

Parameter

The following describes major parameters.

In addition, parameters such as axis feedrate, acceleration/deceleration, and display can be used. Also, digital servo parameters (No. 2000 ~) for the Cs contour axis are not required to be set.

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

[Data type] Bit axis

Type of controlled axis

0 : Linear axis
1 : Rotation axis

NOTE
Inch/metric conversion cannot be made to the rotation axis. The machine coordinate values are rounded in 0 + O 360 deg. Automatic reference position return (G28, G30) is made in the manual reference position return direction and the move amount does not exceed one rotation.

<u>Set 1 as the rotation axis to the Cs contour control axis.</u>

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

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

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

NOTE
1 In the T series, when G code system A is used, neither U, V, nor W can be used as an axis name. Only when G code system B or C is used, U, V, and W can be used as axis names.
2 The same axis name cannot be assigned to more than one axis.
3 When the second auxiliary function is provided, address B cannot be used as an axis name. In the T series, when address A or C is used, set parameter CCR (No. 3405#4) to 0.

Any axis name can be used for Cs contour control axis except for above limitation.

1022	Setting of each axis in the basic coordinate system

[Data type] Byte axis

Only one axis can be set for each of the three basic axes X, Y, and Z, but two or more parallel axes can be set.

Set value	Meaning
0	Neither the basic three axes nor a parallel axis
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

Set 0 to the Cs contour control axis.

1023	Number of the servo axis for each axis

[Data type] Byte axis

Set the servo axis for each control axis.

Generally, the same number shall be assigned to the control axis and the corresponding servo axis.

Set –1 as the number of servo axis to the Cs contour control axis.

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

[Data type] Bit

ZPR Automatic setting of a coordinate system when the manual reference position return is performed

0 : Not set automatically
1 : Set automatically

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

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

1420	Rapid traverse rate for each axis

[Data type] Word axis

[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 each axis.

1620	Time constant used in linear acceleration/deceleration or bell–shaped acceleration/ deceleration in rapid traverse for each axis

[Data type] Word axis

[Unit of data] ms

[Valid data range] 0 to 4000

1820	Command multiply for each axis (CMR)

[Data type] Byte axis

- When command multiply is 1/2 to 1/27

$$\text{Set value}=\frac{1}{(\text{Command multiply})}+100 \text{ [Valid data range: 102 to 127]}$$

- When command multiply is 0.5 to 48

Set value = 2 × command multiply [Valid data range: 1 to 96]

1826	In–position width for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

Set the in–position width for each axis.

1828	Positioning deviation limit for each axis in movement

[Data type] Two–word axis

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Set the positioning deviation limit in movement for each axis.

1829	Positioning deviation limit for each axis in the stopped state

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] 0 to 32767

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

1851	Backlash compensation value used for rapid traverse for each axis

[Data type] Word axis

[Unit of data] Detection unit

[Valid data range] −9999 to +9999

Set the backlash compensation value for each axis.

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

[Data type] Bit

NRF The first positioning command by G00 after the serial spindle is switched to Cs axis contouring control performs:

0 : Positioning after returning to the reference position.
1 : Normal positioning

3900	The number of servo axis that interpolates with Cs contour control axis

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (1st group)

NOTE
Set 0 when there is no servo axis that interpolates with Cs contour control axis.

3901	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3902	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3903	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3904	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (1st group)

3910	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (2nd group)

> **NOTE**
> When there is no servo axis or only one servo axis that interpolates with Cs contour control axis, set this parameter to 0.

3911	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3912	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3913	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3914	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (2nd group)

3920	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (3rd group)

> **NOTE**
> When there is no servo axis or less than three servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3921	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3922	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3923	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3924	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (3rd group)

3930	Number of servo axis that interpolates with Cs contour control

[Data type] Byte

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (4th group)

NOTE

When there is no servo axis or less than four servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3931	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3932	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3933	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3934	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (4th group)

3940	Number of servo axis that interpolates with Cs contour control

[Data type] Byte axis

[Valid data range] 0 to 8

Set the number of servo axis that interpolates with Cs contour control axis (5th group)

> **NOTE**
> When there is no servo axis or less than five servo axes that interpolates with Cs contour control axis, set this parameter to 0.

3941	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (High gear)
3942	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium high gear)
3943	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Medium low gear)
3944	Loop gain of the servo axis that interpolates with Cs contour control axis during interpolation (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the servo loop gain of the servo axis that interpolates with Cs contour control axis on each spindle gear (5th group)

4056	Gear ratio (HIGH)
4057	Gear ratio (MEDIUM HIGH)
4058	Gear ratio (MEDIUM LOW)
4059	Gear ratio (LOW)

[Data type] Word axis

[Unit of data] (Number of motor rotations to one spindle rotation) × 100

[Valid data range] 0 to 32767

Set the gear ratio between spindle and AC spindle motor.

> **NOTE**
> For which gear ratio is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4069	Position gain at Cs contour control (High gear)
4070	Position gain at Cs contour control (Medium High gear)
4071	Position gain at Cs contour control (Medium Low gear)
4072	Position gain at Cs contour control (Low gear)

[Data type] Word axis

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

[Valid data range] 0 to 32767

Set the position gain at Cs contour control.

> **NOTE**
> For which position gain is used in actual spindle operation, it depends on clutch/gear signal (serial spindle) CTH1A, CTH2A.

4135	Grid shift value at Cs contour control

[Data type] Two–word

[Unit of data] 1 pulse unit (360000 p/rev)

[Valid data range] –360000 to +360000

Set the number of pulses from an issue of one–rotation signal to the machine zero point in Cs contour control.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs contour control axis when the signal CON(G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

Warning

> **WARNING**
> In the spindle contour control mode, do not switch the spindle gears. When the gears need to be changed put the system in the spindle speed control mode first.

Note

NOTE
In the T series machines, the spindle contour control function and the spindle positioning function cannot be used at the same time. If both functions are specified simultaneously, the spindle positioning function takes precedence.

Reference item

FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.6	Cs Contour Control Function
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)	2.4	Cs Contour Control Function Start–up Procedure

9.10 MULTI–SPINDLE CONTROL

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.

Also, the maximum spindle speed can be set to each spindle to clamp the spindle speed of each spindle (set by parameters No. 3772, 3802 and 3822).

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.

The spindle serial output option is required to use multi-spindle control.

Difference in multi–spindle control between the M and T series

- For the M series, multi–spindle control is possible only when spindle gear selection type T is selected (when the constant surface speed control option is provided, or when GTT (bit 4 of parameter No. 3706) is set to 1).
- For the M series, rigid tapping spindle selection signals RGTSP1, RGTSP2, and RGTSP3 (G061#4, #5, and #6, when bit 7 of parameter No. 5200 is set to 1) cannot be used. For details of rigid tapping, see Section 9.11.
- When two–path control is performed with the M series, spindle commands and position coder feedback signals cannot be changed between the paths (spindle command select signals SLSPA and SLSPB <G063 #2 and #3>, and spindle feedback select signals SLPCA and SLPCB <G064 #2 and #3> are not available).

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 <G027#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 <G027#3-#5>) to stop its rotation; an unused spindle can still be stopped.

There is a spindle enable signal to control each spindle; ENB <F001#4> controls the first spindle, while ENB2 and ENB3 <F038#2, #3> control the second and third spindles, respectively.

The PMC signal PC2SLC <G028#7> is used to select between the first and second position coders.

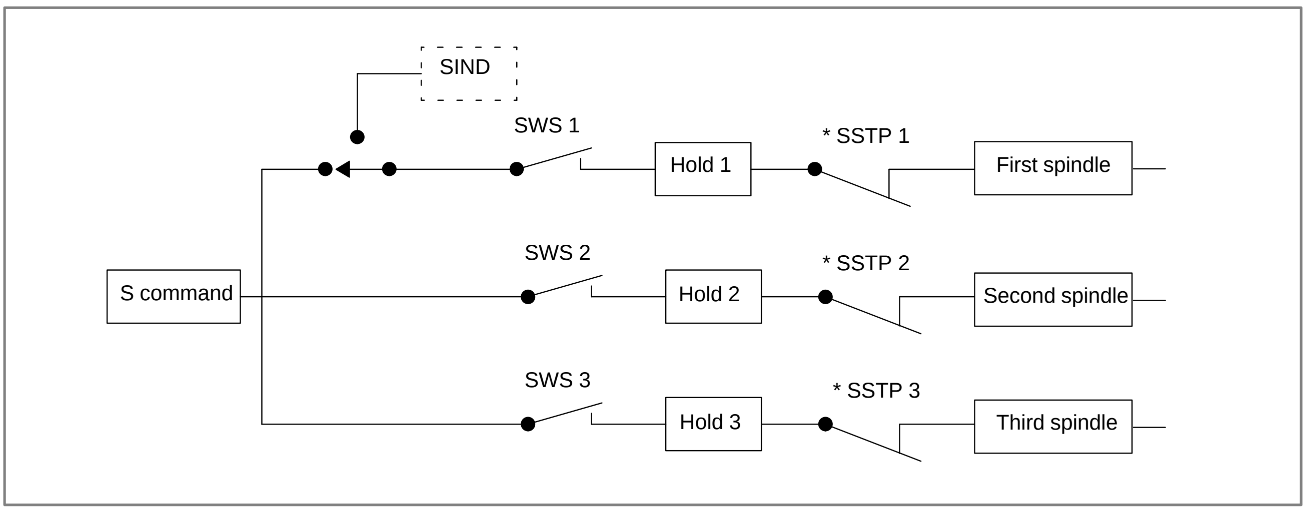
Multi-spindle control (TYPE-A)

When parameter MSI (No. 3709#2)=0, TYPE-A is used.

When the first spindle is selected with the SWS1 signal, the SIND signal <G033#7> is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I <G0033#3 to G0032#0> are used to set that spindle's analog voltage. These signals do not affect the second and third spindles.

The PMC-based polarity (rotation direction) control signals SGN and SSIN <G033#5,#6> will function for any spindle selected by SWS1 to SWS3.

The concept of Type A multi-spindle control is outlined below.



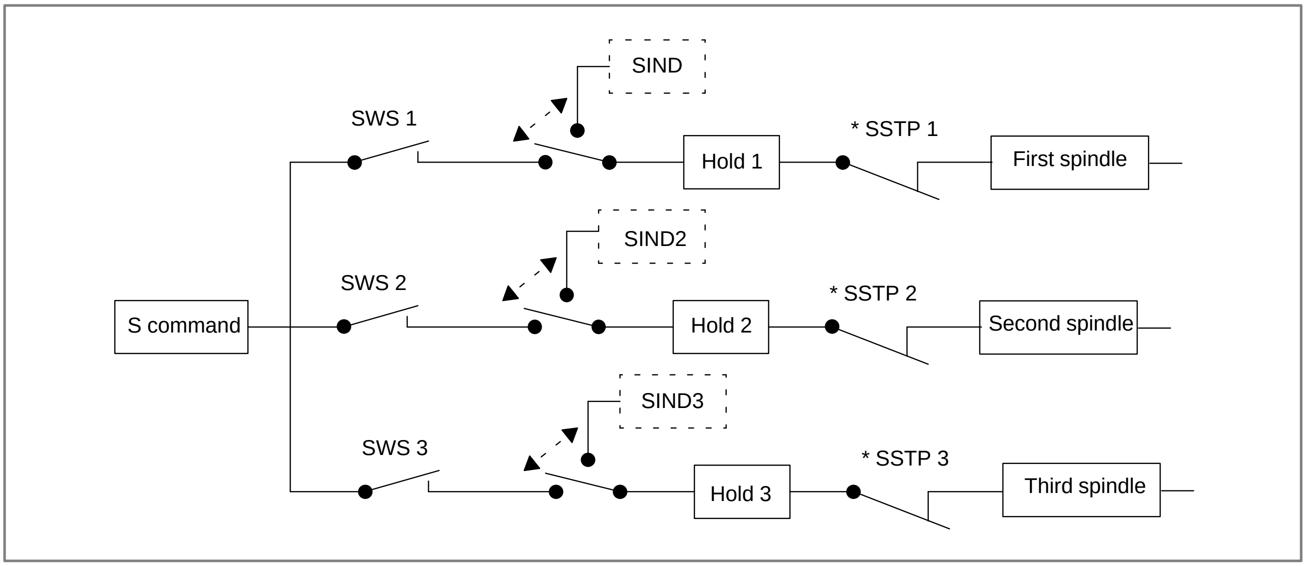
Multi-spindle control (TYPE-B)

Select Type B control by setting parameter MSI No. 3709#2 to "1".

Each spindle has its own SIND, SSIN and SGN signals.
Each signal functions regardless of selection state of the spindle selection signals (SWS1 to SWS3).

When either the spindle selection signal or the SIND signal for the first, second or third spindle is set to "1", the polarity control signals SSIN, SGN will function. The first spindle is controlled by the SGN and SSIN signals; the second and third spindles are controlled by SGN2, SGN3 <G035#5, G037#5>, and SSIN2, SSIN3 <G035#6, G037#6>, respectively.

The concept of Type B multi-spindle control outlined below.



Spindles to be controlled

In multi-spindle control, the first spindle is the first serial spindle, the second spindle is the second serial spindle, and the third spindle is an analog spindle.

A configuration is possible which does not connect the second or third spindles.

Connection of spindle

Spindle configuration when multi-spindle control is used:

Necessary option and parameter	· Multi-spindle control · Spindle serial output · Parameter SS2 (No. 3701#4) = 1 (to use second spindle) · Spindle analog output (to use third spindle)
Connection of each spindle	First spindle → Connect on CNC mother board. Second spindle → Connect on first spindle control unit. Third spindle → Connect on CNC mother board.
Connection of each position coder	First position coder: Feedback information obtained by position coder or equivalent sensor connected to first spindle control unit is fed to CNC via serial interface. Second position coder: Feedback information obtained by position coder or equivalent sensor connected to second spindle control unit is fed to CNC via serial interface through 1st spindle control unit Note) When second spindle is not connected, second position coder cannot be used.

For detailed information about serial spindle connection, refer to the manuals on the serial spindles.

Relationship with other 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 (SWS 1–3) 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 (SWS 1–3).

- **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. 3709#5)=1, the difference of the feedback pulses between the 1st and 2nd position coder can be output irrespective of the state of 2nd position coder selection signal (PC2SLC).

- **Spindle positioning or Cs contour control**

When the spindle motor is used for positioning, as in the case of spindle positioning or Cs contour control, the first spindle 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 (T series)**

Polygon turning rotates a tool axis in phase with the spindle. To perform polygon turning when multi-spindle control issued, select the spindle and the position coder associated with the spindle.

- **Spindle synchronization, polygon turning between spindles, simplified synchronization control**

During spindle synchronization, polygon turning between spindles 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 (SWS 1–3), rigid tapping can use either the first or second spindle as the rigid tap spindle. There are certain restrictions:

- Set the SWS 1 to 3 signals before directing rigid tapping;
- Do not switch the SWS 1 to 3 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.

- **Two-path control option**

Refer to 9.4 “Spindle Control for Two-path Lathe”.

Signal

Spindle Selection Signal SWS1, SWS2, SWS3 <G027#0, #1, #2>

[Classification] Input signal

[Function] Controls whether S command specified to the NC is output to the spindle or not in multi-spindle.

SWS1 1 : Outputs a speed command to the first spindle.
0 : Outputs no speed command to the first spindle.

SWS2 1 : Outputs a speed command to the second spindle.
0 : Outputs no speed command to the second spindle.

SWS3 1 : Outputs a speed command to the third spindle.
0 : Outputs no speed command to the third spindle.

Individual spindle stop signal *SSTP1, *SSTP2, *SSTP3 <G027#3, #4, #5>

[Classification] Input signal

[Function] Effective only to multi-spindle, each spindle can be stopped by this signal.

***SSTP1** 1 : Does not set 0 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 select signal GR21 <G029#0> GR31 <G029#2>

[Classification] Input signal

[Function] Gear selection signals for 2nd and 3rd spindle when multi–spindle is equipped (2–stage). Use GR1 and GR2 <G028#1, #2> for the 1st spindle and up to 4–stage gears can be used.

GR21 1 : Selects the second-stage gear for the second spindle.
0 : Selects the first-stage gear for the second spindle.

GR31 1 : Selects the second-stage gear for the third spindle.
0 : Selects the first-stage gear for the third spindle.

2nd position coder selection signal PC2SLC <G028#7>

[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<F038#2> ENB3<F038#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, <G033#7, #6, #5> (Input)
R12I~R01I<G033#3~G032#0> (Input)

2nd spindle SIND2, SSIN2, SGN2, <G035#7, #6, #5> (Input)
R12I2~R01I2<G035#3~G034#0> (Input)

3rd spindle SIND3, SSIN3, SGN3, <G037#7, #6, #5> (Input)
R12I3~R01I3<G037#3~G036#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 2 (MSI) of parameter No. 3709 is set to 0), the signals for the second and third spindles cannot be used.

For details of each signal, see Section 15.4.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G027			*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029		*SSTP				GR31		GR21
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3

	#7	#6	#5	#4	#3	#2	#1	#0
F038					ENB3	ENB2		

Parameter

The parameters for the 1st spindle and the 1st position coder are the same as usual. This section describes the parameters which are added by this function.

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit

SS2 The number of connections in serial spindle control

0 : 1
1 : 2

	#7	#6	#5	#4	#3	#2	#1	#0
3702							EMS	

[Data type] Bit

EMS Multi–spindle control is

0 : Used
1 : Not used

NOTE
If the multi–spindle control function is not required for one path in two–path control, specify this parameter for the path to which the multi–spindle control function need not be applied.

	#7	#6	#5	#4	#3	#2	#1	#0
3706					PCS			
				GTT				

[Data type] Bit

PCS When multi–spindle control is applied to two tool posts in two–path control, this parameter specifies whether a position coder feedback signal from the other tool post is selectable, regardless of the state of the PC2SLC signal (bit 7 of G028/bit 7 of G1028) of the other tool post:

0 : Not selectable.
1 : Selectable. (To select a position coder for the other tool post, the SLPCA signal (bit 2 of G064) and the SLPCB signal (bit 3 of G064) are used.)

> **NOTE**
> Multi–spindle control based on the same serial spindle must be applied to both tool posts. Refer to 9.4 for details.

GTT Selection of a spindle gear selection method

0 : Type M.
1 : Type T.

> **NOTE**
> 1 Type M:
> The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in parameters. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
> Type T:
> The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.
> 2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.
> 3 When type T spindle gear switching is selected, the following parameters have no effect:
> No. 3705#2 SGB, No. 3751, No. 3752, No. 3705#3 SGT, No. 3761, No. 3762, No. 3705#6 SFA, No. 3735, No. 3736
> However, parameter No. 3744 is valid.

		#7	#6	#5	#4	#3	#2	#1	#0
3707								P22	P21

P22, P21 Gear ratio of spindle to second position coder

Magnification	P22	P21
×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
3709							MSI		

MSI In multi–spindle control, the SIND signal is valid

0 : Only when the first spindle is valid (SIND signal for the 2nd, 3rd spindle becomes ineffective)

1 : For each spindle irrespective of whether the spindle is selected (Each spindle has its own SIND signal).

3772	Maximum spindle speed

[Data type] Word type

[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.

NOTE

1 When the constant surface speed control option is selected, the spindle speed is clamped at the maximum speed, regardless of whether the G96 mode or G97 mode is specified.

2 When the multi–spindle control option is selected, set the maximum speed for each spindle in the following parameters:
Parameter No. 3772: Sets the maximum speed for the first spindle.
Parameter No. 3802: Sets the maximum speed for the second spindle.
Parameter No. 3822: Sets the maximum speed for the third spindle.

3802	Maximum speed of the second spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

Parameter sets the maximum speed for the second spindle.
When a command specifying a speed exceeding the maximum speed of the spindle is specified, or the speed of the spindle exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

WARNING

1 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used.
When 0 is set in parameter No. 3772, the spindle speed is not clamped.

2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the spindle speed is not clamped.

NOTE

1 This parameter is valid when the multi–spindle control option is selected.

2 When the constant surface speed control option is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is specified.

3811	Maximum spindle speed for gear 1 of the second spindle
3812	Maximum spindle speed for gear 2 of the second spindle

[Data type] Word

[Unit of data] 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.

3820	Data for adjusting the gain of the analog output of the third–spindle speed

[Data type] Word

[Unit of data] 0.1%

[Valid data range] 700 to 1250

Set the data used for adjusting the gain of the analog output of the third spindle speed.

> **NOTE**
> This parameter is used for controlling the multi–spindles.

3821	Offset voltage compensation value of the analog output of the third spindle speed

[Data type] Word

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset voltage compensation value of the analog output of the third spindle speed.

1) Set 0 (standard setting) to this parameter.
2) Command a spindle speed that makes the spindle speed analog output 0.
3) Measure output voltage.
4) Set the following value to parameter No. 3821.

$$\text{Setting value} = \frac{-8191 \times \text{offset voltage (V)}}{12.5}$$

5) After the parameter has been set, command a spindle speed whose analog output becomes 0 and confirm the voltage becomes 0V.

3822	Maximum speed of the third spindle

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

This parameter sets the maximum speed for the third spindle.

When a command specifying a speed exceeding the maximum spindle speed is specified, or the spindle speed exceeds the maximum speed because of the spindle speed override function, the spindle speed is clamped at the maximum speed set in the parameter.

WARNING

1 When 0 is set in this parameter, the setting of parameter No. 3772 for the first spindle is used. When 0 is set in parameter No. 3772, the spindle speed is not clamped.

2 When spindle speed command control is applied using the PMC, this parameter has no effect, and the speed of the spindle is not clamped.

NOTE

1 This parameter is valid when the multi–spindle control option is selected.

2 When the constant surface speed control option is selected, the spindle speed is clamped at the specified maximum speed, regardless of whether the G96 mode or G97 mode is set.

3831	Maximum spindle speed for gear 1 of the third spindle
3832	Maximum spindle speed for gear 2 of the third spindle

[Data type] Word

[Unit of data] 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 <G028#7> cannot be used while, for instance, a command for feed per rotation or thread cutting is taking place.

Caution

CAUTION

1 If the primary spindle stop signal *SSTP for stopping all selected (SWS1 to SWS3) spindles' rotation is cleared, the speed command is restored. A spindle not selected by SWS1 to SWS3 and rotating at its previous speed, which is stopped using its respective command *SSTP1 to *SSTP3, cannot be restored to that speed when the signal is cleared.

2 Type A multi-spindle control differs from Type B in the relationship between the SWS1 and SIND signals for the first spindle. In Type B, SIND functions only when SWS1 is set to "1". In Type A, SIND functions whether SWS1 is "1" or "0"; each spindle is selected by either of its respective SWS1 or SIND signals being set to "1".

Note

NOTE

1 The spindle orientation signal, spindle speed override signals, and spindle stop signal *SSTP only function for 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
- Parameters used according to the number of gear stages (No. 5221 to No. 5224, No. 5231 to No. 5234, No. 5241 to No. 5244, No. 5261 to No. 5264, No. 5271 to No. 5274, No. 5281 to No. 5284, No. 5291 to 5294, No. 5321 to No. 5324, etc.)
- M series: Up to three stages T series: Up to four stages
  (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 When M–type gear selection is used for the M series, the maximum spindle speed for rigid tapping (specified with parameters No. 5241 to 5243) must also be set for parameter No. 5243 regardless of the number of gear steps. (For a system having a single gear step, set the same value as that of parameter No. 5241 for parameter No. 5243. For a system having two gear steps, set the same value as that of parameter No. 5242 for parameter No. 5243.)

The descriptions given in this section (such as spindle gear switching and M–type/T–type) are based on the explanation given in Section 9.3. Refer to Section 9.3 as necessary.

Specification of M series/T series

The differences in the specifications for rigid tapping for the M series and T series are described below.

- **Rigid tapping of M series**

The tapping cycle G84 and the reverse tapping cycle G74 can be used to specify M series rigid tapping.

A tapping axis can be arbitrarily selected from the basic axes X, Y, and Z, as well as axes parallel to the basic axes, by setting the corresponding parameters accordingly (bit 0 (FXY) of parameter No. 5101).

The spindle operations of G84 and G74 are reversed with respect to each other.

- **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).

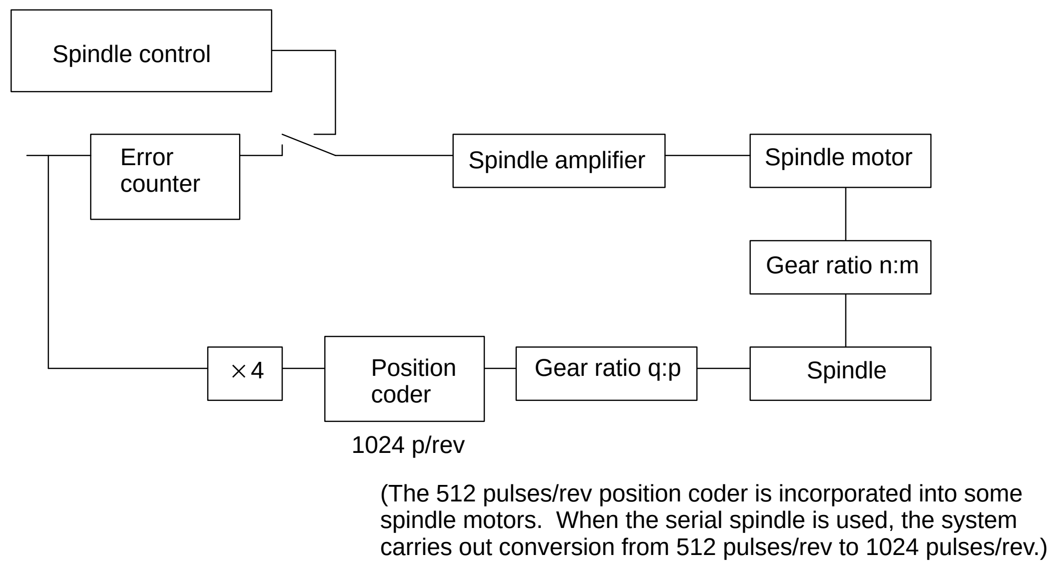
A reverse tapping cycle, like that supported by M series, is not available.

For a two–path lathe, 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. For example, rigid tapping in combination of a tapping axis of tool post 1 and the spindle of tool post 2, by issuing a tapping command to tool post 1, is not supported.

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 (1 st spindle of M series) or four gear stages (1st spindle of T series), two gear stages (2nd spindle) 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 No. 5221 to No. 5223 and No. 5231 to No. 5233, or parameter set PG1 and PG2 No. 3706 #0, #1, parameter P21, P22 (No. 3707 #0, #1) for 2nd spindle. Which parameter set to use is specified by parameter VGR No. 5200 #1.

- **Arbitrary gear ratio (VGR=1)**

This is used if the gear ratio for the spindle motor and position coder (built-in or separate) is not 1:1, 1:2, 1:4, or 1:8, set VGR to 1 and set the gear ratio using parameter No. 5221 to 5224.

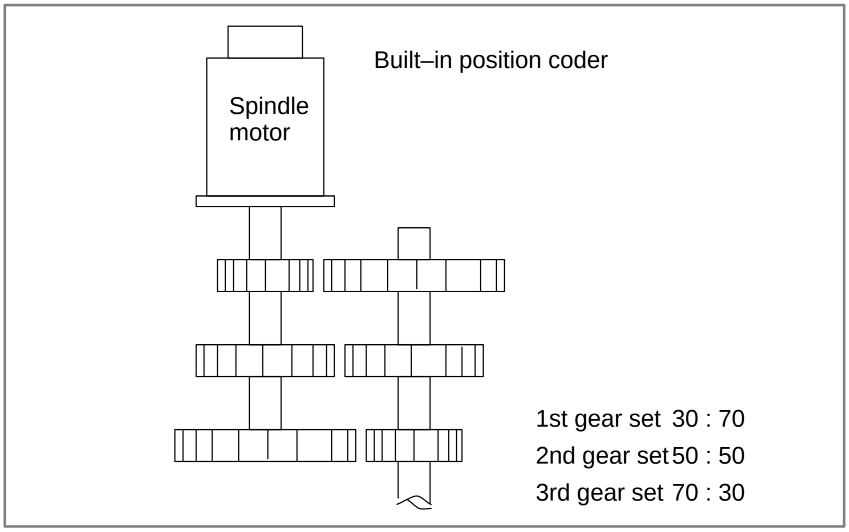
When position coder is mounted on a spindle, the gear ratio for the spindle motor and position coder cannot be changed by shifting the spindle motor and spindle gears. Parameters No. 5221 to 5224 must all specify the same value for the teeth of the individual spindle gears. Parameters No. 5231 to 5234 must all specify the same value for the teeth of individual position coder gears.

The 1024 or 512 pulses/rev position coder is built into the spindle motor. For the 512 pulses/rev version, specify double the number of teeth on each gear for the position coder. (Double the number of teeth need not be specified for the serial spindle.)

The M series allows up to three stages, regardless of which gear selection method has been selected. (Parameter Nos. 5224 and 5234 cannot be used.)

The T series supports up to four stages. (Set parameter Nos. 5221 to 5224 and 5231 to 5234.) When the multi–spindle function is used to perform rigid tapping with the second spindle, up to two stages are supported. (Set parameter Nos. 5221, 5222, 5231, and 5232.)

Example)



Parame-ter No.	Set value		Meaning
	512p/rev Position coder	1024p/rev Position coder	
5221	70		Number of teeth of the 1st gear for the spindle side
5222	50		Number of teeth of the 2nd gear for the spindle side
5223	30		Number of teeth of the 3rd gear for the spindle side
5231	60 Note)	30	Number of teeth of the 1st gear for the position coder side
5232	100 Note)	50	Number of teeth of the 2nd gear for the position coder side
5233	140 Note)	70	Number of teeth of the 3rd gear for the position coder side

NOTE

Double value setting is not required for serial spindle.

- **Gear ratio is 1:1, 1:2, 1:4, 1:8 (VGR=0)**

If the gear ratio is either 1:1, 1:2, 1:4, and 1:8, it is set using parameters PG1 and PG2 (No. 3706 #0, #1). This applies if the position coder is mounted in a spindle or built into a spindle motor when only one stage gear is provided. .
For 2nd spindle, set it to parameter P21, P22 (No.3707#0, #1).

Parameter		Gear ratio		Detection unit
PG2	PG1	Spindle	Position coder	
0	0	1	1	360/4096=0.08789 deg
0	1	1	2	360/4096×2=0.17578 deg
1	0	1	4	360/4096×4=0.35156 deg
1	1	1	8	360/4096×8=0.70313 deg

The spindle motor building in the 512 pulses/rev position coder uses the values set forth in the following table. A serial spindle does not require double-value setting; use the same values as for the spindle motor building in the 1024 pulses/rev position coder.

<table>
<tr><th rowspan="5">Built–in position coder 512p/rev</th><th colspan="2">Gear ratio</th><th colspan="2">Parameter</th><th rowspan="2">Gear ratio of spindle to position coder</th><th rowspan="2">Detection unit (deg)</th></tr>
<tr><th>Spindle motor</th><th>Spindle</th><th>PG2</th><th>PG1</th></tr>
<tr><td>1</td><td>1</td><td>0</td><td>1</td><td>1:2</td><td>0.17578</td></tr>
<tr><td>2</td><td>1</td><td>1</td><td>0</td><td>1:4</td><td>0.35156</td></tr>
<tr><td>4</td><td>1</td><td>1</td><td>0</td><td>1:8</td><td>0.70313</td></tr>
</table>

(3) Rigid tapping and machines with multiple gears

If the M type gear selection method is selected, the CNC determines whether gears need changing using the gear change specification mentioned in section 9.3. If the gears need to be changed, the CNC generates the S function code read signal SF (F007#2) and gear selection signals GR10, GR20, and GR30 (F034#0-#2) to notify the PMC. Change gears using the PMC, based on these signals.

If the T type gear selection method is selected, the CNC does not process gear changes. When the CNC has the S function code, it outputs signal SF and S function code signals S00 to S31 (F022#0-F025#7) to the PMC. (However, parameter No. 3705 and its related parameters need to be set for S code and SF signal output). Using the PMC, determine whether gears need changing, and make the change if needed. Input gear selection signals GR1 and GR2 <G028#1,#2> or GR21 <G029#0> for 2nd spindle for the selected gear, and notify the CNC of them.

To perform rigid tapping with the serial spindle, enter the clutch/gear selection signals CTH1 and CTH2 (G070#3,#2 for the first spindle, and G074#3, #2 for the second spindle) from the PMC. Notify the serial spindle control unit of these signals via the CNC, irrespective of the gear selection method.

Changing gears during rigid tapping requires a different process from that for gear changes during normal machining. As described above, changing gears conforms to the gear change specifications mentioned in section 9.3 when the M type gear selection method has been selected. With the T type gear selection method, changing gears conforms to the logic programmed in the PMC.

Regardless of the option's selection, if the range in which the spindle speed specified by the S function code does not correspond to the currently selected gear, the gears are changed.

The following tables list the spindle speed ranges for each gear during normal machining (assuming no machine restrictions) and rigid tapping:

Gear	Spindle speed range (normal machining)	
	Lower limit	**Upper limit**
Low–speed gear	1 revolution	Maximum low–speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{Medium speed gear ratio}}$
High–speed gear	Maximum medium–speed gear speed +1 revolution	Maximum high–speed gear speed = $\frac{\text{Maximum spindle motor speed} \times L\%}{\text{High–speed gear ratio}}$

NOTE

This table shows an example of three gears. L% indicates a spindle motor protection constant (up to 100). L can be specified for each gear using method B for changing in M type gear selection method (bit 2 (SGB) of parameter No. 3705 =1).

Gear	Spindle speed range (during rigid tapping)	
	Lower limit	**Upper limit**
Low–speed gear	1 revolution	Maximum low–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Low–speed gear ratio}}$
Medium speed gear	Maximum low–speed gear speed + 1 revolution	Maximum medium–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{Medium–speed gear ratio}}$
High–speed gear	Maximum medium–speed gear speed +1 revolution	Maximum high–speed gear speed $= \frac{\text{Basic spindle motor speed} + \alpha}{\text{High–speed gear ratio}}$

NOTE
This table show an example of three gears. For the basic spindle motor speed, refer to the spindle motor description manual. "+ 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 3 (SGT) of parameter No. 3705 = 1) in the tapping cycle to specify the following:

The table above shows the maximum low-speed gear speed during rigid tapping for low-/medium-speed gear change position D (parameter No. 3761).

The table above shows the maximum medium-speed gear speed during rigid tapping for medium-/high-speed gear change position E (parameter No. 3762).

If the T type gear selection method is used, add the rigid tapping logic to the logic programmed in the PMC.
See Section 9.3, "Spindle Control" for details of the spindle gear change specifications.

The loop gain can be specified for each gear. Specify "0" for parameter No. 5280 and specify loop gains for each gear for parameter Nos. 5281 to 5284. Unless "0" is specified for parameter No. 5280, the loop gains for each gear are disabled, and the loop gain common to all gears, the value of parameter No. 5280, is enabled.

Specify the time constant and the maximum spindle speed for each gear. Use parameters Nos. 5261 to 5264 to specify the time constant.
Use parameters Nos. 5241 to 5244 to specify the maximum spindle speed.

For M type gear selection method, set the maximum spindle speed to parameter No. 5243, irrespective of the number of gear stages used.

Setting bit 2 (TDR) of parameter No. 5201 to "1" enables setting of the extraction time constant for each gear set. Specify the extraction time constant for each gear in parameter Nos. 5271 to 5274.

If bit 1 (VGR) of parameter No. 5200 is set to "1", the gear ratio for the spindle and position coder can be set to anywhere between 1:32767 and 32767:1 in one-increment units for three gear sets with M series, four gear sets with T series, or two gear sets with 2nd spindle. 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 (DOV) of parameter No. 5200 to “1”, and setting an appropriate override value in parameter RGOVR of No. 5211. The time constant for withdrawal operations can be modified by bit 2 (TDR) of parameter No. 5201; when it is set to “1”, the values in parameter Nos. 5271 to 5274 are used as the time constant for withdrawal.

- **Acceleration and deceleration**

Linear acceleration/deceleration is valid for rigid tapping.

- **Override**

Override is invalid for rigid tapping. Fixed override can be applied to withdrawal operations by setting bit 4 (DOV) of parameters No. 5200 or RGOVR of No. 5211.

- **Dry run**

Dry run is valid for G84 (G74). When the dry run is applied to the tapping axis speed of G84 (G74), tapping is performed. The spindle speed will match the dry run speed.

- **Machine lock**

Machine lock is valid for G84 (G74).
When G84 (G74) is executed with the machine locked, however the tapping axis and the spindle do not move.

- **Reset**

When the reset operation is performed during rigid tapping, the mode is reset. The spindle motor goes to the ordinary mode, but G84 (G74) mode is not reset.

- **Feed hold, 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 6 (FHD) of parameter No. 5200 to “1”.
As with the machine lock signal, the feed hold and single block functions are also effective for the spindle indirectly, through tapping axis operations.

- **Operation mode**

G84 (G74) can be executed only in the MEM and MDI modes.

- **Manual feed**

Rigid tapping cannot be performed in the manual feed mode.

- **Backlash compensation**

In rigid tapping mode, the backlash is compensated for the lost motion at forward and reverse spindle rotations. Set it using parameter No. 5321 to No 5324. The backlash compensation is normally made for the tapping axis.

9.11.4 Display Data on the Diagnosis Screen

For rigid tapping adjustment, the diagnosis screen displays information related to rigid tapping.

For part of the display data, the user can choose between two sets of data items relating to the synchronization of the spindle and tapping axis by setting bit 0 (DGN) of parameter No. 5204.

- **Common display data**

The following information items are displayed, regardless of the setting of bit 0 (DGN) of parameter No. 5204:

- Spindle position deviation → Diagnosis No. 450
- Number of command pulses distributed to the spindle (momentary value) → Diagnosis No. 451
- Cumulative number of command pulses distributed to the spindle → Diagnosis No. 454

- **Display of rigid tapping synchronization error (When DGN = 0)**

When bit 0 (DGN) of parameter No. 5204 is set to 0, the following information items are displayed.

(Diagnosis Nos. 452 and 453 are not displayed.)

- Spindle–converted move command difference → Diagnosis No. 455
- Spindle–converted position deviation difference → Diagnosis No. 456
- Synchronization error range → Diagnosis No. 457

Spindle–converted move command difference

$$= \Sigma \frac{\text{spindle move command}}{\text{gear ratio}} - \Sigma \frac{(\text{tapping axis move command}) \times 4096}{\text{thread lead}}$$

Spindle–converted position deviation difference

$$= \frac{\text{spindle position deviation}}{\text{gear ratio}} - \frac{(\text{tapping axis position deviation}) \times 4096}{\text{thread lead}}$$

Synchronization error range

= (maximum spindle–converted move position deviation difference on the positive side)

– (maximum spindle–converted position deviation difference on the negative side)

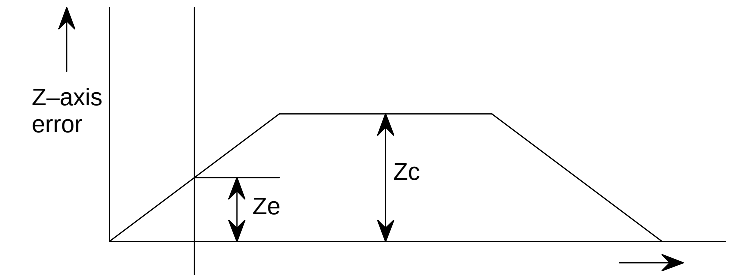
If a maximum allowable synchronization error range is set in parameter No. 5214, the position deviation alarm during spindle movement (alarm No. 741) is issued to indicate that the set synchronization error range has been exceeded. (If 0 is set in parameter No. 5214, no check is performed to detect whether the synchronization error range has been exceeded.)

- **Rigid tapping error display (error difference display) (When DGN = 1)**

When bit 0 (DGN) of parameter No. 5204 is set to 1, the following information items are displayed. (Diagnosis Nos. 455, 456, and 457 are not displayed.)

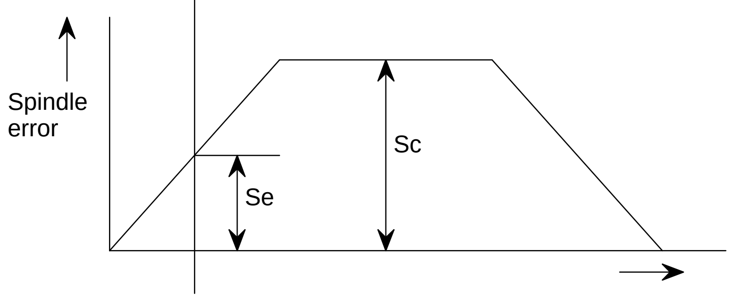
- Momentary error difference between the spindle and tapping axis → Diagnosis No. 452
- Maximum error difference between the spindle and tapping axis → Diagnosis No. 453

Diagnosis No. 0452 is cleared to “0” when rigid tapping mode is set or canceled, and diagnosis No. 0453 is cleared to “0” in the positioning of the rigid tapping cycle.
The following figure shows the tapping axis as the Z axis.



$$Zc = \frac{\text{Speed}}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \text{ (Theoretical value)}$$

Ze = Z–axis error counts (measured value)



$$Sc = \frac{\text{Speed} \times 360}{60} \times \frac{1}{\text{Gain}} \times \frac{1}{\text{Detection unit}} \times 10^2 \text{ (Theoretical value)}$$

Se = Spindle error counts (measured value)

Speed : mm/min or inch/min
Gain : 0.01 s^{-1}
Detection unit :mm, inch, or deg
Speed : 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 ΔS–ΔZ. This value is sampled at intervals of 64 ms.

Diagnosis screen

- **Spindle position deviation**

0450	SPINDLE MOTION ERROR

Spindle position deviation during rigid tapping

[Unit] Pulse

- **Number of pulses distributed to the spindle**

0451	SPINDLE MOTION PULSE

Number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Error difference between the spindle and tapping axis (momentary value)**

0452	RIGID ERROR

Momentary error difference between the spindle and tapping axis during rigid tapping (signed)

[Unit] %

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Error difference between the spindle and tapping axis (maximum)**

0453	RIGID ERROR(MAX)

Maximum error difference between the spindle and tapping axis during rigid tapping (absolute value)

[Unit] %

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 1.

- **Cumulative number of pulses distributed to the spindle during rigid tapping**

0454	SPINDLE PULSE(SUM)

Cumulative number of pulses distributed to the spindle during rigid tapping

[Unit] Pulse

- **Spindle–converted move command difference during rigid tapping (momentary value)**

0455	SYNC. PULSE(SUM)

Momentary spindle–converted move during command difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Spindle–converted position deviation difference during rigid tapping (momentary value)**

0456	SYNC. ERROR

Momentary spindle–converted position deviation difference between the spindle and the tapping axis during rigid tapping

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

- **Synchronization error range during rigid tapping (momentary value)**

0457	SYNC. WIDTH

Synchronization error range during rigid tapping (maximum value)

[Unit] Pulse

> **NOTE**
> This data item is displayed only when bit 0 (DGN) of parameter No. 5204 is set to 0.

9.11.5 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–62764EN) ”.

G□□ X(Z)__ C__ Z(X)__ R__ P__ F__ K__ (M□□) ;

- (M□□) — M code for C–axis clamping
- K — Number of repeats
- F — Cutting feedrate
- P — Dwell time at the bottom face of a tapped hole and upon R point return
- R — Specifies the position of an R point
- Z(X) — Specifies the position of the bottom face of a tapped hole
- X(Z), C — Specifies a tapping position.
- G□□ — G84 — Front tapping cycle; G88 — Side tapping cycle

The rigid tapping mode can be specified by using any of three methods:

- Specification of M29S**** before specifying a tapping cycle
- Specification of M29S****in the same block
- Enabling rigid tapping to be performed without specifying M29S****

When using the third method, specify S**** either before or in a block containing G84 (G88).

Thus, the spindle stops, after which the tapping cycle specified next is placed in rigid tapping mode.

Rigid tapping mode 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 0 (G84) of parameter No. 5200 to 1)

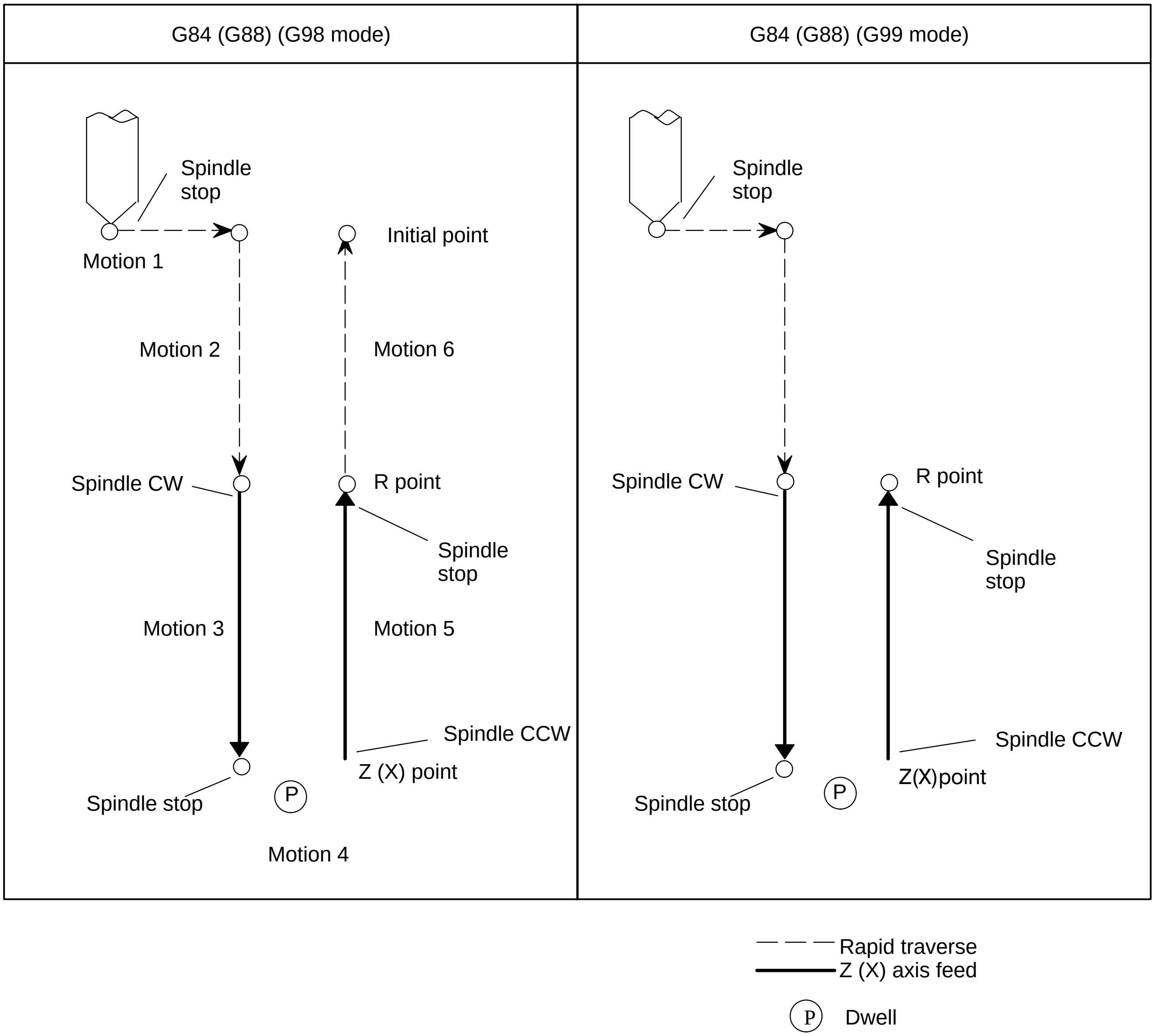
```
G□□X  (Z)  __C__Z (X) __R_P_F_K_S**** (M□□);  ┐
X  (Z)  __C__;                                │ Rigid
X  (Z)  __C__;                                │ tapping
 .                                            │ mode
 .                                            ┘
G80;
```

Notes on the T series

-

NOTE
1. In feed per minute mode, F_/S**** determines a thread lead. In feed per rotation mode, F_ specifies a thread lead.
2. S**** must specify a value that does not exceed the value set in the maximum spindle speed parameter (No. 5241 to 5244) for the gear to be used. Otherwise, P/S alarm No. 200 is issued in a block containing G84 (G88).
3. F_ must specify a value that does not exceed the maximum cutting feedrate. When 0 is specified, P/S alarm No. 201 is issued.
4. Between M29 and G84 (G88), S and a command for movement along an axis must not be specified. Further, M29 must not be specified in a tapping cycle. Otherwise, P/S alarm Nos. 203 and 204 are issued, respectively.

G84·G85 (Tapping cycle)



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 (DOV) of parameter No. 5200 and parameter No. 5211 (RGOVR).

NOTE

G code system A does not include G98 (return to initial level) and G99 (return to R point level). Return to the initial level is always used.

Rigid tapping in feed per rotation mode

Rigid tapping is classified into two types: rigid tapping in feed per rotation mode (G99) and rigid tapping in feed per minute mode (G98).

Example)

The example below specifies rigid tapping in feed per rotation mode for cutting a thread with a lead of 1 mm at a spindle speed of 1,000 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 RGTAP<G061#0>

[Classification] Input signal

[Function] When M29 (miscellaneous function for preparation for rigid tapping) is specified, the PMC enters rigid tapping mode, then turns on this signal to notify the CNC.

1 : The PMC enters in rigid tapping mode.
0 : The PMC does not enter rigid tapping mode.
For an explanation of placing the PMC in rigid tapping mode, see the description of the interface with the PMC, given later.

This signal posts whether the PMC has entered rigid tapping mode. If this signal is not set to 1, even when M29 is specified, a P/S alarm is issued in a G84 (G74) block.

Spindle rotation direction signals RGSPM, RGSPP <F065#1, #0> (M series only)

[Classification] Output signal

[Function] During rigid tapping, these signals notify the PMC of whether the spindle is rotating in the forward or reverse direction.

During rigid tapping, the spindle is:

RGSPP 1 : Rotating in the forward direction (CW).
0 : Not rotating in the forward direction.

RGSPM 1 : Rotating in the reverse direction (CCW).
0 : Not rotating in the reverse direction.

[Output condition] These signals are output when the spindle is rotating in rigid tapping mode. This means that, even in rigid tapping mode, these signals are not output, for example, when the spindle is being positioned to a hole position, or a dwell operation is in progress at the bottom of a hole or at an R point.

These signals are not output in the feed hold state or single block stop state. When the spindle is placed in the interlock stop state, machine lock state, or Z–axis ignore state, however, the spindle is not regarded as having stopped; these signals are output.

These signals are valid only in rigid tapping mode. In normal spindle control, these signals are not output; both RGSPP and RGSPM are set to “0”.

Rigid tapping in–progress signal RTAP<F076#3>

[Classification] Output signal

[Function] This signal notifies the PMC that rigid tapping mode is set.

RTAP 1 : Rigid tapping mode is currently set.

0 : Rigid tapping mode is not currently set.

By latching M29, the PMC knows that rigid tapping mode has been specified, and thus performs the required processing on the PMC side. This signal can substitute for the latching of M29. Even in this case, however, FIN for M29 cannot be omitted.

9.11.6.2 Signals related to S code output

Spindle enable signal ENB<F001#4>

Second spindle enable signal ENB2<F001#4>

[Classification] Output signal

[Function] These signals post whether the spindle output is 0. In rigid tapping mode, these signals are used to cancel rigid tapping in a PMC sequence associated with rigid tapping.

For details, see the explanation of the interface with the PMC, given later.

Spindle–speed function code signals (binary output) S00 to S31 <F022 to F025>

Spindle–speed function strobe signal SF<F007#2>

[Classification] Output signal

[Function] These signals send S codes specified for the CNC, in binary format, to the PMC.

[Output condition] When an S code is specified, the specified value is output, in binary format, with 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 6 (SFA) of parameter No. 3705.

[2] T–type gear selection method

SF output depends on the setting of bit 5 (NSF) of parameter No. 3705.

T series: The following parameter needs to be set to output S codes and SF: Bit 4 (EVS) of parameter No. 3705 = 1

In rigid tapping, when SF is to be used by the PMC to read an S code output signal for gear switching or output switching, set the above parameters as required.

NOTE

1 The timing charts, given later, give examples of gear switching by setting the parameters as follows:
M series: SFA = 0, NSF = 0
T series: EVS = 1

2 When the constant surface speed control function is being used, an S code (specifying a surface speed) used for constant surface control (G96) is output. Such an S code can be distinguished from an S code used for specifying a rotation speed. One method is to use, for example, the constant surface speed control in–progress signal (F002#2) for the processing performed on the PMC side. Another method is to mask the S code and SF signal, output by setting bit 0 (ESF) of parameter No. 3705.

9.11.6.3 Signals related to gear switching

Gear selection signals (output) GR3O, GR2O, GR1O <F034#2, #1, #0> (M series only)

[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	×	×	○
2nd (medium) speed gear	×	○	×
3rd (high) speed gear	○	×	×

Gear selection signals (input) GR2, GR1<G028#2, #1>

[Classification] Input signal

[Operation] When T–type gear selection is being used, these signals are used in a PMC sequence for rigid tapping.

The signals post, to the CNC, information about a spindle gear to be used.

Reference information: The table below shows the relationship between the output signals and spindle gear selection.

	GR2	GR1
1st (low) speed gear	×	×
2nd (medium) speed gear	×	○
3rd (high) speed gear	○	×
4th (high) speed gear	○	○

← In M series rigid tapping, the specification of the 4th (high) speed gear is invalid. If specified, the system assumes that the 3rd (high) speed gear has been specified.

Gear selection signal (input) GR21<G029#0>

[Classification] Input signal

[Operation] When rigid tapping with the second spindle is being performed, the signal is used in a PMC sequence.

The signal notifies the CNC of spindle gear information when the second 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 spindle gear.
0 : The first stage is currently selected as the second spindle gear.

	GR21
1st speed gear	×
2nd speed gear	○

When a serial spindle is used, the serial spindle clutch/gear selection signals (G070#3, #2 for the first spindle, and G074#3, #2 for the second spindle) must be set in addition to the setting of the gear selection signal described above.

9.11.6.4 Signals related to second spindle rigid tapping

Gear selection signal (input) GR21<G029#0>

See the description of the signals related to gear switching, given above.

Signals related to multi–spindle control

Spindle selection signals SWS1, SWS2 <G027#0, #1>

Rigid tapping spindle selection signals RGTSP2, RGTSP1 <G061#5, #4> (T series only)

[Classification] Input signal

[Operation] SWS1 and SWS2 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. (The signals can be used for this purpose when bit 7 (SRS) of parameter No. 5200 is set to 0.)

RGTSP2 and RGTSP1 are used to select a spindle used for rigid tapping, independently of the SWS1 and SWS2 signals, when the multi–spindle control option is being used. (The RGTSP2 and RGTSP1 signals can be used when bit 7 (SRS) of parameter No. 5200 is set to 1. These signals are supported only by the T series.)

See the tables below for details of the settings of these signals.

When bit 7 (SRS) of parameter No. 5200 is set to 0, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	SWS1	SWS2
First spindle	“1”	“1”
First spindle	“1”	“0”
Second spindle	“0”	“1”
P/S alarm No. 205 is issued.	“0”	“0”

When bit 7 (SRS) of parameter No. 5200 is set to 1, to select a spindle to be used for rigid tapping, set the signals as indicated below.

Spindle used for rigid tapping	Signal state	
	RGTSP1	RGTSP2
First spindle	“1”	“1”
First spindle	“1”	“0”
Second spindle	“0”	“1”
P/S alarm No. 205 is issued.	“0”	“0”

WARNING
These signals must be applied before the command for rigid tapping (M29 S....; G84 X...) is specified. The states of these signals must not be changed before rigid tapping has been completed.

Spindle–by–spindle stop signals *SSTP1, *SSTP2 <G027#3, #4>

[Classification] Input signal

[Operation] These signals are used to stop each spindle when the multi–spindle control option is used. In a PMC sequence for rigid tapping, the ENB and ENB2 signals are used. Accordingly, the logic of the signals used for a spindle selected to perform rigid tapping must match the logic of the spindle stop signal *SSTP.

***SSTP1** 1 : The output to the first spindle 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.

Second position coder selection signal PC2SLC<G028#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 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
G027				*SSTP2	*SSTP1		SWS2	SWS1
G028	PC2SLC					GR2	GR1	
G029								GR21
G061			RGTSP2	RGTSP1				RGTAP

	#7	#6	#5	#4	#3	#2	#1	#0
F001				ENB				
F007						SF		
F034						GR3O	GR2O	GR1O
F038						ENB2		
F065							RGSPM	RGSPP
F076					RTAP			

9.11.6.6 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)

The PMC must manage rigid tapping mode as follows: rigid tapping mode is set using M29, and is canceled upon the issue of a reset or at the falling edge of the spindle enable signal ENB in rigid tapping mode. ENB is used during rigid tapping in this way, so the spindle stop signal *SSTP must not be set to “0”.

However, *SSTP and SOR may be used for gear switching. To do so, ensure that the PMC does not cancel rigid tapping mode on a falling edge of ENB while *SSTP is “0”. Rigid tapping mode may be set on a rising edge of the RTAP signal instead of by using M29, and canceled on a falling edge of the RTAP signal instead of the ENB signal.

In rigid tapping using the second spindle, the ENB2 signal must be used for rigid tapping mode management.

Controlling spindle output by the PMC

When the SIND signal is set to “1”, spindle output is controlled by the signals (SSIN, SGN, R1I to R12I) output from the PMC.

At this time, the effect of ENB is as described above. In addition, when rigid tapping mode is canceled in a block containing G80;, the momentary rotation of the spindle, caused by a delay in the PMC processing, can result. Accordingly, the PMC’s control over spindle output must be disabled in rigid tapping mode by setting SIND to “0”.

For the same reason, the PMC’s control over second spindle output must be disabled in rigid tapping mode by setting SIND2 to “0”.

T–type gear selection method

When T–type gear selection is used, the PMC must determine whether gear switching is to be performed, and subsequently perform gear switching as required. For this purpose, each time a spindle–speed function code is specified, the spindle–speed function code read signal (SF) and spindle–speed function code signals (S00 to S31) must be output to the PMC. The required parameter settings are described below.

· M series: Set bit 5 (NSF) of parameter No. 3705 to 0 to output SF.

· T series: Set bit 4 (EVS) of parameter No. 3705 to 1 to output SF.

Gear switching timing

In general, a block containing M29 (miscellaneous function for preparation for rigid tapping) specifies S****, S**** being output when a block containing G84 (G74) is executed. This means that gear switching is performed in the block specifying G84 (G74).

When rigid tapping mode is specified

M29 (miscellaneous function for preparation for rigid tapping) and S**** specify rigid tapping mode. When M29 is accepted by the PMC, the following processing must be performed:

- Stop the spindle when it is rotating.
- Check that the spindle has stopped completely, then set the rigid tapping signal RGTAP <G061#0> to on.
- Activate the spindle motor. Activate the motor so that a positive speed command rotates the spindle in the forward direction (CCW when viewed from the – side of the tapping axis).
- Return FIN at least 250 ms after activation.

NOTE

The condition “at least 250 ms after activation” results from there being no way of checking the completion of spindle motor activation. Therefore, this wait period serves as an alternative. The time required for activation to be completed varies with the spindle motor and amplifier. Therefore, this value of 250 ms is given as a guideline only.

In an M29 block, S**** is not executed, merely being read in. S**** is executed in a G84 block. Spindle output is equivalent to the specification of S0.

The timing chart is shown in the chart indicating the execution of G84 (G74).

Execution of G84 (G74)

When M29S****; is specified, S**** is read in, spindle output being equivalent to the specification of S0;. S**** is output when G84 (G74) is executed. Thus, the processing described below is performed.

● When M–type gear selection is used

When using a machine that features multiple gear stages for use with the spindle motor and spindle, and S**** is outside the previously selected gear range, the spindle–speed function strobe signal SF <F007#2> and gear selection signals (output) GR3O, GR2O, GR1O <F034#2, #1, #0> are output to the PMC.

At this time, perform gear switching at the PMC.

- **When T–type gear selection is used**

The spindle–speed function strobe signal SF <F007#2> and spindle–speed function code signals S00 to S31 <F022 to F025> are output to the PMC. (However, parameter setting is required to enable output of the S codes and SF signal. See the description of each bit of parameter No. 3705.)

At this time, the PMC must determine whether gear switching is to be performed, and perform gear switching as required. The selected gear must be reflected in the gear selection signals (input) GR2 and GR1 <G028#2, #1> for notification to the CNC.

From GR2 and GR1, the CNC determines which gear is selected.

However, note the difference between the M series and T series, as described below.

M series: Up to three gear stages are supported for the first spindle, and up to two gear stages for the second spindle. (The gear selection for the second spindle is notified to the CNC by the signal GR21 <G029#0>)

T series: Up to four gear stages are supported for the first spindle, and up to two gear stages for the second spindle. (The gear selection for the second spindle is notified to the CNC by the signal GR21 <G029#0>.)

An S code is output in the first block (positioning to tapping position) of G84 (G74) execution. However, the spindle motor position loop is closed in the next block (R point positioning). Accordingly, spindle speed offset must be adjusted accurately until the position loop has been closed in the second block of G84 (G74) execution after the PMC activates the spindle motor with M29. Otherwise, the spindle motor may rotate slightly. (This applies only to an analog spindle. No offset adjustment is required for a serial spindle.)

Rigid tapping mode may be specified by specifying M29 before G84, specifying M29 and G84 in the same block, or by specifying G84 as a G code for rigid tapping. In each case, PMC processing is the same. (The M29 code is always output.)

9.11.7 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:	T–type	only
M series:	M–type	when constant surface speed control is not being used and bit 4 (GTT) of parameter No. 3706 is set to 0
	T–type	when constant surface speed control is being used, or bit 4 (GTT) of parameter No. 3706 is set to 1

9.11.7.1 When M29 is specified before G84 (G74) M type gear selection method

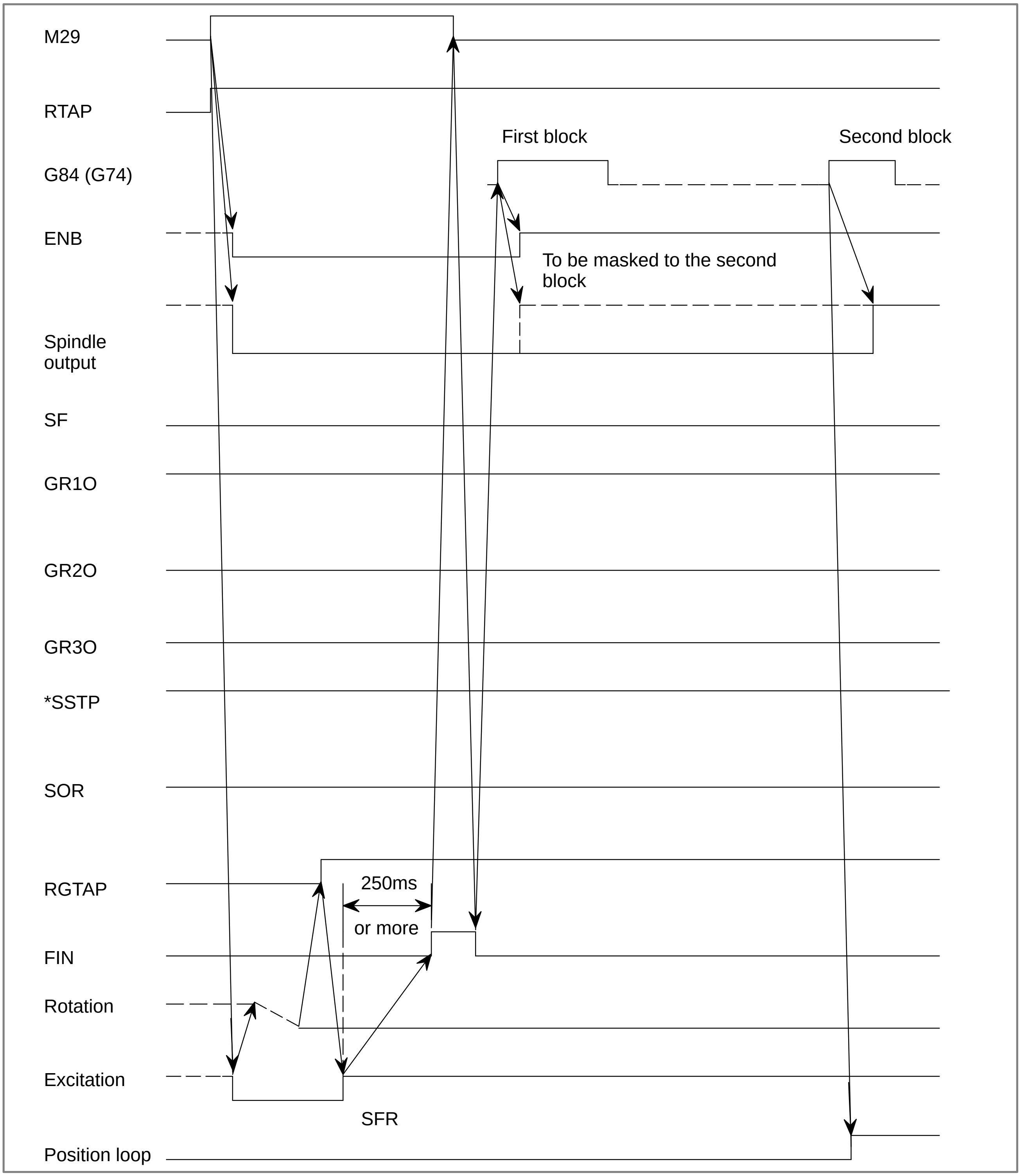
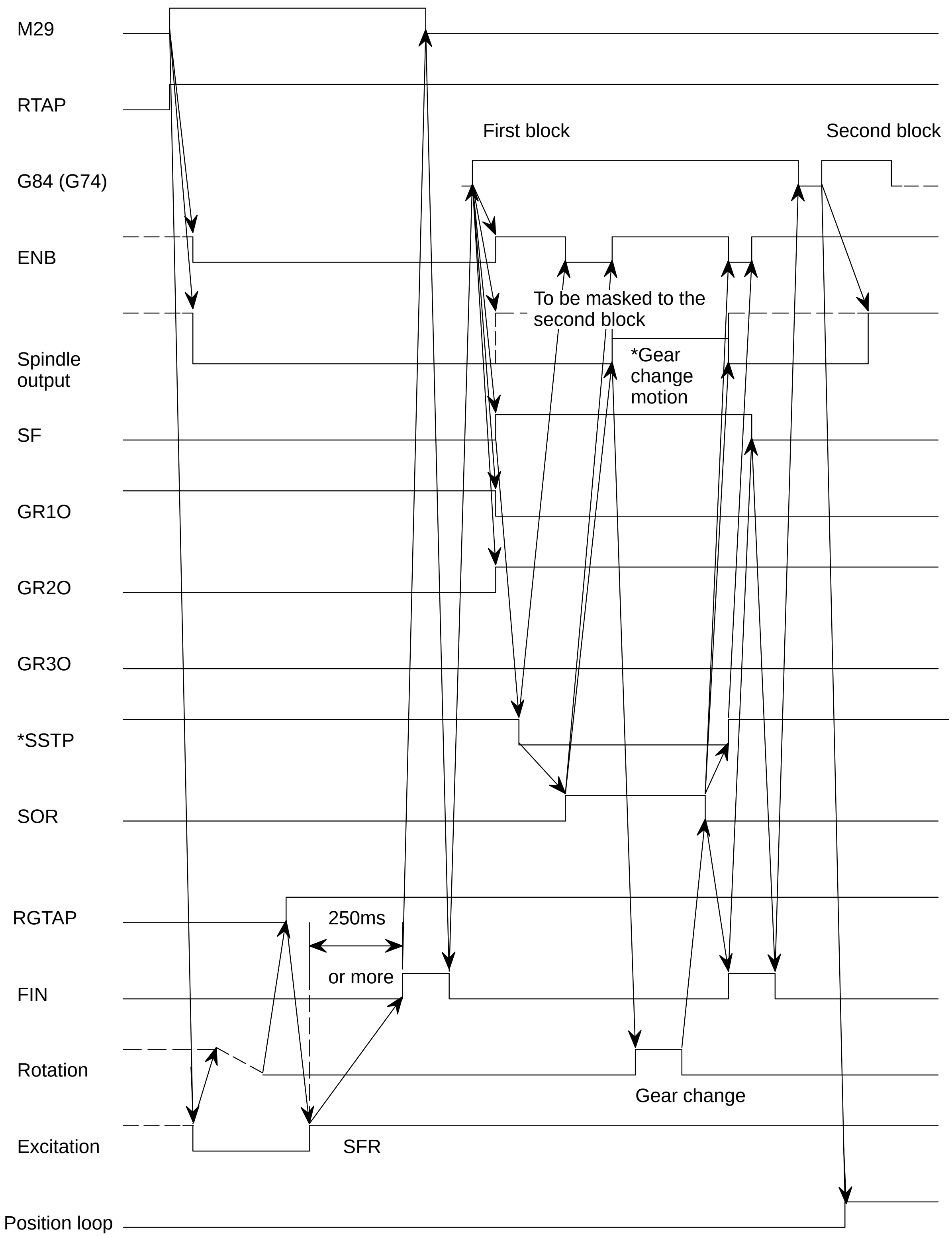


Fig. 9.11.7.1 (a) Gear is not changed



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

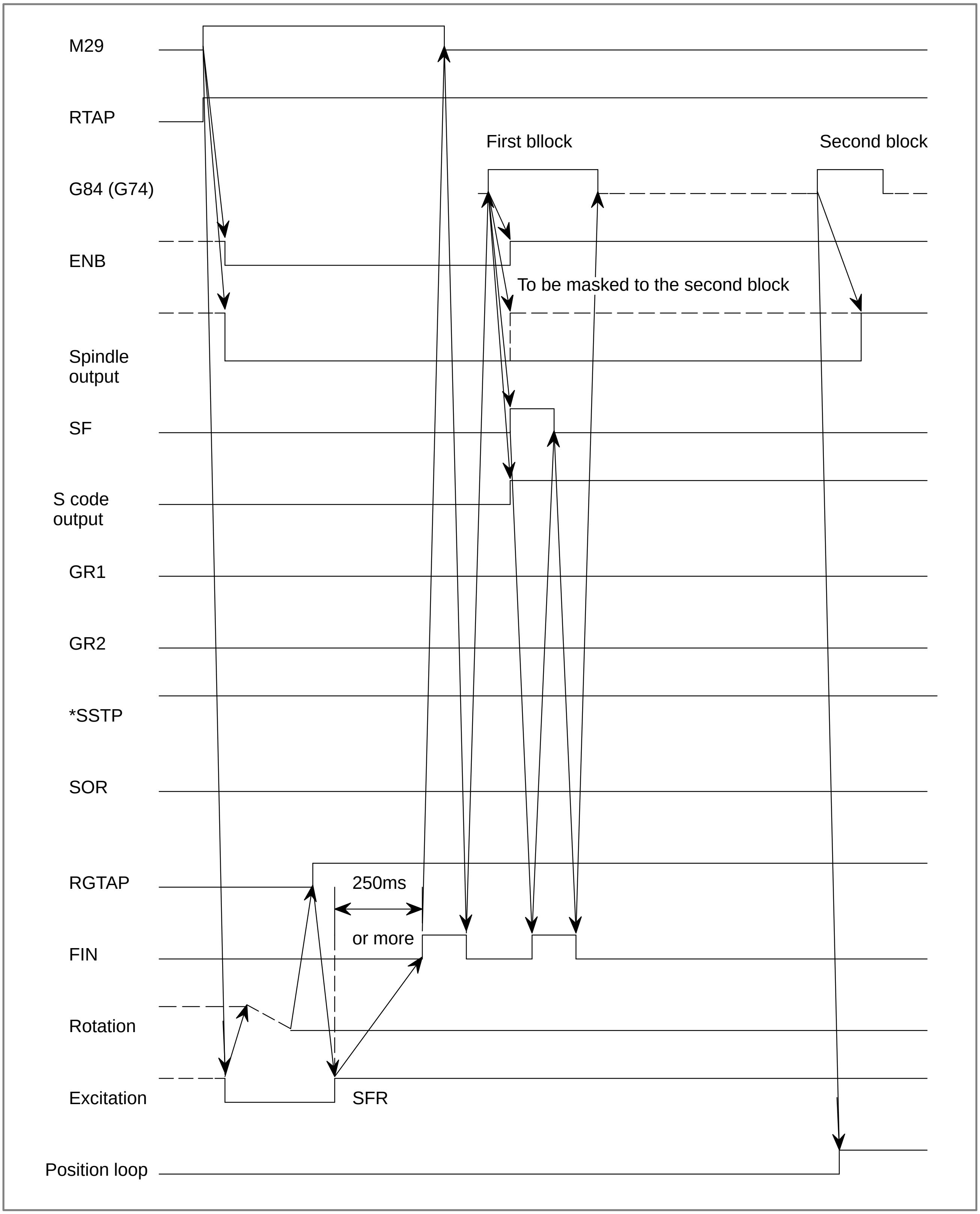
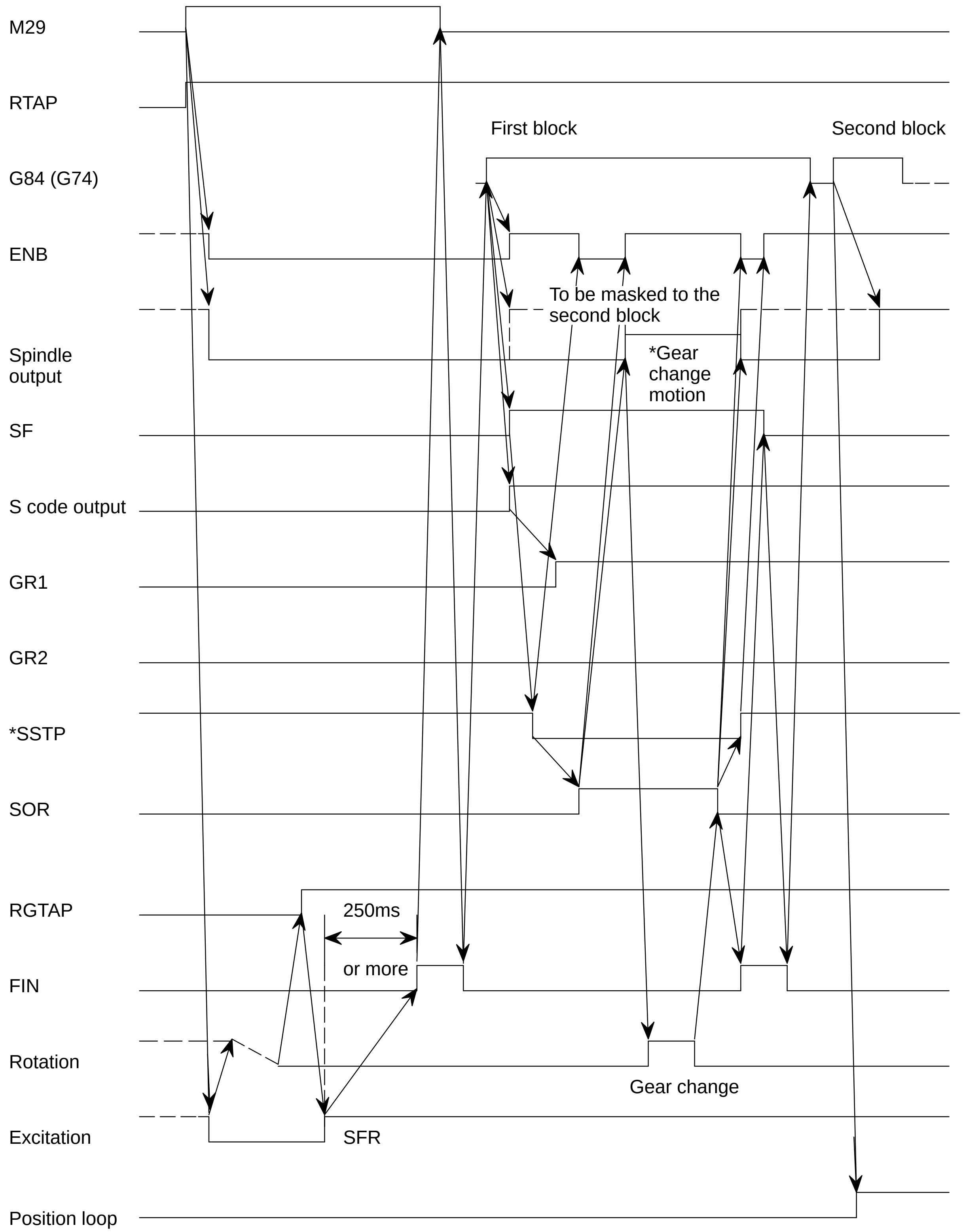


Fig. 9.11.7.1 (c) Gear change is not performed



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

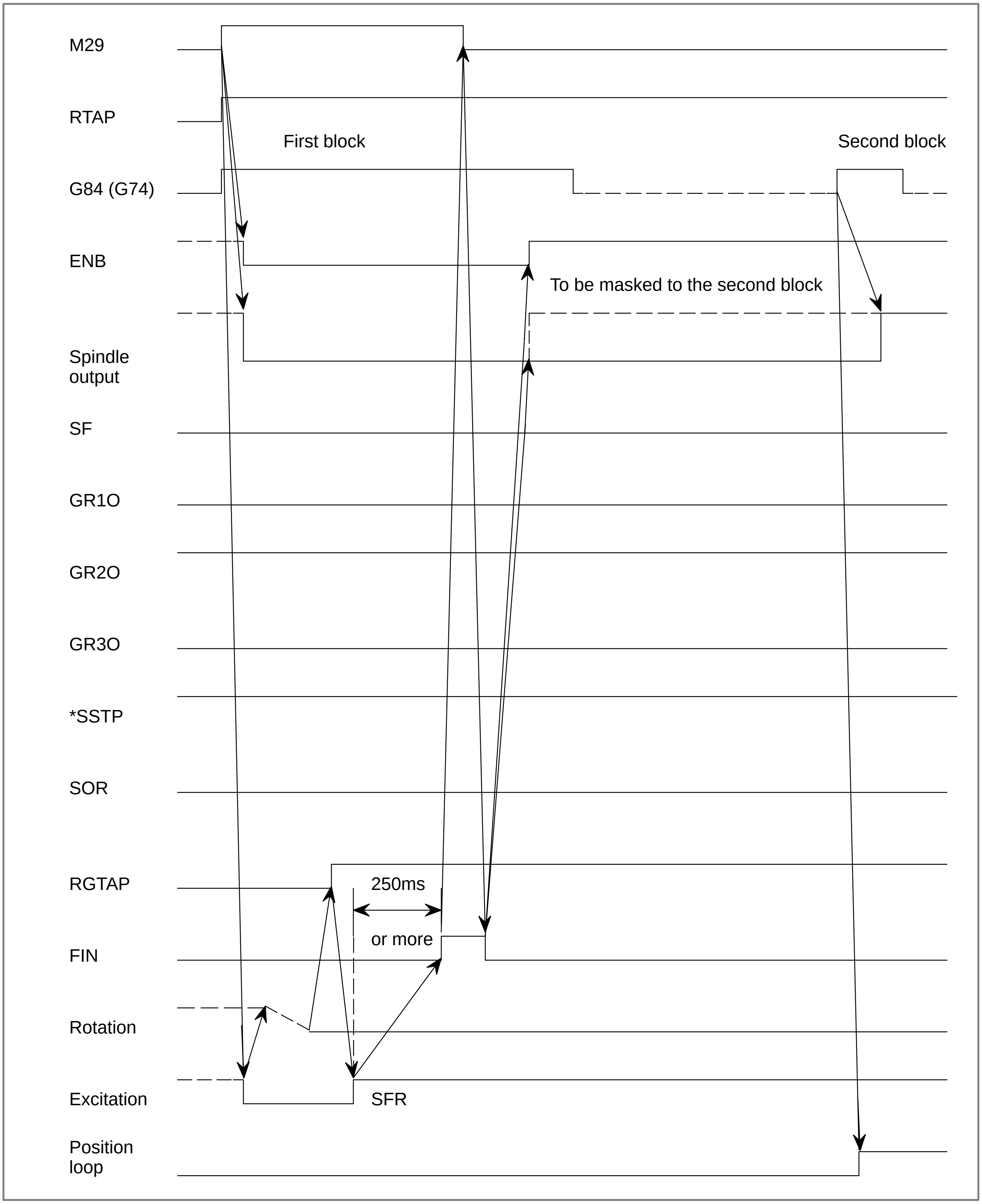
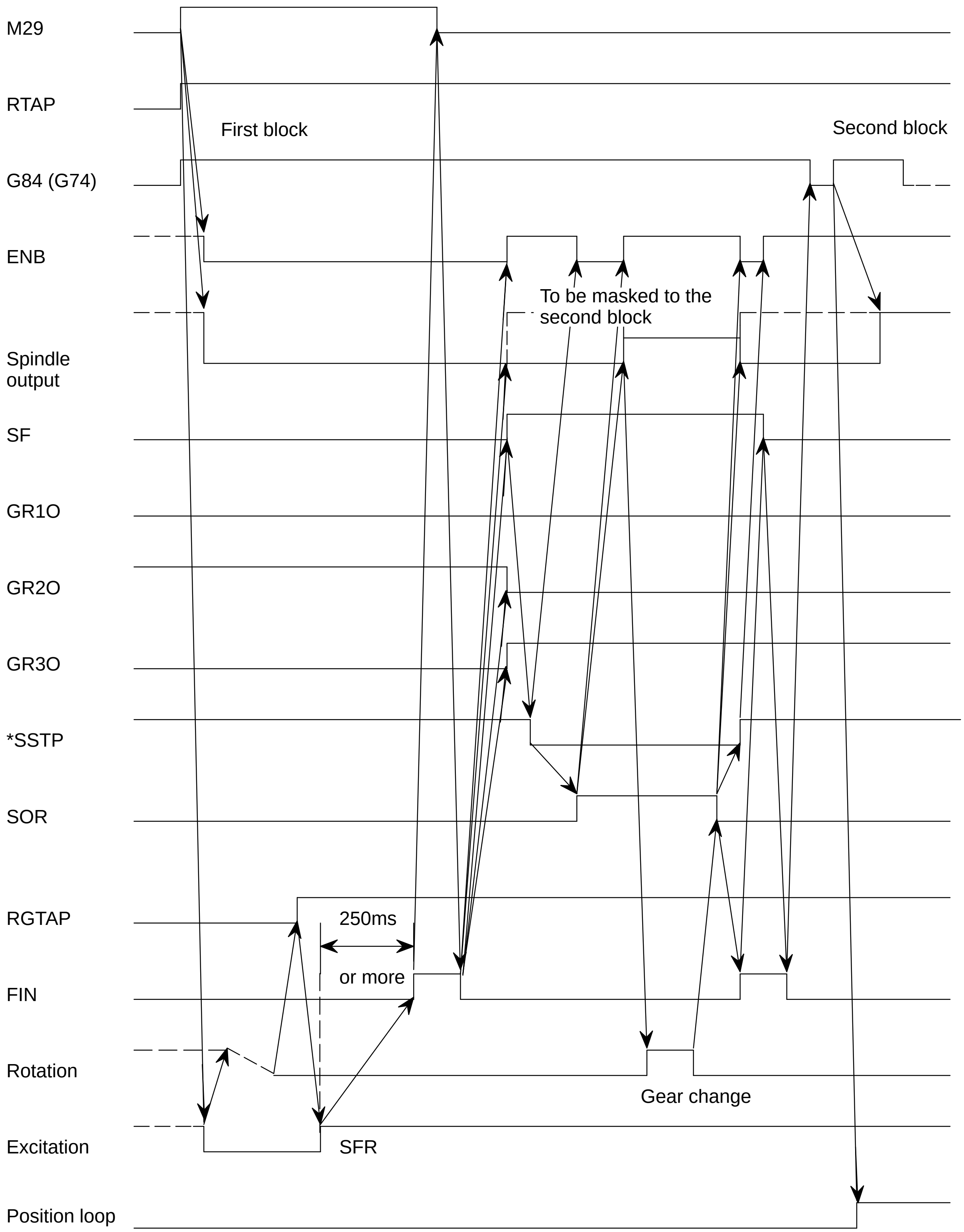


Fig. 9.11.7.2 (a) When gear–change is not performed



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

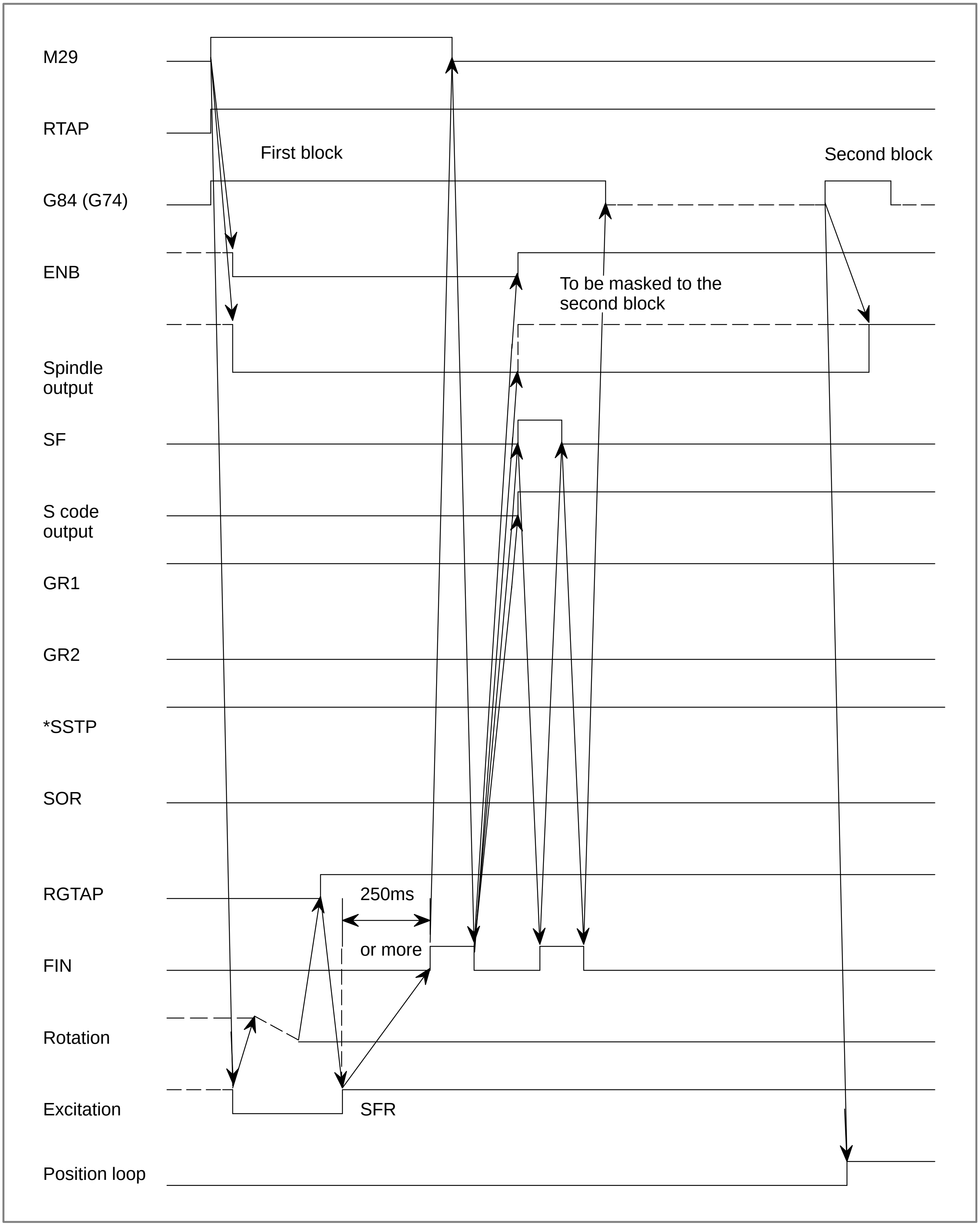
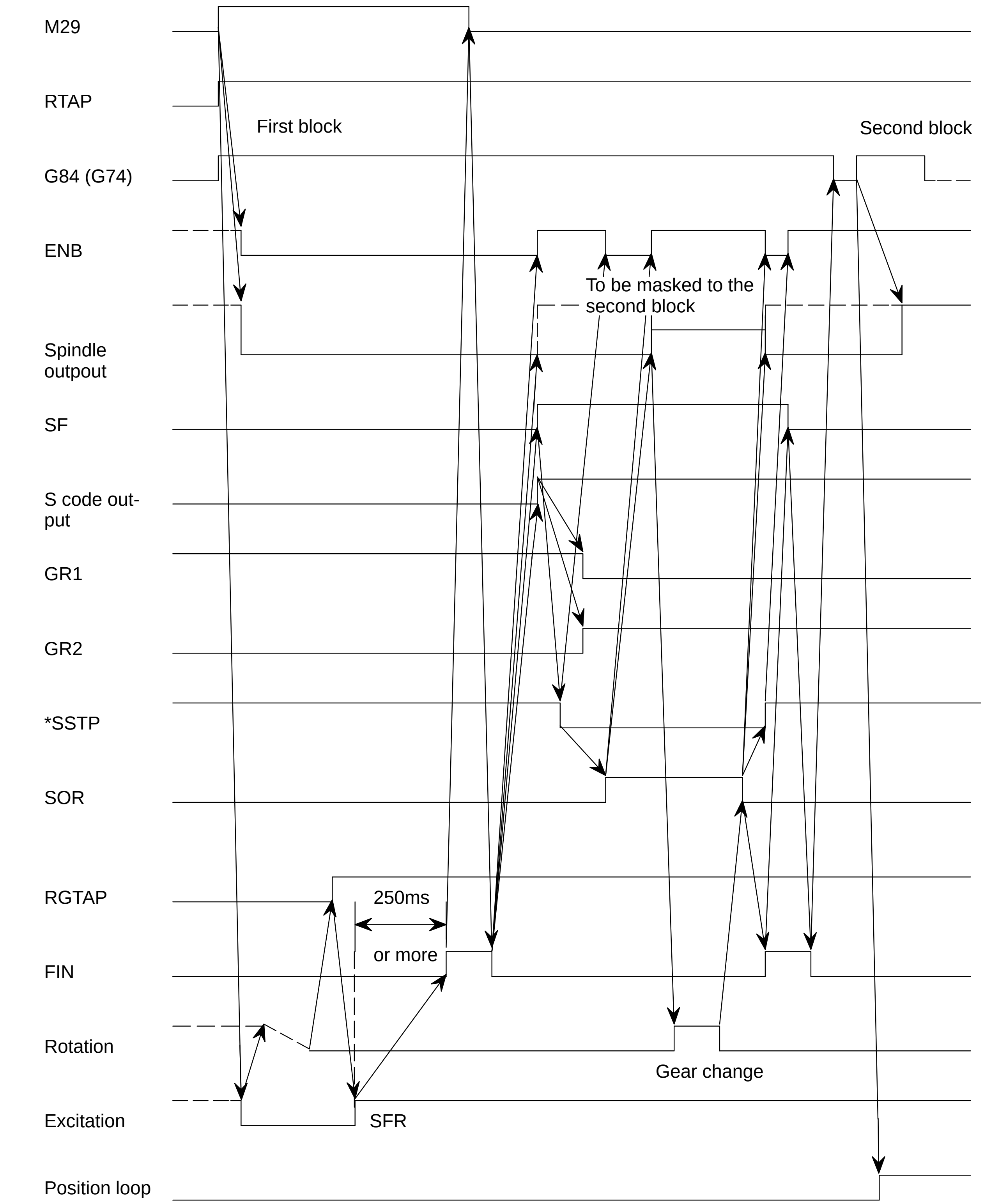


Fig. 9.11.7.2 (c) When gear change is not performed



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

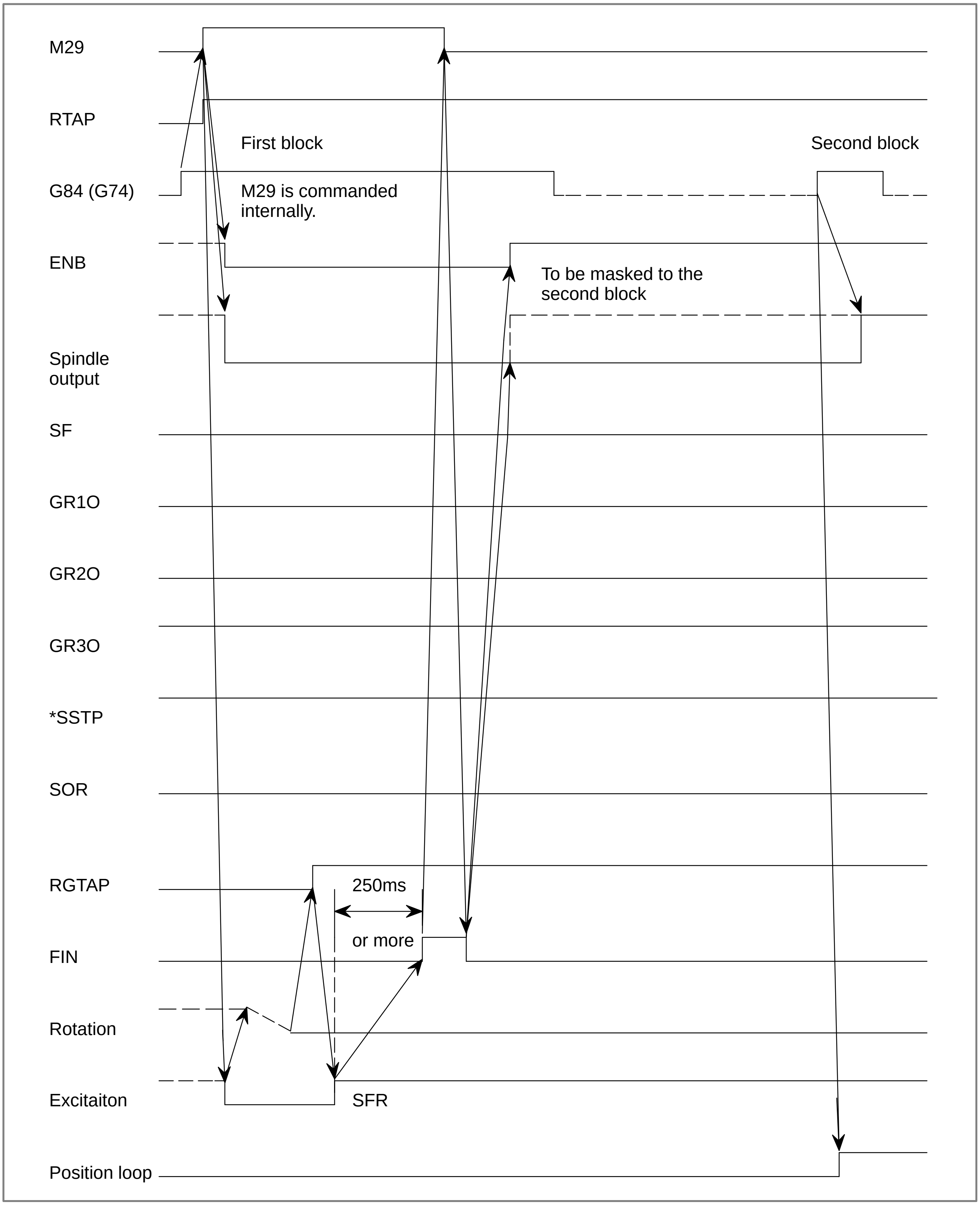
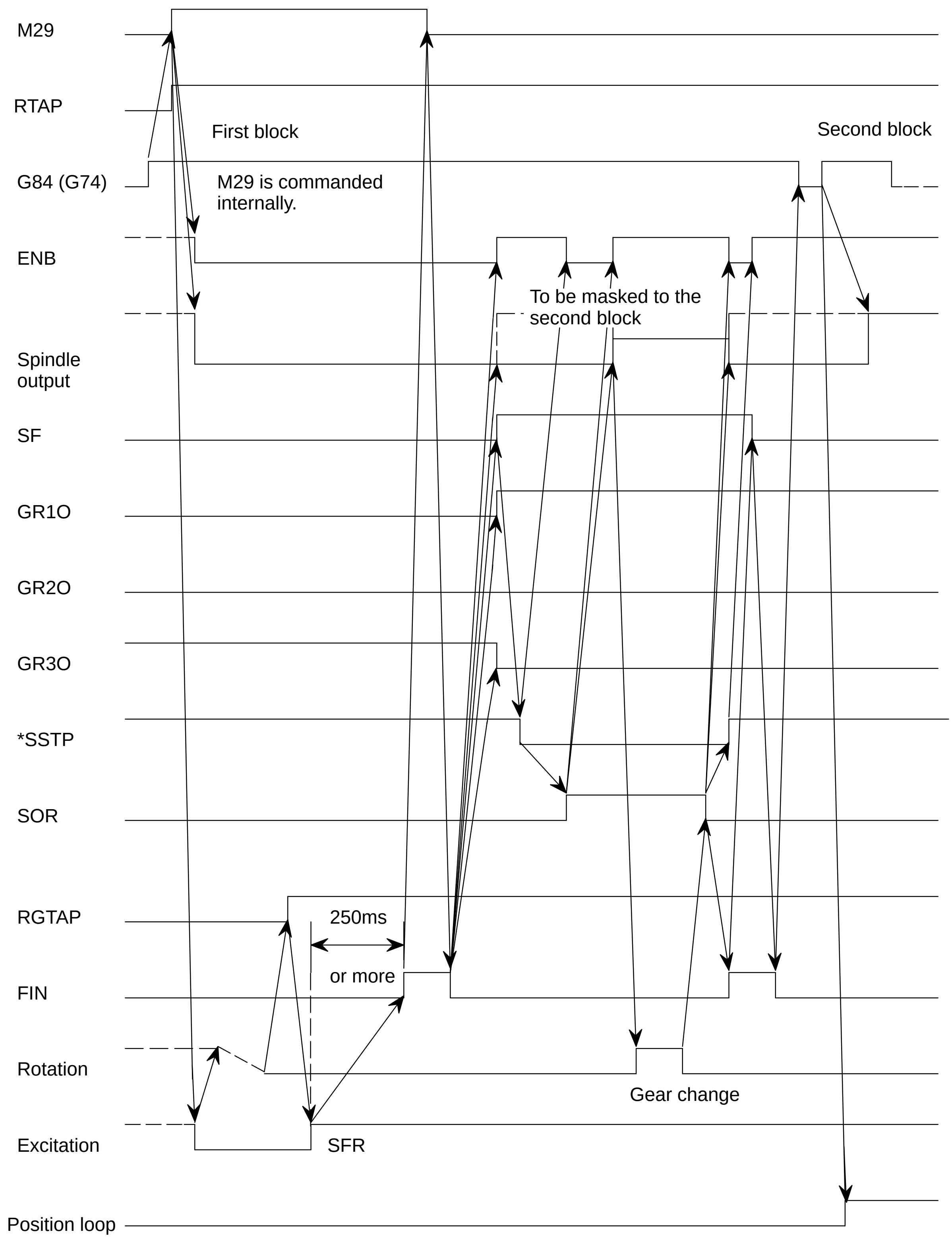


Fig. 9.11.7.3 (a) When gear–change is not performed



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

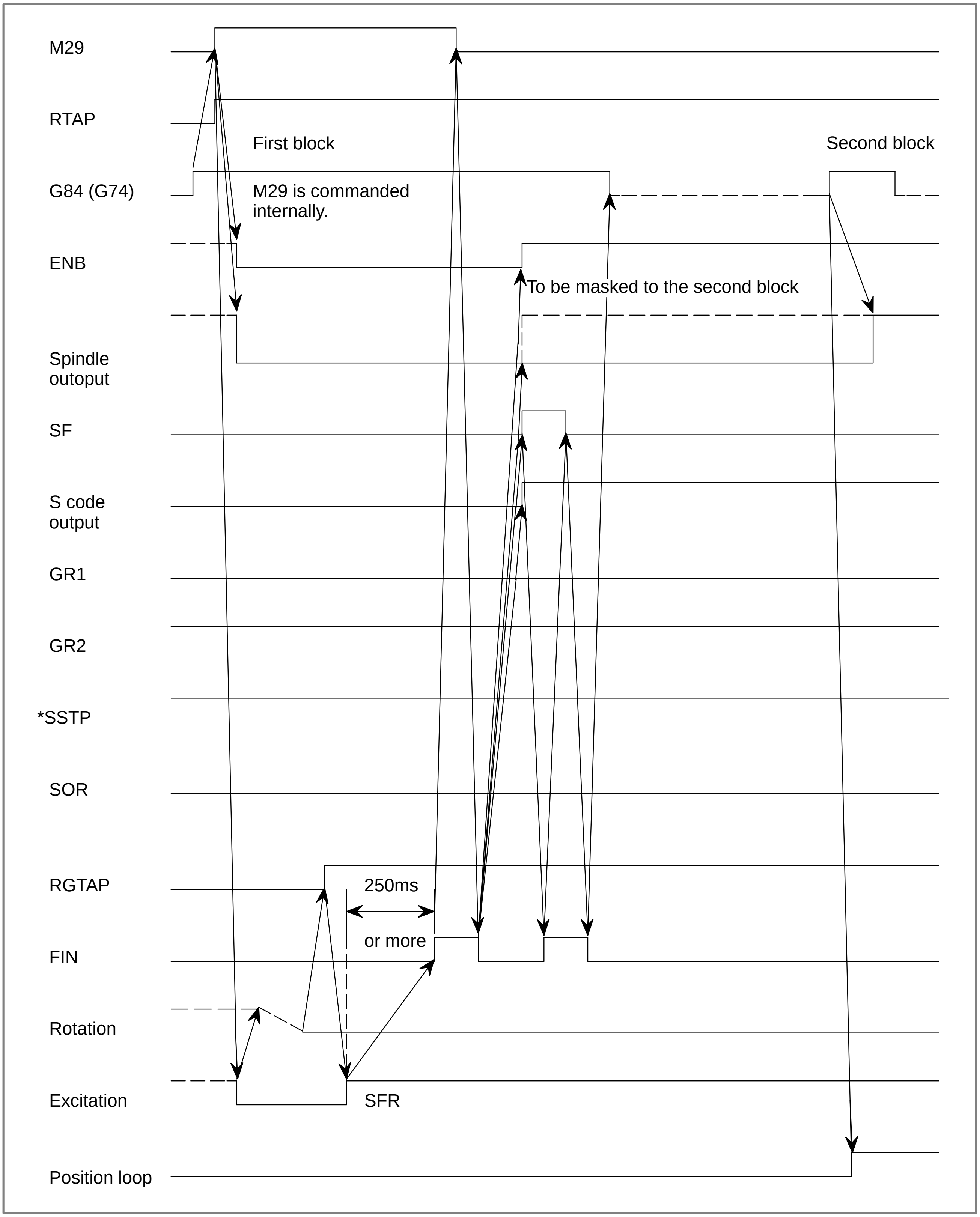
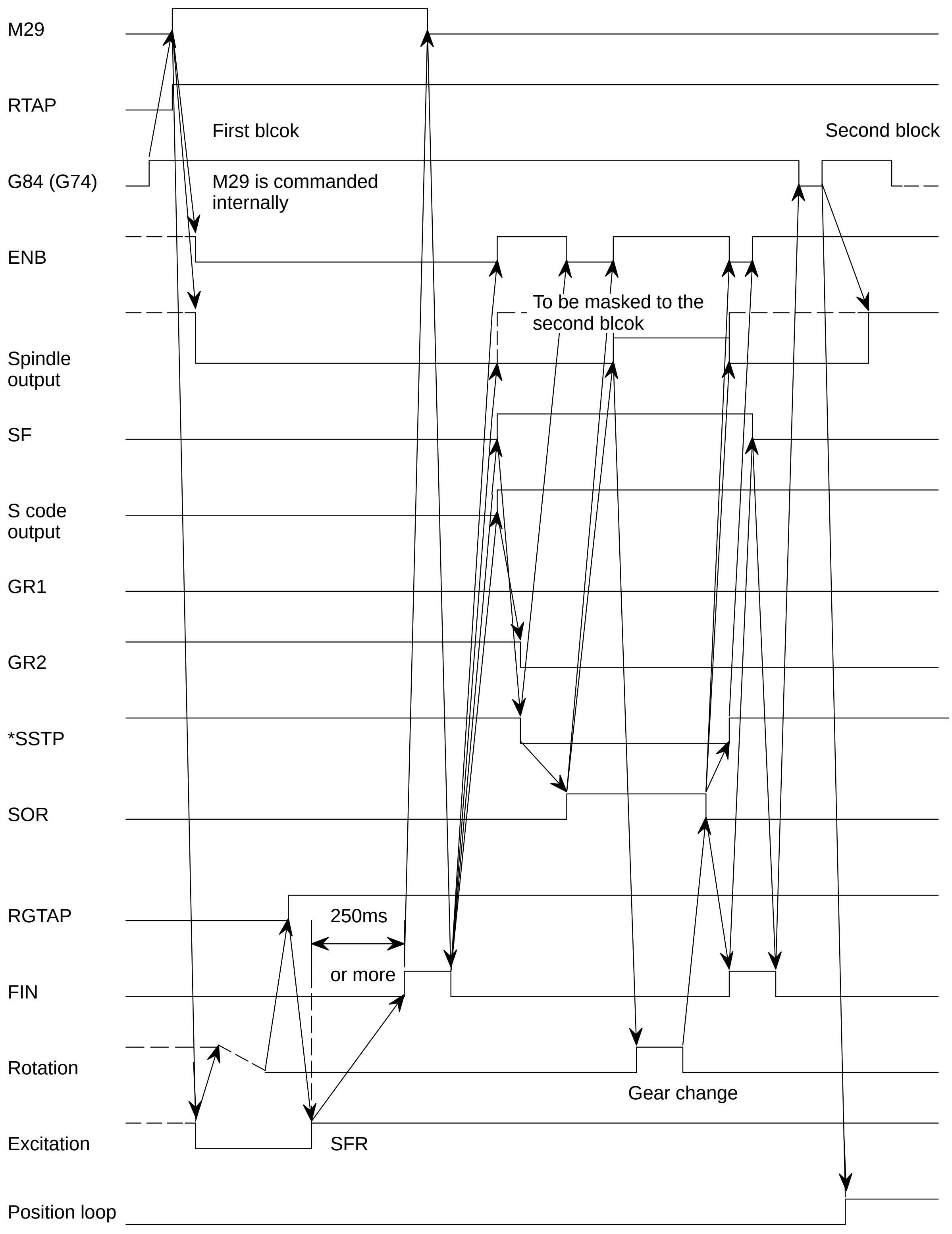


Fig. 9.11.7.3 (c) When gear change is not performed



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.

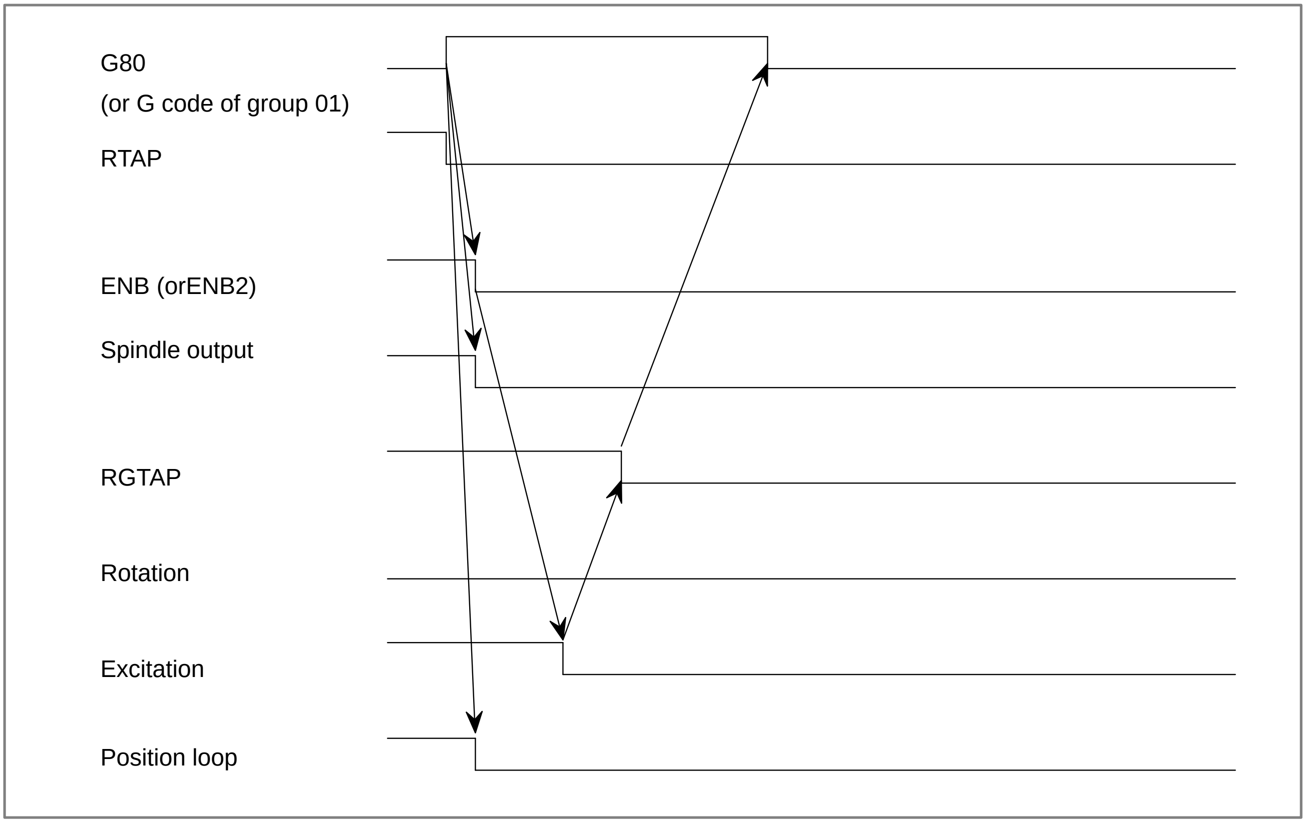
Fig. 9.11.7.3 (d) When gear–change is performed (high to low gear)

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 signal for 2nd 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 CRG (parameter No. 5200#2) is “1”, the system goes directly to the next block without checking that the rigid tapping signal is “0”. Set CRG to “1” for systems in which the rigid tapping signal is always “1”.



WARNING

1 If rigid tapping mode is canceled by a Group 01 G code, such as G00 or G01, the block containing the G code is executed at the same time the ENB signal is turned to “0”. Therefore, if a block contains an M code for controlling the spindle, an error may occur during processing in the PMC.

2 When CRG (Parameter No. 5200#2) is 1, if the next block contains an M code for controlling the spindle, an error may occur during processing in the PMC, when:

· Rigid tapping mode is canceled by issuing G80

· Rigid tapping mode is canceled by issuing a Group 01 G code , such as G00 or G01

NOTE

Rigid tapping mode is canceled as described above regardless of the gear selection method of M–type or T–type.

9.11.8 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF		SGT			ESF

[Data type] Bit

ESF When the spindle control function (S analog outpu or S serial output) is used, and the consatant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:

0 : S codes and SF are output for all S commands.

1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S–––;).

NOTE

For the T system, this parametar is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1.

For the M series, SF is not output:

- For an S command used to specify maximum spindle speed clamping (G92S–––;) in constant surface speed control mode
- When bit 5 (NSF) of parameter No. 3705 is set to 1

SGT Gear switching method during tapping cycle (G84, G74)

0 : Method A (Same as normal gear switching method)

1 : Method B (Gears are switched during tapping cycle according to the spindle speed set in parameters 3761 and 3762)

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.

1 : Output for an S command.

NOTE

The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S–––;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface–speed control,

0 : SF is output.

1 : SF is not output:

SFA: The SF signal is output:

0 : When gears are switched

1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PG2	PG1
				GTT			PG2	PG1

[Data type] Bit

PG2, PG1 Gear ratio of spindle to position coder

Magnification	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

$$\text{Magnification} = \frac{\text{Number of spindle revolutions}}{\text{Number of position coder revolutions}}$$

GTT Selection of a spindle gear selection method

0: Type M
1 : Type T

NOTE

1 Type M:
The gear selection signal is not entered. In response to an S command, the CNC selects a gear according to the speed range of each gear specified beforehand in a parameter. Then the CNC reports the selection of a gear by outputting the gear selection signal. The spindle speed corresponding to the gear selected by the gear selection signal is output.
Type T:
The gear selection signal is entered. The spindle speed corresponding to the gear selected by this signal is output.

2 When the constant surface speed control option is selected, type T is selected, regardless of whether this parameter is specified.

3761	
	Spindle speed when switching from gear 1 to gear 2 during tapping

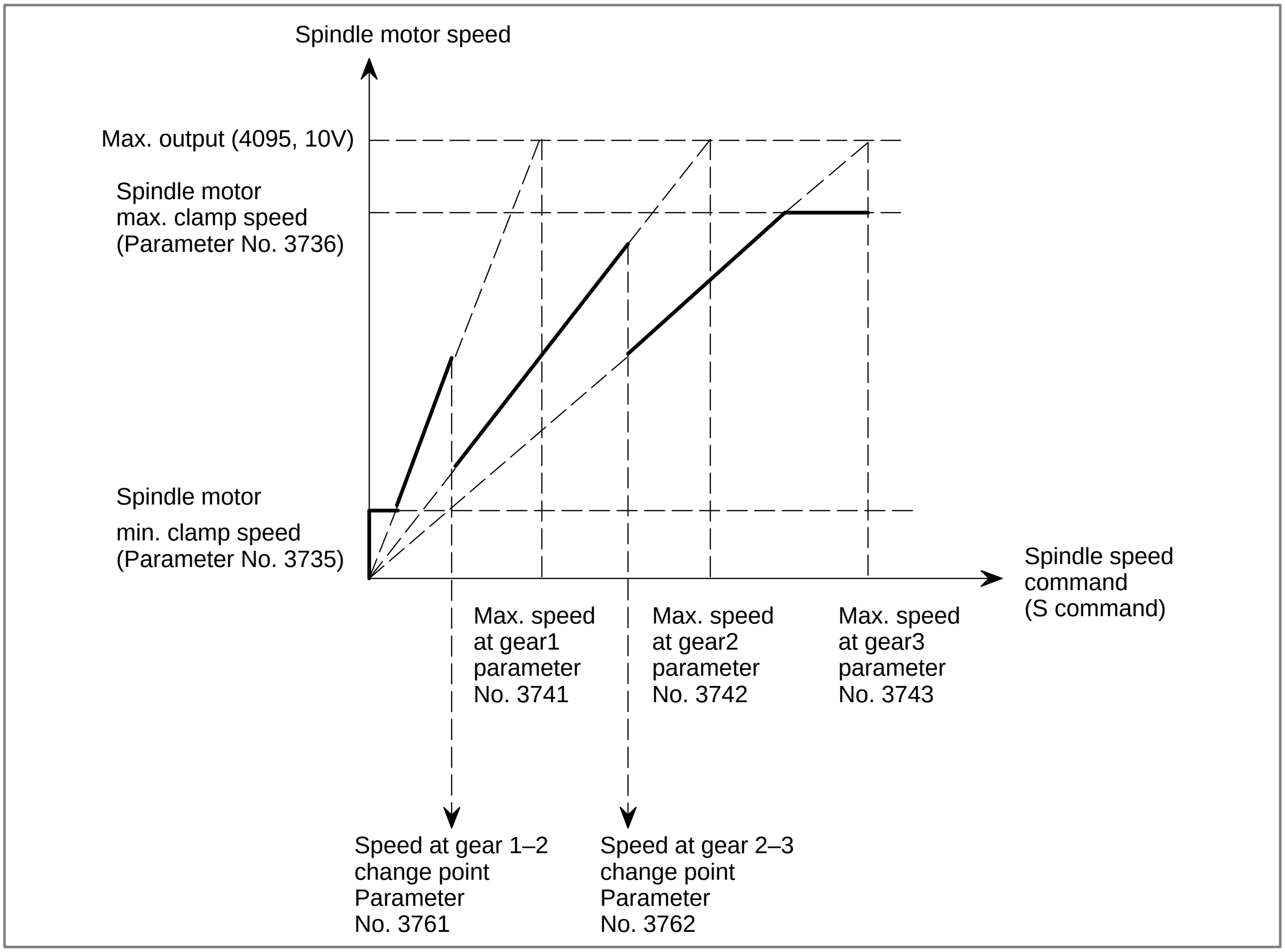
3762	
	Spindle speed when switching from gear 2 to gear 3 during tapping

[Data type] Word

[Unit of data] rpm

[Valid data range] 0 to 32767

When method B is selected (SGT,#3 of parameter 3705, is set to 1) for the tapping cycle gear switching method, set the spindle speed when the gears are switched.



	#7	#6	#5	#4	#3	#2	#1	#0
5101								
								FXY

[Data type] Bit

FXY The drilling axis in the drilling canned cycle is:

0 : Always the Z–axis
1 : The axis selected by the program

> **NOTE**
> For the M series, this parameter enables rigid tapping by using a basic axis (X, Y, or Z) perpendicular to the program–selected plane, or an axis parallel to that basic axis, as the tapping axis.

	#7	#6	#5	#4	#3	#2	#1	#0
5200	SRS	FHD		DOV	SIG	CRG	VGR	G84
		FHD	PCP	DOV	SIG	CRG	VGR	G84

[Data type] Bit

G84 Method for specifying rigid tapping

0 : An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No. 5210).

1 : An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

VGR Any gear ratio between spindle and position coder in rigid tapping

0 : Not used (The gear ratio is set in parameter No. 3706.)

1 : Used (The gear ratio is set by parameters Nos. 5221 through 5224 and 5231 through 5234.)

CRG Rigid mode when a rigid mode cancel command is specified (G80, G01 group G code, reset, etc.)

0 : Canceled after rigid mode signal RGTAP is set to 0.

1 : Canceled before rigid mode signal RGTAP is set to 0.

SIG When gears are changed for rigid tapping, the use of SIND <G032 and G033> is

0 : Not permitted.

1 : Permitted.

DOV Override during extraction in rigid tapping

0 : Invalidated

1 : Validated (The override value is set in parameter No. 5211.)

PCP Rigid tapping

0 : Used as a high–speed peck tapping cycle

1 : Not used as a high–speed peck tapping cycle

FHD Feed hold and single block in rigid tapping

0 : Invalidated

1 : Validated

SRS To select a spindle used for rigid tapping in multi–spindle control:

0 : The spindle selection signals SWS1 and SWS2 (bits 0 and 1 of G027) are used. (These signals are used also for multi–spindle control.)

1 : The rigid tapping spindle selection signals RGTSP1 and RGTSP2 (bits 4 and 5 of G061) are used. (These signals are provided expressly for rigid tapping.)

	#7	#6	#5	#4	#3	#2	#1	#0
5201						TDR		
						TDR		NIZ

[Data type] Bit

NIZ Smoothing in rigid tapping is:

0 : Not performed.

1 : Performed.

TDR Cutting time constant in rigid tapping

0 : Uses a same parameter during cutting and extraction (Parameter Nos. 5261 through 5264)

1 : Not use a same parameter during cutting and extraction

Parameter Nos. 5261 to 5264: Time constant during cutting

Parameter Nos. 5271 to 5274: Time constant during extraction

	#7	#6	#5	#4	#3	#2	#1	#0
5202								
								ORI

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

[Data type] Bit

ORI When rigid tapping is started:

0 : Spindle orientation is not performed.

1 : Spindle orientation is performed.

NOTE
This parameter can be used only for a serial spindle.
The spindle orientation is a zero return in the serial spindle servo mode The stop position can be changed by serial spindle parameter No. 4073.

	#7	#6	#5	#4	#3	#2	#1	#0
5204								DGN

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

[Data type] Bit

DGN On the diagnosis screen:

0 : A rigid tapping synchronization error is displayed. (Nos. 455 to 457)

1 : An error difference between the spindle and tapping axis is displayed. (Nos. 452 and 453)

5210	Rigid tapping mode specification M code

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets an M code that specifies the rigid tapping mode.
To set an M code larger than 255, set it to parameter No. 5212.

NOTE
The M code is judged to be 29 (M29) when “0” is set.

5211	Override value during rigid tapping extraction

[Data type] Byte

[Unit of data] 1 %

[Valid data range] 0 to 200

The parameter sets the override value during rigid tapping extraction.

NOTE
The override value is valid when DOV in parameter No. 5200#4 is “1”.

5212	M code that specifies a rigid tapping mode

[Data type] Two–word

[Unit of data] Integer

[Valid data range] 0 to 65535

This parameter sets the M code that specifies the rigid tapping mode.

The M code that specifies the rigid tapping mode is usually set by parameter 5210. To use an M code whose number is greater than 255, specify the code number with parameter 5212.

NOTE
If the setting of this parameter is 0, the M code specifying the rigid tapping mode is determined by the setting of parameter 5210. Otherwise, it is determined by the setting of parameter 5212. The setting of parameter 5212 must always be within the above valid range.

5213	Return or clearance in peck tapping cycle

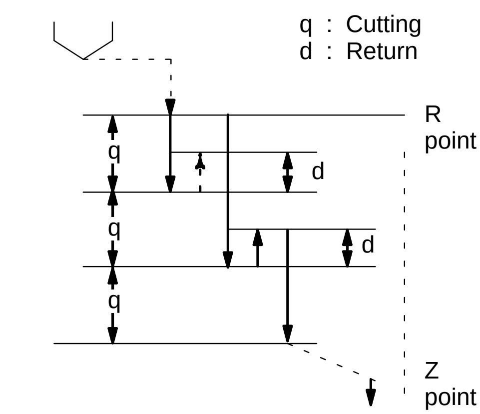
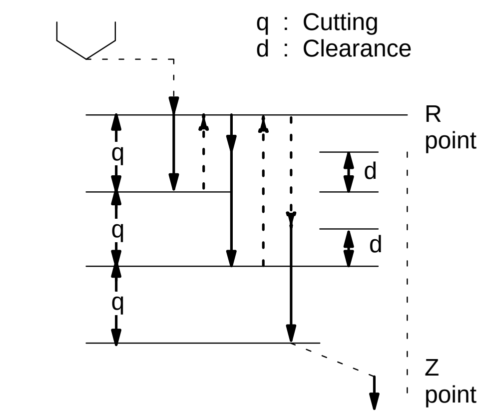
[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
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 PCP in parameter No. 5200 is 0 (High–speed peck tapping cycle)	When bit PCP in parameter No. 5200 is 1 (Peck tapping cycle)
	

5214	Rigid tapping synchronization error range setting

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets an allowable synchronization error range for rigid tapping.

When the synchronization error exceeds the allowable range set with this parameter, alarm No. 741 is issued. Note that when 0 is set with this parameter, no synchronization error check is performed.

5221	Number of gear teeth on the spindle side in rigid tapping (First gear)
5222	Number of gear teeth on the spindle side in rigid tapping (Second gear)
5223	Number of gear teeth on the spindle side in rigid tapping (Third gear)
5224	Number of gear teeth on the spindle side in rigid tapping (Fourth gear)

[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 VGR, #1 of parameter No. 5200, is “1”.

Set the same value to parameter Nos. 5221 to 5224 when the spindle has a position coder.

5231	Number of gear teeth on the position coder side in rigid tapping (First gear)
5232	Number of gear teeth on the position coder side in rigid tapping (Second gear)
5233	Number of gear teeth on the position coder side in rigid tapping (Third gear)
5234	Number of gear teeth on the position coder side in rigid tapping (Fourth gear)

[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 VGR, #1 of parameter No. 5200, is "1".
> Set the same value to parameter Nos. 5231 to 5234 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).

5241	Maximum spindle speed in rigid tapping (First gear)
5242	Maximum spindle speed in rigid tapping (Second gear)
5243	Maximum spindle speed in rigid tapping (Third gear)
5244	Maximum spindle speed in rigid tapping (Fourth gear)

[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. 5241 to parameter No. 5243. In a system having two–stage gear, set the same value as parameter No. 5242 to parameter No. 5243. If it is not set as such, P/S alarm no. 200 will be informed.
These are applicable to M series.

5261	Acceleration/deceleration time constant for every gear in rigid tapping (First gear)
5262	Acceleration/deceleration time constant for every gear in rigid tapping (Second gear)
5263	Acceleration/deceleration time constant for every gear in rigid tapping (Third gear)
5264	Acceleration/deceleration time constant for every gear in rigid tapping (Fourth gear)

[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. 5241 and greater). The actual time constant is a proportional value between the maximum spindle speed and the specified spindle speed.

5271	Acceleration/deceleration time constant during extraction in rigid tapping (First gear)
5272	Acceleration/deceleration time constant during extraction in rigid tapping (Second gear)
5273	Acceleration/deceleration time constant during extraction in rigid tapping (Third gear)
5274	Acceleration/deceleration time constant during extraction in rigid tapping (Fourth gear)

[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 TDR, #2 of parameter No. 5201, is “1”.

5280	Position control loop gain of spindle and tapping axis in rigid tapping (Common in each gear)

[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. 5281 through 5284. 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.

5281	Position control loop gain of spindle and tapping axis in rigid tapping (First gear)
5282	Position control loop gain of spindle and tapping axis in rigid tapping (Second gear)
5283	Position control loop gain of spindle and tapping axis in rigid tapping (Third gear)
5284	Position control loop gain of spindle an tapping axis in rigid tapping (Fourth gear)

[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. 5280 to “0”.

5291	Spindle loop gain multiplier in the rigid tapping mode (for gear 1)
5292	Spindle loop gain multiplier in the rigid tapping mode (for gear 2)
5293	Spindle loop gain multiplier in the rigid tapping mode (for gear 3)
5234	Spindle loop gain multioplier in the rigid tapping mode (for gear4)

[Data type] Word

[Unit of data]

[Valid data range] 0 to 32767

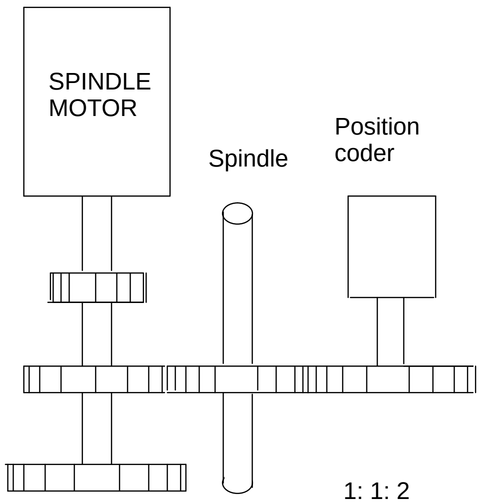
Set the spindle loop gain multipliers for gears 1 to 4 in the rigid tapping mode. The thread precision depends on the multipliers. Find the most appropriate multipliers and 1OOP gain by conducting the cutting test.

> **NOTE**
> These parameters are used for analog spindles.

$$\text{Loop gain multiplier} = 2048 \times \frac{E}{L} \times \alpha \times 1000$$

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 × 1.667/360 × 0.17578 × 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.

5300	In–position width of tapping axis in rigid tapping

[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.

5301	In–position width of spindle in rigid tapping

[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.

5310	Limit value of tapping axis positioning deviation during movement in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a tapping axis positioning deviation during movement in rigid tapping.
To set a value larger than this value, set it to No. 5314.

WARNING
The setting value is represented in a 10–times unit when a high–resolution transducer is used.

5311	Limit value of spindle positioning deviation during movement in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a spindle positioning deviation during movement in rigid tapping.

$$\text{Limit value} = \frac{S \times 360 \times 100 \times 1.5}{60 \times G \times \alpha}$$

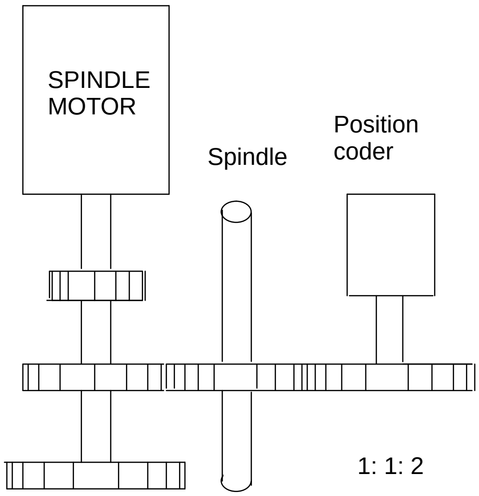
where

S: Maximum spindle speed in rigid tapping
(Setting value of parameter Nos. 5241 and greater)

G: Loop gain of rigid tapping axis
(Setting value of parameter Nos. 5280 and greater)

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.

5312	Limit value of tapping axis positioning deviation during stop in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a tapping axis positioning deviation during stop in rigid tapping.

5313	Limit value of spindle positioning deviation during stop in rigid tapping

[Data type] Word

[Unit of data] Detection unit

[Valid data range] 1 to 32767

This parameter sets the limit value of a spindle positioning deviation during stop in rigid tapping.

5314	Limit value of position deviation during movement along the tapping axis for rigid tapping

[Data type] Two–word

[Unit of data] Detection unit

[Valid data range] 0 to 99999999

Parameter No. 5310 usually sets the limit of positional deviation during movement along the tapping axis for rigid tapping. To specify a setting exceeding the valid range specified in parameter No. 5310 according to the resolution of the detector to be used, specify the limit value with parameter No. 5314.

> **NOTE**
> If the setting of this parameter is 0, the setting of parameter No. 5310 is enabled. Otherwise, the setting of parameter No. 5310 is disabled, and the setting of parameter No. 5314 is enabled.

5321	Spindle backlash in rigid tapping (First gear)
	Spindle backlash in rigid tapping
5322	Spindle backlash in rigid tapping (Second gear)
5323	Spindle backlash in rigid tapping (Third gear)
5324	Spindle backlash in rigid tapping (Fourth gear)

[Data type] Byte

[Unit of data] Detection unit

[Valid data range] 0 to 127

These parameters set the spindle backlash in rigid tapping.

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 (No.5241 to 5243). 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)	Plane changeover was instructed in the rigid mode. Correct the program.
207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) stops is larger than the set value. Note) Limit value must be set to parameter No.5312 for each axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–8 of rigid tapping axis) moves is larger than the set value. Note) Limit value must be set to parameter No. 5310 or 5314 for each axis.
413	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.

Number	Message	Description
740	RIGID TAP ALARM; EX-CESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping.
741	RIGID TAP ALARM; EX-CESS ERROR	Position deviation value of spindle at move exceeded a set value during rigid tapping or synchronous error exceeded a set value (parameter No. 5214) during rigid tapping.
742	RIGID TAP ALARM; LSI OVER FLOW	LSI overflow has occurred on the spindle side during rigid tapping.

9.11.10 Notes

NOTES ON SPINDLES

CAUTION

1 When using an analog spindle, set the spindle speed offset value parameter (No. 3731) accurately. For the standard system, a value within –8191 to 8191 must be specified in this parameter. To perform rigid tapping, a value within –1023 to 1023 must be specified.
If the spindle speed offset is set inaccurately, the spindle is stopped and placed in in–position wait state when tapping is started.
In rigid tapping with a serial spindle, no setting is required for parameter No. 3731. Be sure to set 0.

2 When the threading and synchronous feed functions are enabled, the actual spindle speed during rigid tapping is indicated correctly. When an arbitrary gear ratio is used (by setting bit 1 (VGR) of parameter No. 5200 to 1), however, the actual spindle speed will not be indicated correctly in normal spindle mode.
When the T series is used, for example, information about the actual spindle speed is important for lathe machining. So, be particularly careful when using an arbitrary gear between the spindle and position coder.

NOTE

1 A spindle pitch error is not compensated for in rigid tapping mode. Drift compensation is not made with an analog spindle.

2 The maximum number of pulses that can be distributed to the spindle is:
- 32,767 pulses per 8 msec for a serial spindle
- 4,096 pulses per 8 msec for an analog spindle
(This information is displayed by selecting No. 451 on the diagnosis screen.)

These values vary with the position coder gear ratio setting and rigid tapping specification. If a value greater than the maximum allowable number is specified, P/S alarm No. 202 is issued.

Notes on using functions such as the spindle positioning function at the same time

CAUTION

1 When the spindle orientation function is to be used at the same time
The spindle orientation function positions the spindle by using sensors and the PMC, without being directly controlled by the CNC.
The CNC has no direct control over this processing, instead following the specifications of the spindle orientation function being used.

2 When the spindle positioning function is to be used at the same time
When the spindle positioning function is to be used together with rigid tapping, rigid tapping mode must not be specified in spindle indexing mode, and spindle indexing mode must not be specified in rigid tapping mode. (Spindle positioning and rigid tapping cannot be performed simultaneously for a single spindle.)
This restriction does not apply, however, when multi–spindle control is applied; rigid tapping can be performed using the second spindle.
The spindle positioning function is effective for the first spindle only. This means that when spindle indexing is performed with the first spindle, rigid tapping can be specified with the second spindle.

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 2nd spindle, the rigid tapping can be specified to the 2nd spindle during the Cs contouring control of the 1st spindle.

Position control loop gain switching and serial spindle parameters

In rigid tapping, the loop gain of the tapping axis is switched so that the loop gains for position control of the tapping axis and spindle match each other.

This switching processing is specified by parameter Nos. 5280, and 5281 to 5284. The contents of the processing vary with whether the spindle is an analog or serial spindle, as described below.

· When the spindle is an analog spindle, the loop gains of the spindle and tapping axis are switched according to the values set in these parameters.

· When the spindle is a serial spindle, the loop gain of the tapping axis is switched according to the values set in these parameters. The loop gain of the spindle depends on the values set in the serial spindle parameters and applied gear signals (CTH2, CTH1).

Accordingly, to perform rigid tapping with a serial spindle, the loop gain for position control of the spindle must be set in the serial spindle parameters used for rigid tapping.

When multi–spindle control is being used, rigid tapping can also be performed for the second spindle. For the serial spindle used for rigid tapping, set the parameters indicated below.

The parameters indicated below are the major serial spindle parameters required for the setting and adjustment needed to use a serial spindle.

For details of the serial spindle parameters, refer to the “FANUC AC Spindle Motor α series DESCRIPTIONS (B–65152E) or FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E).

4044	Proportional gain of the velocity loop in servo mode (gear 1, gear 2)
4045	Proportional gain of the velocity loop in servo mode (gear 3, gear 4)

[Unit of data]

[Valid data range] 0 to 32767

Set a proportional gain for the velocity loop in a servo mode (such as rigid tapping mode).

4052	Integral gain of the velocity loop in the servo mode (gear 1, gear 2)
4053	Integral gain of the velocity loop in the servo mode(gear 3, gear 4)

[Unit of data]

[Valid data range] 0 to 32767

Set an integral gain of the velocity loop in a servo mode (such as rigid tapping mode).

4065	Position gain in the servo mode (HIGH) (CFPGH)
4066	Position gain in the servo mode (MEDIUM HIGH) (CFPGMH)
4067	Position gain in the servo mode (MEDIUM LOW) (CFPGML)
4068	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

1 Set a loop gain for spindle position control in rigid tapping using a serial spindle. In these parameters, basically, set the same values as those set in parameter Nos. 5280 and 5281 to 5284 (loop gains for position control of the tapping axis).
Which serial spindle parameter (i.e., loop gain) is actually used to operate the spindle depends on the serial spindle clutch/gear selection signals CTH1 and CTH2 (G070#3, #2 for the first spindle, and G074#3, #2 for the second spindle). Accordingly, which parameter is to be used must be determined by considering the gear switching and PMC software.

The table below indicates the relationship between the spindle gear selection signals and selected gear numbers.

<table>
<tr><th>CTH1</th><th>CTH2</th><th>Gear selected</th><th colspan="3">Parameter No. to be used</th></tr>
<tr><td>0</td><td>0</td><td>HIGH</td><td>4065</td><td rowspan="2">4044</td><td rowspan="2">4052</td></tr>
<tr><td>0</td><td>1</td><td>MEDIUM HIGH</td><td>4066</td></tr>
<tr><td>1</td><td>0</td><td>MEDIUM LOW</td><td>4067</td><td rowspan="2">4045</td><td rowspan="2">4053</td></tr>
<tr><td>1</td><td>1</td><td>LOW</td><td>4068</td></tr>
</table>

9.11.11 Reference Item

<table>
<tr><td rowspan="2">Series 16i/160i/18i/180i</td><td>OPERATOR’S MANUAL
(For Machining Center)
(B–63014EN)</td><td>II.13.2</td><td>RIGID TAPPING</td></tr>
<tr><td>OPERATOR’S MANUAL
(For Lathe) (B–63004EN)</td><td>II.13.8</td><td>RIGID TAPPING</td></tr>
<tr><td rowspan="2">Series 21i/210i</td><td>OPERATOR’S MANUAL
(For Machining Center)
(B–63094EN)</td><td>II.13.2</td><td>RIGID TAPPING</td></tr>
<tr><td>OPERATOR’S MANUAL
(For Lathe) (B–63084EN)</td><td>II.13.7</td><td>RIGID TAPPING</td></tr>
<tr><td rowspan="2"></td><td rowspan="2">CONNECTION MANUAL
(This manual)</td><td>9.3</td><td>SPINDLE SPEED CON-
TROL</td></tr>
<tr><td>9.10</td><td>MULTI–SPINDLE</td></tr>
<tr><td></td><td>FANUC AC SPINDLE
MOTOR α series
(Serial Interface)
DESCRIPTIONS (B–65162E)</td><td>11.4</td><td>RIGID TAPPING</td></tr>
<tr><td></td><td>FANUC AC SPINDLE
MOTOR α series
PARAMETER MANUAL</td><td>2.3</td><td>RIGID TAPPING</td></tr>
</table>

9.12 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
T series (two–path control)	First serial spindle at tool post 1	First serial spindle at tool post 2
M series (two–path control)	First serial spindle at each path	Second serial spindle at each path

Supplementary description

For details of synchronous–spindle connection, see the description of serial spindles.

The following description relates to this CNC.

- Synchronous control of spindle phase is executed when the signal for controlling the spindle phases in synchronization is entered in spindle synchronization control mode (after output of the signal indicating that the synchronous control of spindle speed has been completed). The signal indicating that the synchronous control of spindle phase is completed is output when the difference between the error pulses of the two spindles does not exceed the number of pulses specified in parameter No. 4810 of the NC function.
  The positions of spindle phase synchronization can be specified in spindle parameter No. 4034 on each of tool post 1 and tool post2.

  When the two spindles are subject to spindle–phase synchronous–control (until the spindle–phase synchronous–control completion signal, FSPPH <F044#3>, turns to “1”), they are not synchronized with each other.

  Do not specify spindle–phase synchronous control while the two spindles are holding a workpiece. Specifying this item causes phase synchronous control to start automatically.
- PMC signal, SYCAL <F044#4> is provided to monitor a synchronization errors between spindles for which spindle synchronization control or synchronous control of spindle phase is in effect. The synchronization error between the two spindles is always monitored. The SYCAL signal is set to 1 when the error (the absolute value of the error pulse) specified in parameter No. 4811 of tool post 1 is exceeded, and set to 0 when not exceeded.

- Constant surface speed control can be executed in synchronization control even while a workpiece is being held with the two spindles. However, if the speed is to change in excess of the specified time constant, the speed changes within the extent specified by time constant.
- The maximum speed in synchronization control is determined by the maximum speed of the spindle motor of master spindle (parameter No. 4020).

(Example) Maximum speed of the spindle motor of tool post 1: 6000 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 No. 3741, 3742, 3743 and 3744. When the value set in the parameter corresponding to the selected gear is 0, the rotations of the spindles are not synchronized even if a command is entered in the R01I to R12I signals.
- The S command for the master spindle and the PMC control signal for spindle control become effective when issued before spindle synchronization control or synchronous control of spindle phase are put into effect. The S command issued in synchronization control becomes effective for the first spindle immediately after synchronization control is canceled.
- The load may change due to cutting (or threading). When the load changes in spindle synchronization control, the spindle speed may change and the signal indicating that the synchronous control of spindle speed is completed may go off temporarily.
- Parameters No. 4800 #0 (for the master spindle) and #1 (for the slave spindle) are used to set the direction of rotation of the first spindle and second spindle, respectively.
- The gear ratio of the spindle to the position coder must be set to one–to–one.
- In spindle synchronization control , the compensation value for spindle speed offset (parameter No. 3731) is disabled.

- A spindle–phase synchronous control command is effective only in synchronous spindle control mode. The specified phase can be repeatedly changed under synchronous control.

Signal

See the manual of serial spindles.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
4800							ND2	ND1

[Data type] Bit type

ND1 In controlling the spindle synchronization, the direction of the first spindle (master spindle) motor rotation is:

0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

ND2 In controlling the spindle synchronization, the direction of the 2nd spindle (slave spindle) motor rotation is:

0 : The direction indicated by the command sign
1 : The opposite direction to that indicated by the command sign

4810	Error pulse between two spindles when synchronizing phases in the serial spindle synchronization control mode

[Data type] Byte type

[Unit of data] Pulse

[Valid data range] 0 to 255

Set the difference in error pulses between two spindles when synchronizing phases in the serial spindle synchronization control mode.

When the difference in error pulse between two spindles is within the value set in this parameter, the spindle phase synchronization completion signal FSPPH <F044#3> becomes “1”.

This parameter is used to check the difference in phase in synchronization control and to confirm the completion of synchronization in the serial spindle synchronization control mode.

4811	Allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode

[Data type] Word type

[Unit of data] Pulse

[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode.

This parameter is used to output the inter–spindle phase error detection signal SYCAL in the serial spindle synchronization control mode. The SYCAL <F044#4> signal becomes “1” when a phase error exceeding the value set in this parameter is found.

Alarm and message

Number	Message	Description
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.

Note

NOTE

Signal SYCAL <F044#4> is used for monitoring a phase shift in synchronous control. The processing performed when a phase shift is detected depends on the specifications determined by the machine tool builder.

Reference item

FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.7	Spindle synchronization control
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)	2.5	Spindle synchronization control

9.13 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 <G078, G079>

SHB00 to SHB11 for the second spindle <G080, G081>

SHC00 to SHC11 for the third spindle <G208, G209>

[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
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08
G208	SHC07	SHC06	SHC05	SHC04	SHC03	SHC02	SHC01	SHC00
G081					SHC11	SHC10	SHC09	SHC08

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3702					OR2	OR1		OR3

[Data type] Bit

OR1 Whether the stop–position external–setting type orientation function is used by the first spindle motor

0 : Not used
1 : Used

OR2 Whether the stop–position external–setting type orientation function is used by the second spindle motor

0 : Not used
1 : Used

OR3 Whether the stop–position external–setting type orientation function is used by the third spindle motor

0 : Not used
1 : Used

Caution

CAUTION

1 To perform spindle orientation by using the spindle control unit, the signals of the spindle control unit must be used.
  To perform serial spindle orientation by using a position coder (to perform serial spindle orientation with the stop position set externally), the serial spindle control unit signals must be used.
2 When the spindle orientation function of stop position external setting type is used, the stop position parameters in spindle orientation with a position coder (No. 4031 and 4204) are invalid.

Note

NOTE

Spindle orientation with the spindle positioning function differs from that described in this section. For details, see Section 9.8, “Spindle Positioning.”

Reference item

FANUC AC SPINDLE MOTOR α series DESCRIPTIONS (B–65162E)	11.1 11.2	Position coder method spindle orientation Magnetic sensor method spindle orientation
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)	2.1 2.2	Position coder method spindle orientation Magnetic sensor method spindle orientation

9.14 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 <F034 #0, #1>, are used (for machining centers in which constant surface speed control is not provided and GTT, bit 4 of parameter No. 3706, is set to 0)

  Set two gears, which are almost the same.

  (Example: Value of parameter No. 3741 = value of parameter No. 3742 – 1, value of No. 3742 = Maximum spindle speed)

  When parameter No. 3741 is equal to parameter No. 3742, the CNC judges that one gear is used, and does not output the GR2O signal.

- The parameters related to gear switching points, SGT, bit 3 of parameter No. 3705, and SGB, bit 2 of parameter No. 3761, parameter Nos. 3761 and 3751 can be used.

- In usual spindle control, depending on the speed at switching points, the speed specified by the spindle speed command may differ slightly from the actual speed in the area where the maximum spindle speed is set to the maximum speed ±1. (This is because the spindle motor speed, specified by the speed command, is calculated based on the settings of parameter Nos. 3741 to 3744.)

  This does not apply to rigid tapping. (Because the machine tool is controlled using the feedback signal from the detector in the position loop.)

- When gear selection input signals, GR1 and GR2 <G028 #1, #2>, are used (for lathes or machining centers in which constant surface speed control is provided or GTT, bit 4 of parameter No. 3706, is set to 1)

  Parameter settings are read according to the input signal information. Unlike the GR2O and GR1O signals, these signals do not require special parameter settings.

Example) When parameter Nos. 3741 and 3742 are set to the maximum spindle speed.

Create a PMC sequence that specifies the following.
For gear 1, set GR1 and GR2 to 0.
For gear 2, set GR2 to 0 and set GR1 to 1.

The PMC must determine the switching timing on the basis of some information.

Reference item

CONNECTION MANUAL (This manual)	9.3 9.11	Spindle control Rigid tapping
FANUC CONTROL MOTOR AMPLIFIER α series DESCRIPTIONS (B–65162E)	11.9	Output switching control
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL (B–65160E)	2.7	Output switching control

9.15 THREE–SPINDLE SERIAL OUTPUT

General

Serial spindle can be connected three in this three–spindle serial control.

The third serial spindle operates as an ordinary third analog spindle. For the third as well as the first and second serial spindles, all the functions supported by the serial spindle control unit (spindle orientation, spindle output switching, and spindle switching) can be used.

When the third spindle orientation function is used, stop–position external–setting type orientation can also be performed for the third spindle.

This section provides information related to the third serial spindle added by the three–spindle serial output function. The spindle configuration, supported functions, and related signal addresses, parameters, alarms, and diagnosis screens are explained.

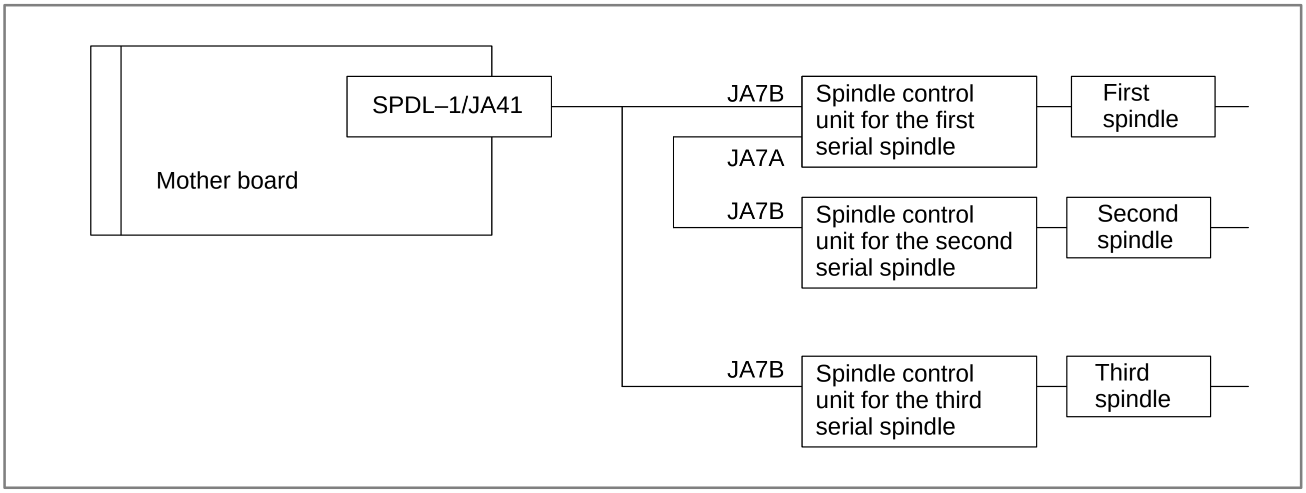
For details of the spindle control method and spindle–related functions, refer to the description of each function.

The relationships between the spindle control interfaces and spindle configuration are shown below. (This table relates to a table that appears in Section 9.2.)

Spindle serial output	Spindle analog output	Three–spindle serial 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.
○	×	×	First serial spindle The PC can be used.	Second serial spindle The PC can be used(*).	—
×	○	×	Analog spindle The PC can be used.	—	—
○	×	○ (*1)	First serial spindle The PC can be used.	Second serial spindle The PC can be used(*).	Third serial spindle The PC cannot be used.
×	×	×	Spindle function (S code output) → Controlled by the PMC via an external interface.		

- PC = Position coder
- (*) : The multi–spindle function is required to enable the use of the position coder for the second spindle.
- For an explanation of how to control the speed of the second and third spindles, see Sections 15.4 and 9.10.

- The serial spindles are connected as follows:



The table below lists the relationship between the spindles and functions. (This table relates to a table that appears in Section 9.2.)

○ = Available × = Unavailable

Function \ Spindle		Serial spindle: First spindle	Serial spindle: Second spindle	Serial spindle: Third spindle	Analog spindle: First spindle (with no serial SP option)	Analog spindle: Third spindle (with serial SP option)
Thread cutting/feed per rotation (synchronous feed)		○	○ (*3)	×	○	×
Constant surface speed control		○	○ (*3)	×	○	×
Spindle speed fluctuation detection		○	○ (*3)	×	○	×
Actual spindle speed output (T series)		○	○ (*3)	×	○	×
Polygon turning (T series) (using the servo motor axis and spindle)		○	○ (*3)	×	○	×
Polygon turning between spindles (T series) (using two spindles)		○ : Master (*4)	○ : Slave (*4)	×	×	×
Spindle positioning (T series)		○	×	×	○	×
Cs contour control		○	×	×	×	×
Multi–spindle (*1)		○ : First spindle	○ : Second spindle	○ : Third spindle	×	○ : Third axis
Spindle output control using PMC		○	○	○	○	○
Rigid tapping		○ (*5)	○ (*5)	○ (*5)	○	×
Spindle synchronous control		○ : Master (*6)	○ : Slave (*6)	×	×	×
Simple spindle synchronization		○ : Master (*7)	○ : Slave (*7)	×	×	×
Motor speed detection function		○	○	○	×	×
Abnormal load detection function		○	○	○	×	×
Functions provided by spindle control unit(*3)	Spindle orientation	○ (*8)	○ (*8)	○ (*8)	○	○
	Spindle output switching	○	○	○	○	○
	Spindle switching	○	○	○	○	○

NOTE

1 The multi–spindle function can control the speed of the three spindles and switch the feedback signal between two position coders. It can operate without the second or third spindle.
When the multi–spindle function is used with the M series, the constant surface speed control option must be set, or the spindle gear selection method must be set to type T with GTT (bit 4 of parameter No. 3706).

2 These functions are provided by the spindle control unit. They cannot be used unless supported by the spindle control unit.

3 The multi–spindle function is required. The function cannot be applied to the first and second spindles simultaneously.

4 When a two–path lathe is used, spindle polygon turning cannot be performed using the spindle on tool post 1 and the spindle on tool post 2.

5 The first, second, or third spindle can be selected and used for rigid tapping. In this case, the multi–spindle function is required. If the multi–spindle function is not available, rigid tapping can be performed using the first spindle only.

6 For a two–path lathe application, the first spindle on tool post 1 is the master, while the first spindle on tool post 2 is the slave. The second spindle of either tool post cannot be used in spindle synchronization.

7 For a two–path lathe application, simple spindle synchronous control cannot be applied to the spindle on tool post 1 and the spindle on tool post 2.

8 The function used to change the spindle orientation stop position (stop–position external–setting type spindle orientation function) can be used by writing G signal addresses from the PMC.

Signal

- **Spindle control unit signals for the third serial spindle <G0204 to G0207> (input), <F0168 to F0171> (output) → for the third serial spindle**

These addresses are on the CNC. Actually, however, they are input/output signals for the serial spindle control unit.

For details of the signals belonging to these addresses, refer to the following manuals:

“FANUC CONTROL MOTOR AMPLIFIER α Series Descriptions” (B–65162E)
“FANUC AC SPINDLE MOTOR α Series Descriptions” (B–65152E)
“FANUC CONTROL MOTOR α Series Maintenance Manual” (B–65165E)

- **Spindle orientation stop position external command for the third spindle SHC00 to SHC11 <G0208, G0209>**

For details, see Section 9.13.

- **Motor speed detection for the third spindle DSP3 <Y(n + 1) #2> (n = setting in parameter No. 1891)**

See Section 2.11 for details.

- **Abnormal load detection for the third spindle ABTSP3 <F0090#3>**

See Section 2.10 for details.

Signal address

- **Spindle control unit signals for the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
G0204	MRDYC	ORCMC	SFRC	SRVC	CTH1C	CTH2C	TLMHC	TLMLC
G0205	RCHC	RSLC	INTGC	SOCNC	MCFNC	SPSLC	*ESPC	ARSTC
G0206	RCHHGC	MFNHGC	INCMDC	OVRIDC	DEFMDC	NRROC	ROTAC	INDXC
G0207						MPOFC	SLVC	MORCMC

	#7	#6	#5	#4	#3	#2	#1	#0
F0168	ORARC	TLMC	LDT2C	LDT1C	SARC	SDTC	SSTC	ALMC
F0169	MORA2C	MORA1C	PORA2C	SLVSC	RCFNC	RCHPC	CFINC	CHPC
F0170							INCSTC	PC1DEC
F0171								

- **Spindle orientation stop position external command for the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
G0208	SHC07	SHC06	SHC05	SHC04	SHC03	SHC02	SHC01	SHC00
G0209					SHC11	SHC10	SHC09	SHC08

- **Motor speed detection for the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
Y (n+1)						DSP3		

NOTE
"n" above is the value set in parameter No. 1891.

- **Abnormal load detection for the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
F0090					ABTSP3			

Parameter

- **Connection of serial spindle control unit**

	#7	#6	#5	#4	#3	#2	#1	#0
3701			SS3	SS2			ISI	

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

[Data type] Bit type

ISI Specifies whether to use the first and second spindle interfaces.

0 : Use these interfaces.
1 : Do not use these interfaces.

> **NOTE**
> This parameter is valid when the spindle serial output option is provided.
> This bit is used when the CNC is started with serial interface control for the first and second serial spindles temporarily disabled to enable startup adjustment of the CNC.
> This bit should normally be set to 0.
> Similarly, when the serial interface for the third serial spindle is disabled to enable CNC startup adjustment, set SS3, bit 5 of parameter No. 3701, to 0. (ISI cannot disable the serial interface for the third serial spindle.)

SS2 Specifies whether to use the second serial spindle in serial spindle control.

0 : Do not use the second serial spindle.
1 : Use the second serial spindle.

> **NOTE**
> This bit is valid when the spindle serial output option is provided, and ISI, bit 1 of parameter No. 3701 is set to 0.

SS3 Specifies whether to use the third serial spindle in serial spindle control.

0 : Do not use the third serial spindle.
1 : Use the third serial spindle.

NOTE
This bit is valid when the spindle serial output and three–spindle serial output options are provided.

Parameter setting		Serial spindles to be used
SS3	SS2	
×	×	First serial spindle only
×	○	First and second serial spindles
○	○	First, second, and third spindles

- **Parameters of serial spindle control unit**

No. 4000 – 4351: S1 → For 1st serieal spindle
S2 → For 2nd serial spindle
S3 → For 3rd 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 CONTROL MOTOR α series Maintenance Manual (B–65165E).

- **Stop–position external–setting type spindle orientation**

	#7	#6	#5	#4	#3	#2	#1	#0
3702								OR3

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

[Data type] Bit type

OR3 Specifies whether to use the stop–position external–setting type spindle orientation function for the third spindle.

0 : Do not use the function.
1 : Use the function.

NOTE
When the stop–position external–setting type spindle orientation function is used, the parameters for setting the spindle orientation stop position using a position coder (parameter Nos. 4031 and 4204) are ignored.

Alarm and message

Number	Message	Contents
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be the disconnection of an optical cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated. The spindle for which this error occurred can be determined from diagnosis No. 408 (the first or second spindle) and diagnosis No. 438 (the third spindle).
750	SPINDLE SERIAL LINK ERROR	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) Noise, 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 SS2, bit 4 of parameter No. 3701, is 1) or the third spindle (when SS3, bit 5 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). This alarm does not occur after the system including the spindle control unit is activated. See diagnostic display No. 409 (the first or second spindle) or No. 439 (the third spindle) for details.
771	SPINDLE–3 ALARM DETECT (AL–XX)	This alarm indicates in the NC that an alarm is generated in the third spindle unit of the system with the third serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to **(11) Alarms displayed on third spindle servo unit** .The alarm number XX is the number indicated on the third spindle amplifier. The CNC holds this number and displays on the screen.
772	SPINDLE–3 MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include 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.
774	SPINDLE–3 ABNORMAL TORQUE ALM	An abnormal load on the first spindle motor was detected. The alarm can be released by a reset.

Diagnosis screen

- **Information relating to third serial spindle control**

	#7	#6	#5	#4	#3	#2	#1	#0
430					SS3	SSR		SIC

SIC 0 : The module required for the three–spindle serial output function is not installed.

1 : The module required for the three–spindle serial output function is installed.

SSR 0 : The options required for the three–spindle serial output function are not installed.

1 : The options required for the three–spindle serial output function are installed.

SS3 0 : The third serial spindle is not used in the three–spindle serial output function.

1 : The third serial spindle is used in the three–spindle serial output function.

431	Alarm condition for the serial spindle unit for the third spindle (AL–??)

- **Communication error for the spindle serial output interface of the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
438	SSA		SCA	CME	CER	SNE	FRE	CRE

CRE 1 : CRC error (warning)

FRE 1 : Framing error (warning)

SNE 1 : Mismatch between sending and receiving sections

CER 1 : Abnormal reception

CME 1 : No answer during auto scanning

SCA 1 : Communication alarm in the spindle amplifier

SSA 1 : System alarm in the spindle amplifier

(The above are errors related to the third serial spindle. They are reflected in spindle alarm 749. They are usually caused by noise, disconnection, or instantaneous power interruption.

- **Information related to the activation of the spindle serial output interface for the third serial spindle**

	#7	#6	#5	#4	#3	#2	#1	#0
439					SPE		S3E	SHE

SHE 1 : Abnormal operation in serial spindle communication module on the mother board of the CNC

S3E 1 : Abnormal operation of the third spindle during activation

SPE 1 : Serial spindle parameter does not satisfy activation conditions

(The above are errors related to the third serial spindle. They are reflected in spindle alarm 750.)

- **Load and speed meter readings for the third serial spindle**

440	Third serial spindle: Load meter reading (%)
441	Third serial spindle: Speed meter reading (rpm)

To display the load and speed meter readings, the following parameters must be specified:
Maximum motor speed: Set for each axis in parameter Nos. 4020 (main) and 4196 (sub)
Load meter reading at maximum output: Set for each axis in parameter Nos. 4127 (main) and 4276 (sub)

NOTE
The spindle switch function is used for main/sub switching. Select main when the spindle switch function is not being used.

9.16 SIMPLE SPINDLE SYNCHRONOUS CONTROL

General

In simple spindle synchronous control mode, the second spindle can be controlled as a slave axis of the first spindle.

Thus, control based on the Cs contour axis control function, rigid tapping function, and spindle positioning function (T series) can be exercised over the second spindle, under the control of the first spindle.

Note, however, that unlike spindle synchronous control, simple spindle synchronous control does not guarantee synchronization between the first and second spindles. (For details of spindle synchronous control, see Section 9.12.)

To realize simple spindle synchronous control, two serial spindle systems suitable for two–spindle connection are required. Moreover, both spindles must be fitted with spindle–related hardware such as detectors for the functions used with simple spindle synchronous control (Cs contour axis control function, rigid tapping function, and spindle positioning function (T series)).

Simple spindle synchronous control mode is set by applying simple spindle synchronous control signal ESRSYC from the PMC.

During simple spindle synchronous control, the same commands as those that are usually used for the first spindle are used in spindle mode, Cs contour axis control mode, rigid tapping mode, and spindle positioning (T series).

When the parking function is used, one spindle can be stopped in each of the control modes described below. For details, see the description of the parking function.

- **Operation in each control mode**

1. Spindle mode (ordinary spindle control)
   The second spindle rotates upon the issue of the same command as that used for the first spindle.
   The command does not specify the speed of the spindle, instead specifying the ratio of the spindle motor speed to the maximum speed. So, if the spindle unit configuration (maximum motor speed, motor–spindle gear ratio, and so forth) of the first spindle is the same as that of the second spindle, both spindles will rotate at about the same speed. (The synchronization of rotation cannot be not guaranteed. If the configurations of the two spindles differ, the speed of one spindle will differ from that of the other.) For details of spindle control, see Section 9.3.

2. Cs contour control (contour axis control based on the spindle motor)
   When the Cs contour axis control option is selected, contour control can also be applied to the second spindle. Note, however, that independent control is not possible, the motion of the second spindle being controlled by the first spindle. The display of the move command address and position data is limited to the first spindle. Reference position return is also performed in the same way as usual. However, both spindles are checked for the completion of reference position return, so that the reference position return signal for the Cs contour control axis is turned on only after reference position return has been completed by both spindles.
   While simple spindle synchronous control is applied, the user can switch between spindle mode and Cs contour control mode. For details of Cs contour control, see Section 9.9.

3. Rigid tapping
   When the rigid tapping option is selected, the second spindle supports rigid tapping under the control of a rigid tapping command for the first spindle. (Rigid tapping for the second spindle cannot be performed independently of the first spindle.)
   During diagnosis, spindle data such as any positional deviation is displayed for the first spindle.
   For details of rigid tapping, see Section 9.11.

4. Spindle positioning (T series)
   When the spindle positioning option is selected, the spindle positioning function for the second spindle is enabled under the control of a spindle positioning command for the first spindle. (The spindle positioning function for the second spindle cannot be used independently of the first spindle.)
   In the same way as with Cs contour axis control, the display of command address and position data is limited to the first spindle. Operations such as orientation are performed in the same way as usual. However, both spindles are checked for the completion of orientation, and the spindle positioning sequence is processed only after the completion of positioning by both spindles has been confirmed.
   For details of spindle positioning, see Section 9.8.

• Parking function

In simple spindle synchronous control mode, the parking function stops the motion of the first or second spindle, regardless of the mode (spindle mode, spindle positioning mode, Cs contour axis control mode, or rigid tapping mode) of the spindle.

In the parking state, the spindle is placed in the following state, depending on its mode:
In spindle mode: Same state as when S0 is specified
In other modes: Zero distribution state

Spindle orientation in spindle positioning mode and reference position return in Cs contour axis control mode are also disabled for a spindle that has been placed in the parking state.

The parking function stops only spindle motion. So, the parking function can be used to stop only one spindle to enable machining to continue on the other spindle when both spindles are being controlled. The parking function can also be used to place the first spindle in the parking state so that Cs contour axis control, spindle positioning, or rigid tapping can be performed only on the second spindle.

WARNING

When using parking signal PK7 or PK8 for spindle synchronous control (T series) while both simple spindle synchronous control and spindle synchronous control (T series) are being used, set the SPK bit (bit 7 of parameter No. 4800) to 1. This sets parking signals PKESS1 and PKESS2, used for simple spindle synchronous control, to #6 and #7 of G031.

CAUTION

1 If the parking function is activated in a mode featuring a position loop, such as Cs contour axis control mode, spindle positioning mode, and rigid tapping mode, the actual machine position of the spindle placed in the parking state will differ from the coordinates recognized by the CNC.
This error is causes by parking, so that an excess error alarm is not issued.
2 The parking signal becomes active immediately upon simple spindle synchronous control mode being set. However, if the parking signal is applied during reference position return in Cs contour axis control mode or during spindle orientation in spindle positioning mode, reference position return or spindle orientation continues until the completion of the operation; the parking state is set once the reference position has been established.
3 While both spindles are placed in the parking state, never specify reference position return in Cs contour axis control mode, or spindle orientation in spindle positioning mode.
When both spindles are in the parking state, reference position is not established.

NOTE

1 When the parking function is activated for a spindle in a mode featuring a position loop, such as Cs contour axis control mode, spindle positioning mode, and rigid tapping mode, the spindle is stopped at the point where the parking function is activated. If the spindle is shifted from the stop position because of the application of an external force, for example, an excess error alarm is issued. (In simple spindle synchronous control mode, both spindles are checked for any positional error, regardless of whether the parking function is being used.)

2 If reference position return is specified in Cs contour axis control mode, or if spindle orientation is specified in spindle positioning mode when one spindle is placed in the parking state, the reference position return completion signal is turned on when the reference position is established for the other spindle.

- **PMC signal control**

Even for simple spindle synchronous control, those functions (including the spindle orientation function) and input/output signals that are to be directly specified for spindle control from the PMC are controlled independently for the first and second spindles.

Specifically, control over the signals in the following areas is exercised independently for the first and second spindles, regardless of whether simple spindle synchronous control mode is enabled.
First spindle: (DGN DI–>G0070 to G0073, DO–>F0045 to F0048)
Second spindle: (DGN DI–>G0074 to G0077, DO–>F0049 to F0052)

When using the simple spindle synchronous control function, use the signals for the second spindle (DGN DI–>G0074 to G0077, DO–>F0049 to F0052) as required in addition to simple spindle synchronous control signal ESRSYC.

Basically, no conventional PMC signal addresses and sequences need to be modified for simple spindle synchronous control, except for the addition of PMC control over the second spindle.

Simple spindle synchronous control can be turned on and off by turning simple spindle synchronous control signal ESRSYC on and off.

NOTE

In simple spindle synchronous control mode, the states of the first and second spindles are checked. So, if simple spindle synchronous control is specified in Cs contour axis control mode before the second spindle is activated, for example, the VRADY OFF alarm may be issued. Careful attention should also be paid to PMC control over the second spindle.

- **Simple spindle synchronization and phase error monitor signal**

Simple spindle synchronous control does not guarantee synchronous spindle operation. However, in a control mode featuring a position loop, such as Cs contour axis control mode, rigid tapping mode, and spindle positioning mode, the synchronization of both spindles can be guaranteed by matching the position gain of one spindle to that of the other.

For this purpose, phase error monitor signal SYCAL is provided to monitor both spindles for phase error. The error between the two spindles is constantly monitored. If an error that exceeds the value set in parameter No. 4811 is detected, the signal is set to 1. If a detected error does not exceed the value set in parameter No. 4811, the signal is set to 0.

In spindle mode, however, this signal SYCAL is ignored, being set to 0 at all times.

Note that even if phase error monitor signal SYCAL is output, no CNC alarm is issued.

This signal is designed to be used by the PMC to monitor the machine synchronization state. Its usage varies depending on the machine tool builder. The use of this signal is not mandatory.

NOTE

During simple spindle synchronous control, synchronization between the first and second spindles is not guaranteed in spindle mode. However, in Cs contour axis control mode, rigid tapping mode, and spindle positioning mode, the synchronization of the spindles can be guaranteed by matching the position gain of one spindle with that of the other.
Note that spindle synchronization cannot be guaranteed if either spindle is undergoing reference position return in Cs contour axis control mode or orientation in spindle positioning mode, or if it is using a function (such as spindle orientation) that is specified directly from the PMC.

- **Relationship between simple spindle synchronous control and spindle synchronous control**

When the spindle synchronous control option is selected, ensure that simple spindle synchronous control signal ESRSYC is applied when synchronous control is not exercised.

Also, ensure that spindle synchronous control signal SRSYC is not applied during simple spindle synchronous control.

If the spindle synchronous control signal and simple spindle synchronous control signal are applied at the same time, the current mode remains set, and the NC issues alarm PS194 as a warning.

This alarm returns the value of the signal (SRSYC or ESRSYC) applied later to 0, and remains on until reset.

In a mode other than spindle synchronous control mode, simple spindle synchronous control signal ESRSYC can be issued at any time. However, the response of the second spindle varies according to the first spindle state existing when ESRSYC is applied.

No particular restrictions are imposed on the timing at which the simple spindle synchronous control signal is set to 0. When simple spindle synchronous control signal ESRSYC is set to 0, the second spindle is immediately initialized to spindle mode, regardless of its previous mode. The mode of the first spindle remains as is.

The table below summarizes the changes in the first spindle and second spindle states depending on the transition of simple spindle synchronous control signal ESRSYC and the state of spindle synchronous control signal SRSYC.

<table>
<tr><th>Transition of simple spindle synchronous control signal</th><th colspan="3">ESRSYC
0 → 1</th><th colspan="3">ESRSYC
1 → 0</th></tr>
<tr><td>State change of first spindle</td><td>SP → SP
(Remains as is)</td><td>CT → CT
(Remains as is)
(Note 1)</td><td>SV → SV
(Remains as is) (Note 2)</td><td>SP → SP
(Remains as is)</td><td>CT → CT
(Remains as is)</td><td>SV → SV
(Remains as is)</td></tr>
<tr><td>State change of second spindle</td><td>SP → SP
(Note 3)</td><td>CT → CT
(Note 1)</td><td>SV → SV
(Note 2)</td><td>SP → SP
(Note 4)</td><td>CT → CT
(Note 4)</td><td>SV → SV
(Note 4)</td></tr>
<tr><td>When the spindle synchronous control option is selected</td><td colspan="3">SRSYC 1 (during spindle synchronous control)
→ Alarm PS194 is issued.
SRSYC 0 (not during spindle synchronous control)
→ The same changes as those indicated above occur.</td><td colspan="3">SRSYC 1 (during spindle synchronous control)
→ Alarm PS194 is released by a reset.
SRSYC 0 (not during spindle synchronous control)
→ The same changes as those indicated above occur.</td></tr>
</table>

Legend
SP : Spindle mode CT: Cs contour axis control mode
SV : Rigid tapping mode, or spindle positioning mode (T series only)

NOTE

(Note 1)

The second spindle is initialized to Cs contour axis control mode. At this time, the position of the second spindle will be undefined, so that reference position return must be performed for the first and second spindles. In this case, the reference position return command that is usually used can be applied. (See Section 9.9.)

(Note 2)

While the first spindle is in rigid tapping mode or spindle positioning mode, the input of simple spindle synchronous control signal ESRSYC is masked in the NC to prevent the setting of simple spindle synchronous control mode. When the first spindle is initialized to spindle mode or Cs contour axis control mode after the cancellation of rigid tapping mode or Cs contour axis control mode, the second spindle enters simple spindle synchronous control mode, and is initialized together with the first spindle. After simple spindle synchronous control mode has been set, initialization to rigid tapping mode or spindle positioning mode, command specification, and cancellation are performed for both the first and second spindles.

(Note 3)

The mode is not changed. The second spindle, however, is initialized to spindle mode. After being initialized to spindle mode, the second spindle operates according to commands issued for the second spindle, such that the feedrate may change.

(Note 4)

The second spindle is immediately initialized to spindle mode, regardless of its previous mode. The state of the first spindle, however, remains as is.

- **Spindle rotation direction**

In simple spindle synchronous control mode, the same output and move commands are issued to both the first and second spindles.

When one spindle is to rotate in the opposite direction to that of the other spindle, the direction of rotation can be reversed by using a PMC signal (SFR or SRV signal) or serial spindle parameter according to the currently set mode.

- **Error monitoring and alarm display**

When simple spindle synchronous control is exercised in a mode other than spindle mode, feedback information relating to the first and second spindles is monitored for positional deviation. That is, an in–position check is made, and an excess error check is made upon stop and move.

The usually specified parameter is used for both spindles.

Moreover, upon the issue of an alarm to indicate the occurrence of an error, no distinction is made between the first and second spindles, the alarm being issued as a first spindle alarm.

- **Positional deviation display**

During simple spindle synchronous control, indications such as the positional deviation of the first spindle are output in the usual way. For the second spindle, however, only the positional deviation of the second spindle is indicated in diagnostic data No. 415.

The positional deviation of the first spindle is indicated in diagnostic data No. 414.

The absolute phase deviation between the first and second spindles is indicated as a synchronization error in diagnostic data No. 414.

Signal

Simple spindle synchronous control signal ESRSYC<G064#6>

[Classification] Input signal

[Function] This signal specifies switching between simple spindle synchronous control mode and ordinary spindle synchronous control mode when simple spindle synchronous control is applied.

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

– Enters simple spindle synchronous control mode.

First spindle parking signal PKESS1<G122#6> <G031#6>

[Classification] Input signal

[Function] This signal activates the parking function for the first spindle when simple spindle synchronous control is applied.

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

– Activates the parking function for the first spindle placed under simple spindle synchronous control.

When the SPK bit (bit 7 of parameter No. 4800) is set to 1, #6 of G031 functions as this signal.

Second spindle parking signal PKESS2<G122#7> <G031#7>

[Classification] Input signal

[Function] This signal activates the parking function for the second spindle when simple spindle synchronous control is applied.

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

– Activates the parking function for the second spindle placed under simple spindle synchronous control.

When the SPK bit (bit 7 of parameter No. 4800) is set to 1, #7 of G031 functions as this signal.

Phase error monitor signal SYCAL<F044#4>

[Classification] Input signal

[Function] This signal monitors the synchronization error between the two spindles when simple spindle synchronous control is applied.

[Operation] This signal is set to 1 in the following case:

– When the synchronization error between the two spindles is equal to or greater than the value set in parameter No. 4811

This signal is set to 0 in the following case:

– When the synchronization error between the two spindles is less than the value set in parameter No. 4811

Phase error monitor signal SYCAL is not a latch signal, instead being switched between 0 and 1 by constantly monitoring whether the position error between the two spindles is within the value set in parameter No. 4811.

In addition to the above signals, the following spindle control unit signals for the serial spindles must be controlled so that the second spindle functions in the same way as the first:
<G0070 to G0073> (input), <F0045 to F0048> → For the first serial spindle
<G0074 to G0077> (input), <F0049 to F0052> → For the second serial spindle

For details of the signals at these addresses, see Section 9.2. Also, refer to the following manuals:
FANUC Control Motor Amplifier α series Descriptions (B–65162E)
FANUC AC Spindle Motor α series Descriptions (B–65152E)
FANUC CONTROL MOTOR α series Maintenance Manual (B–65165E)

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G31	PKESS2	PKESS1						
G064		ESRSYC						
G122	PKESS2	PKESS1						

	#7	#6	#5	#4	#3	#2	#1	#0
F044				SYCAL				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit type

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

SS2 Under serial spindle control, the second serial spindle is:

0 : Not used.
1 : Used.

> **NOTE**
> 1. This parameter is enabled when the spindle serial output option is selected, and bit 1 (ISI) of parameter No. 3701 is set to 0.
> 2. If the spindle synchronous control option is selected, this parameter is automatically set at power–on. Once this parameter is set, the following is possible:
>   1 Confirmation of connection with, and communication with the second serial spindle amplifier
>   2 Control of the second spindle when asynchronous control is used (SIND2)
>
>   Before the simple spindle synchronous control function can be used, two serial spindles must be connected. This means that the user must set this parameter beforehand; it is not set automatically.
>   When this parameter is set, the serial spindle parameter for the second spindle must also be set.

	#7	#6	#5	#4	#3	#2	#1	#0
4800	SPK							

[Data type] Bit

> **NOTE**
> To put this parameter setting in effect, switch the power off then back on again.

SPK Specifies what is to be used as the parking signal for spindle simple synchronization control.

0 : PKESS1 <G122#6> (first spindle) and PKESS2 <G122#7> (second spindle)
1 : PKESS1 <G031#6> (first spindle) and PKESS2 <G031#7> (second spindle)

4811	Allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode

[Data type] Word type

[Unit of data] Pulse

[Valid data range] 0 to 32767

Set the allowable error count for the error pulses between two spindles in the serial spindle synchronization control mode.

NOTE

This parameter is used to output the inter–spindle phase error detection signal SYCAL in the serial spindle synchronization control mode. The SYCAL signal becomes high when a phase error exceeding the value set in this parameter is found.

When specifying this parameter to enable error pulse detection during simple spindle synchronous control, set this parameter as required, noting the mode of the spindle. (In spindle mode, this parameter cannot be used. This parameter can be specified only in Cs contour axis control mode, rigid tapping mode, and spindle positioning mode, but note that the detection unit per pulse differs.)

Alarm and message

Number	Message	Contents
194	SPINDLE COMMAND IN SYNCHRO–MODE	The Cs contour axis control mode, spindle positioning mode, rigid tapping mode, or simple spindle synchronous control is specified in spindle synchronous control mode. Modify the program to cancel spindle synchronous control mode beforehand. Alternatively, spindle synchronous control can be specified in simple spindle synchronous control mode. Modify the program to cancel simple spindle synchronous control mode before specifying spindle synchronous control.

Diagnosis screen

- **Positional deviation display while spindle synchronous control is applied**

414	Master spindle motion error while spindle synchronous control or simple spindle synchronous control is applied
415	Slave spindle motion error while spindle synchronous control or simple spindle synchronous control is applied
416	Absolute value of synchronization error while spindle synchronous control or simple spindle synchronous control is applied

The display units for diagnostic data 414 through 416 are pulses; in spindle synchronous control mode, one pulse corresponds to an error of 360/4096 (degrees).

In simple spindle synchronous control mode, the detection unit per pulse depends on whether Cs contour axis control mode, rigid tapping mode, or spindle positioning mode is set, and also on the type and mounting of the detector.

10 TOOL FUNCTIONS

10.1 TOOL FUNCTION

General

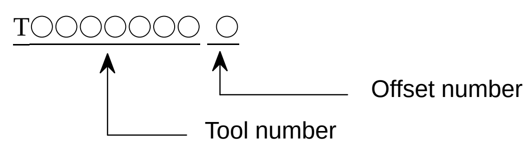
(M series)

Selection of tools can be done by commanding tool numbers with up to an 8-digit numeral after address T.

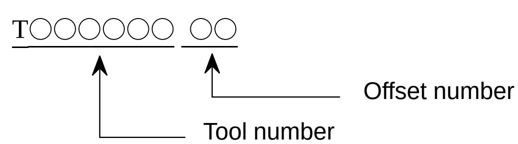
(T series)

Selection of tools and offset amounts can be done by commanding tool numbers and offset numbers with up to an 8-digit numeral after address T. The offset number is specified with the last one or two digits of the T code. The tool number is specified with the remaining digits after excluding the one or two digits used to specify the offset number.

When the last one digit is used to specify the offset number:
(Parameter LD1 (No. 5002#0)=1)



When the last two digits are used to specify the offset number:
(Parameter LD1 (No. 5002#0)=0)



When a T code is specified, the code signal and strobe signal corresponding to the specified tool number are issued. The machine selects a tool according to the issued signals. The code signal is held until another T code is specified.

In a block, no more than one T code can be specified. The maximum number of digits that can follow T can be specified in parameter 3032. If this number is exceeded, an alarm occurs.

Signal

See Section 8.1.

Parameter

3032	Allowable number of digits for the T code

[Data type] Byte

[Valid data range] 1 to 8

Set the allowable numbers of digits for the T code.

	#7	#6	#5	#4	#3	#2	#1	#0
5002							LGN	LD1

[Data type] Bit

LD1 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

LGN 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
5006							TGC	

[Data type] Bit

TGC When a T code is specified in a block containing G50, G04, or G10:

0 : No alarm occurs.
1 : P/S alarm No. 254 occurs.

Alarm and message

Number	Message	Description
030	ILLEGAL OFFSET NUMBER (T series)	The offset number in T function specified for tool offset is too large. Modify the program.
043	ILLEGAL T–CODE COMMAND (M series)	In a system using the DRILL–MATE with an ATC, a T code was not specified together with the M06 code in a block. Alternatively, the T code was out of range.
245	T–CODE NOT ALOWED IN THIS BLOCK (T series)	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.

Note

NOTE

When a move command and a tool function are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and tool function commands.

(ii) Executing tool function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the sequence program of PMC.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.10.1	TOOL SELECTION FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.10.1	TOOL SELECTION FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.10.1	TOOL SELECTION FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.10.1	TOOL SELECTION FUNCTION
	CONNECTION MANUAL (This manual)	8	AUXILIARY FUNCTION

10.2 TOOL COMPENSATION VALUE/ TOOL COMPENSATION NUMBER/ TOOL COMPENSATION MEMORY

General

(M series)

Tool compensation values include tool geometry compensation values and tool wear compensation values (Fig. 10.2 (a)).
The geometry compensation and wear compensation can be unified to the tool compensation.

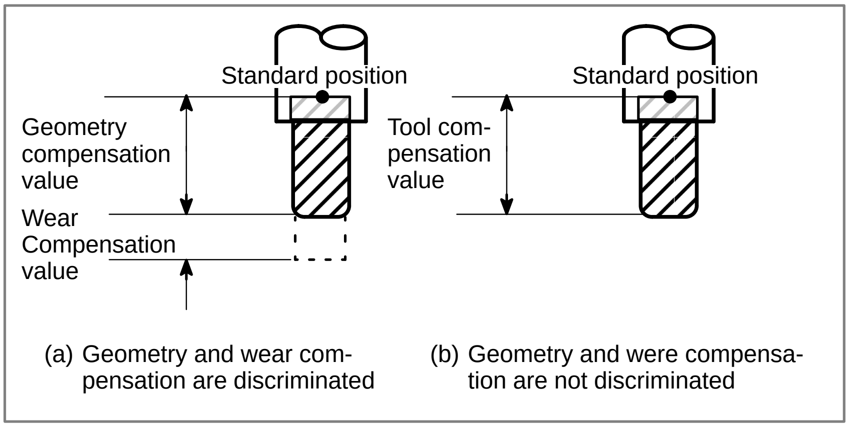


Fig. 10.2(a) Geometric compensation and wear compensation

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

- **Tool compensation number**

The memory can hold 32, 64, 99, 200, 400, 499 or 999 sets of tool compensation values.

- **Tool compensation memory**

One of the tool compensation memory A/B/C can be selected according to the configuration of offset amount.

(1) Tool compensation memory A

There is no difference between geometry compensation memory and wear compensation memory in 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).

(2) 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).

(3) Tool compensation memory C

Memory for geometry compensation and 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).

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

(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)).

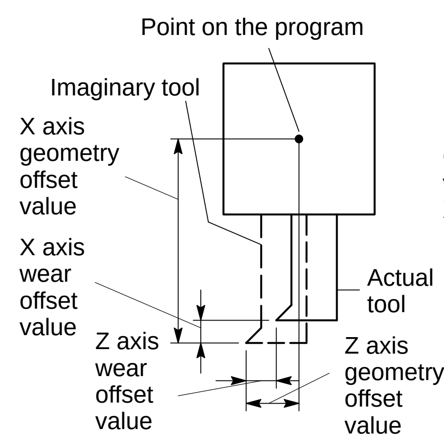


Fig. 10.2 (b) Difference the tool geometry offset from tool wear offset

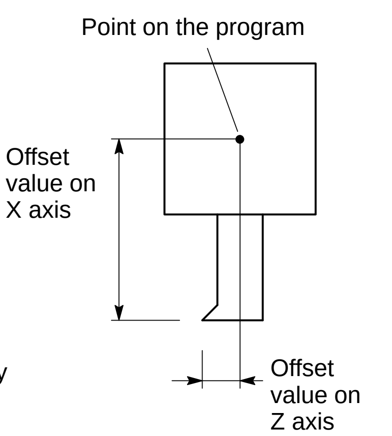


Fig.10.2 (c) Not difference the tool geometry offset from tool wear offset

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

Range of tool compensation value can be expanded by option.

Increment system	Tool compensation value (geometry compensation, wear compensation)	
	Metric input	Inch input
IS–B	±9999.999 mm	±999.9999 inch
IS–C	±9999.9999 mm (±4000.0000 mm)	±999.99999 inch (±160.00000 inch)

NOTE
1 When parameter OIM (No. 5006#0)=1, the range in parenthesis is available.
2 The above table does not apply to the B–axis offset in B–axis control function.

- **Tool compensation number**

The memory can hold 16, 32, 64 or 99 sets of tool compensation values.

- **Tool compensation memory**

There are two types of tool offset amount memory, which can be selected according to the configuration of 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 follows:

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
3109							DWT	

[Data type] Bit

DWT Characters G and W in the display of tool wear/geometry compensation amount

0 : The characters are displayed at the left of each number.
1 : The characters are not displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3290							GOF	WOF

[Data type] Bit

WOF Setting the tool wear compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

GOF Setting the tool geometry compensation value by MDI key input is:

0 : Not disabled
1 : Disabled (With parameter No. 3294 and No. 3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Data type] Word

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No. 3290 and bit 1 (GOF) of parameter No. 3290, parameter Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No. 3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No. 3295, set the number of such values.

Example:

The following setting disables the modification of both the tool geometry compensation values and tool wear compensation values corresponding to offset numbers 100 to 110:

Bit 1 (GOF) of parameter No. 3290 = 1 (Disables tool geometry compensation value modification.)

Bit 0 (WOF) of parameter No. 3290 = 1 (Disables tool wear compensation value modification.)

Parameter No. 3294 = 100

Parameter No. 3295 = 11

If bit 0 (WOF) of parameter No. 3290 is set to 0, the modification of the tool geometry compensation values alone is disabled. The tool wear compensation values may be modified.

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP						LGN	

[Data type] Bit

LGN Geometry offset number of tool offset (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

WNP Imaginary tool tip direction used for tool nose radius compensation, when the geometry/wear compensation option is selected, is the direction specified by:

0 : Geometry offset number
1 : Wear offset number

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

[Data type] Bit

ORC Tool offset value

0 : Set by the diameter specification (Can be set in only the axis under diameter programming)
1 : Set by the radius specification

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:

0 : Not performed
1 : Performed

5013	Maximum value of tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation. The following alarm or warning will be informed when the tool wear compensation (absolute value) exceeding this setting value is set.

Input from MDI	Too many digits
Input by G10	P/S 32 offset value is out of range by G10

5014	Maximum value of incremental input for tool wear compensation

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range]

Increment system	IS–A	IS–B	IS–C
Metric input	0 to 99999	0 to 999999	0 to 9999999
Inch input	0 to 99999	0 to 999999	0 to 9999999

This parameter sets the maximum value of tool wear compensation at an incremental input. If the incremental value exceeds the set value, the following alarm or warning message is indicated:

Input from MDI	Data is out of range
Input by G10	P/S 32 offset value is out of range by G10

Alarm and message

Number	Message	Description
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.

Warning message	Content
DATA IS OUT OF RANGE	The value searched exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.

Note

NOTE
In the two–path control, the number of specified tool compensation values equals the number of tool compensations for each tool post.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II 14.8	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II 14.5	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II 14.6	TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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 <F064#0>

[Classification] Output signal

[Function] Reports the end of the life of the last tool of a group.

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

· The life of the last tool of a group ends, after tool change has been performed each time the end of the life of each tool in a group is detected.

The signal is set to 0 when:

· Tool–change reset is completed for all groups in which no available tools remain.

NOTE
The TLCH signal turns to “1” when the CNC is reset by M02 or M30, for instance after the tool life, based on the frequency of times used, is reached. When tool life is specified by usage time, TLCH turns to “1” when the tool life limit is reached. The signal will change during machine operation, but machining will continue until the end of the program.

Tool change reset signal TLRST <G048#7>

[Classification] Input signal

[Function] Clears all executable data, including the life count of the group, *, and @.

To clear the data, specify a group number by tool group number selection signal after replacing the worn–out tools that are displayed on the CRT. The data can also be cleared from the MDI.

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

· Clears all executable data, including the life count of the group.

If the same group is specified after machining is resumed, the first tool in the group is selected.

> **NOTE**
> Tool change reset signal TLRST is valid only when the automatic operating signal OP is “0”.

Individual tool change signal TLCHI <F064#2> (M series)

[Classification] Output signal

[Function] Reports the end of the life of the current tool. The following processing can be programmed: A running program is interrupted by a tool-change program when the signal turns to “1”. Execution of the interrupted program is resumed when the tool is changed.

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

· The end of the life of the current tool is detected.

The signal is set to “0” when:

· Individual tool-change reset is executed.

Individual tool change reset signal TLRSTI <G048#6> (M series)

[Classification] Input signal

[Function] Sets the individual tool change signal TLCHI to “0”.

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

· Sets the individual tool change signal to “0”.

> **NOTE**
> 1 These signals are valid only when tool life management is performed on the basis of the tool life calculated in terms of time or cutting length.
> 2 Individual tool change signal TLCHI is not cleared by reset.

Tool skip signal TLSKP <G048#5>

[Classification] Input signal

[Function] A tool which has not reached its lifespan may be changed by one of two methods:

(i) Designate the group number for the tool by tool group number selection signal then turn the tool skip signal TLSKP to “1”. The next T-code command will pass over the current tool in the group for which the skip was designated, and select the next tool.

(ii) Turn the TLSKP signal to “1” without designating a group number, and the machine will skip to the next tool in the group currently in use.

Either of these methods is set using parameter SIG (No. 6800#3). Tool life is counted from zero. When the TLSKP signal is “1” and the last tool in the group is being used, the TLCH signal turns to “1”.

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

· Selects the next tool in the group for which a skip is specified with the next T code.

· Assumes the number of the group to which the current tool belongs.

CAUTION
The cycle start lamp signal (STL) and feed hold lamp signal (SPL) must both be “0” before inputting the TLSKP signal.

New tool select signal TLNW <F064#1>

[Classification] Output signal

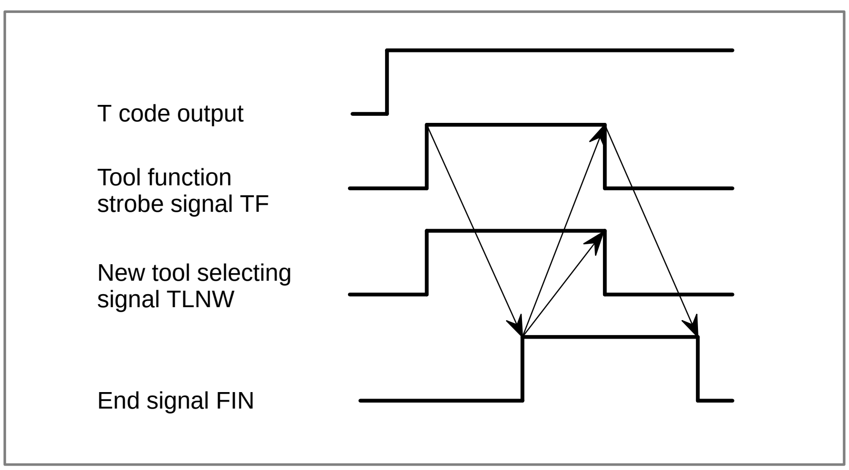
[Function] Reports that a new tool of a certain group is selected.
This signal can be used when, for example, a compensation value is to be measured automatically when a new tool is selected.
The new tool select signal is issued at the same timing as TF (tool function strobe signal).

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

· A new tool of a certain group is selected.

The signal is set to “0” when:

· The completion signal is set to “1”.



Tool group number select signal TL01 to TL256 (M series) <G047#0 to G048#0> TL01 to TL 64 <G47#0 to #6> (T series)

[Classification] Input signal

[Function] When the TLRST or TLSKP signals are input, the tool group number must be given in advance, using the tool group number selection signals TL01 to TL64 (T series) or TL01 to TL128 (M series).
Command the following value in binary form:
Tool group number to be specified –1

[Operation] A specified tool group is selected.

Tool life count override signal *TLV0 to *TLV9 <G049#0 to G050#1> (M series)

[Classification] Input signal

[Function] Overrides the life count (time) if parameter LFV (No. 6801#2) is specified.

Each of the ten binary code signals has a unique override value that becomes valid when the signal is set to “0”. The life count is overridden by the sum of the valid override values. The override value can be specified in steps of 0.1, within the range of 0 to 99.9.

$$\text{Override value} = \sum_{i=0}^{9} \{2^i \times Vi\}$$

*TLV0	× 0.1
*TLV1	× 0.2
*TLV2	× 0.4
*TLV3	× 0.8
*TLV4	× 1.6
*TLV5	× 3.2
*TLV6	× 6.4
*TLV7	×12.8
*TLV8	×25.6
*TLV9	×51.2

(Example) When *TLV7, *TLV6, and *TLV3 are set to “0”, the override value is calculated as follows:

12.8 + 6.4 + 0.8 = 20.0

The life count is multiplied by 20.0.

[Operation] The actual cutting time is counted and multiplied by the override value obtained by the signals. The calculated time is used as the basis for tool-life management.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0
G050							*TLV9	*TLV8

	#7	#6	#5	#4	#3	#2	#1	#0
F064						TLCHI	TLNW	TLCH

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6800			SNG	GRS	SIG	LTM	GS2	GS1
	M6T	IGI	SNG	GRS	SIG	LTM	GS2	GS1

[Data type] Bit

GS1, GS2 This parameter sets the combination of the number of tool life groups which can be entered, and the number of tools which can be entered per group as shown in the table below.

GS2	GS1	M series		T series	
		Group count	Tool count	Group count	Tool count
0	0	1– 16 1– 64	1– 16 1– 32	1– 16 1– 16	1– 16 1– 32
0	1	1– 32 1–128	1– 8 1– 16	1– 32 1– 32	1– 8 1– 16
1	0	1– 64 1–256	1– 4 1– 8	1– 64 1– 64	1– 4 1– 8
1	1	1–128 1–512	1– 2 1– 4	1– 16 1–128	1– 16 1– 4

Lower side ranges in M series column are for the tool life management of 512 pairs.

Lower side range in T series column are for tool life management of 128 pairs.

LTM Tool life

0 : Specified by the number of times
1 : Specified by time

SIG Group number is

0 : Not input using the tool group number selection signal during tool skip (The current group is specified.)
1 : Input using the tool group signal during tool skip

GRS Tool change reset signal

0 : Clears only the execution data of a specified group
1 : Clears the execution data of all entered groups

SNG Input of the tool skip signal when a tool that is not considered tool life management is selected.

0 : Skips the tool of the group used last or of the specified group (using SIG, #3 of parameter No. 6800).
1 : Ignores a tool skip signal

IGI Tool back number

0 : Not ignored
1 : Ignored

M6T T code in the same block as M06

0 : Judged as a back number
1 : Judged as a next tool group command

	#7	#6	#5	#4	#3	#2	#1	#0
6801		EXG					TSM	
	M6E	EXT			EMD	LFV		CUT

[Data type] Bit

CUT The tool life management using cutting distance is

0 : Not performed (Usually set this parameter to 0).
1 : Performed

TSM When a tool takes several tool numbers, life is counted in tool life management:

0 : For each of the same tool numbers.
1 : For each tool.

LFV Specifies whether life count override is enabled or disabled when the extended tool life management function is used.

0 : Disabled
1 : Enabled

EMD An asterisk (*) indicating that a tool has been expired is displayed,

0 : When the next tool is selected
1 : When the tool life is expired

EXG Tool life management data registration by G10 (T system) is:

0 : Performed after the data for all tool groups has been cleared.
1 : Performed by adding/changing or deleting the data for a specified group.

NOTE

When EXG = 1, address P in the block including G10 can be used to specify whether data is to be added/changed or deleted (P1: add/change, P2: delete). When P is not specified, the data for all tool groups is cleared before the tool life management data is registered.

EXT Specifies whether the extended tool life management function is used.

0 : Not used
1 : Used

M6E When a T code is specified in the same block as M06

0 : The T code is processed as a return number or as a group number selected next. Either is set by parameter M6T No. 6800#7.

1 : The tool group life is counted immediately.

6810	
	Tool life management ignored number

[Data type] Word

[Valid data range] 0 to 9999

This parameter sets the tool life management ignored number.

When the set value is subtracted from a T code, a remainder is used as the tool group number of tool life management when a value exceeding the set value is specified in the T code.

6811	Tool life count restart M code

[Data type] Byte

[Valid data range] 0 to 255 (not including 01, 02, 30, 98, and 99)
When zero is specified, it is ignored.

When the life is specified by the number of times, the tool exchange signal is output when a tool life count restart M code is specified if tool life of at least one tool group is expired. A tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified. A tool life counter is then incremented by one.

When the life is specified by time, a tool in life is selected in the specified group when a T code command (tool group command) is specified after the tool life count restart M code is specified.

Alarm and message

Number	Message	Description
149	FORMAT ERROR IN G10L3	A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T–CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP (M series)	When the group is not commanded, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T–CODE IN M06 (M series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
	ILLEGAL T–CODE IN M06 (T series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.10.2	Tool Life Management Function
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.10.2	Tool Life Management Function
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.10.2	Tool Life Management Function
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.10.2	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.

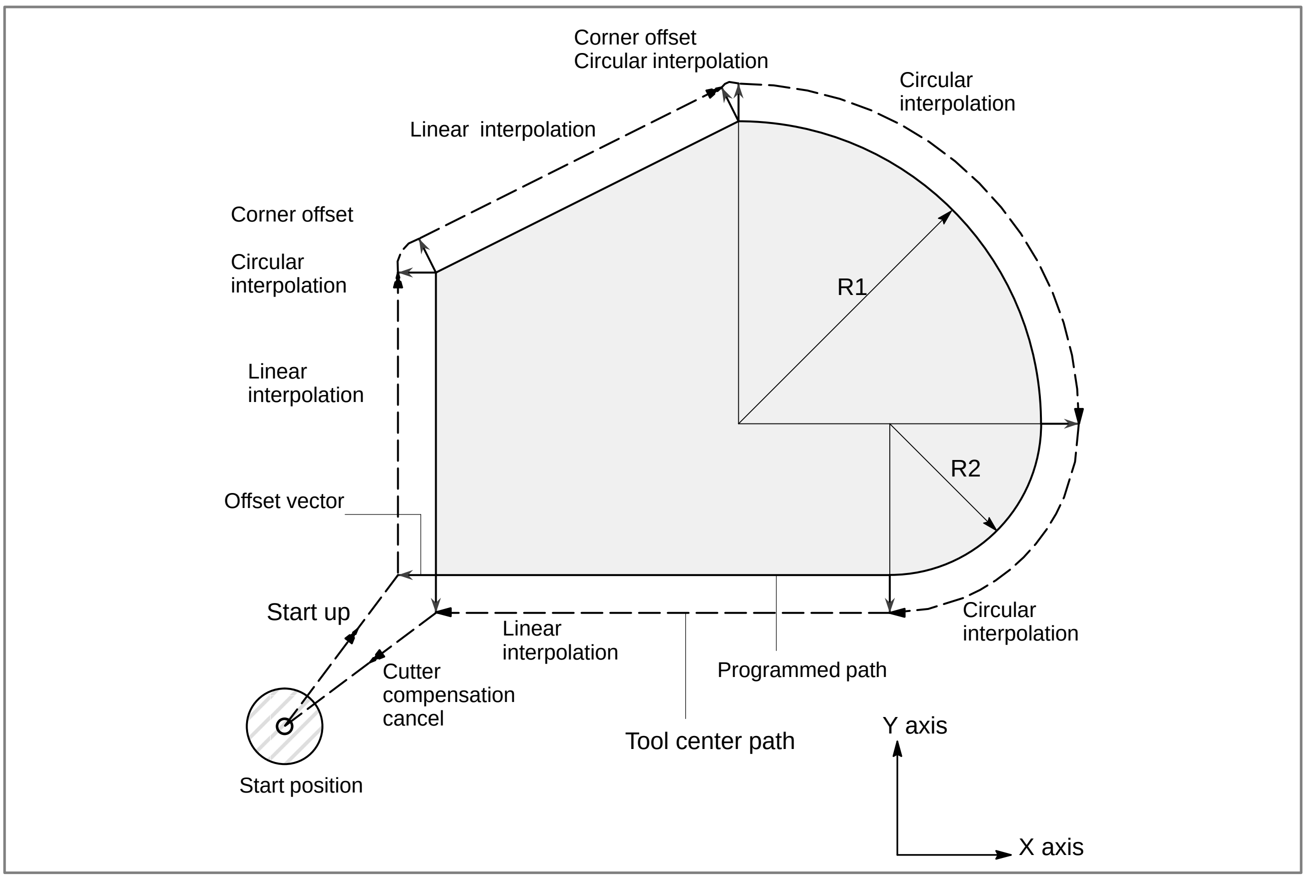


Fig. 10.4 (a) Outline of cutter compensation B

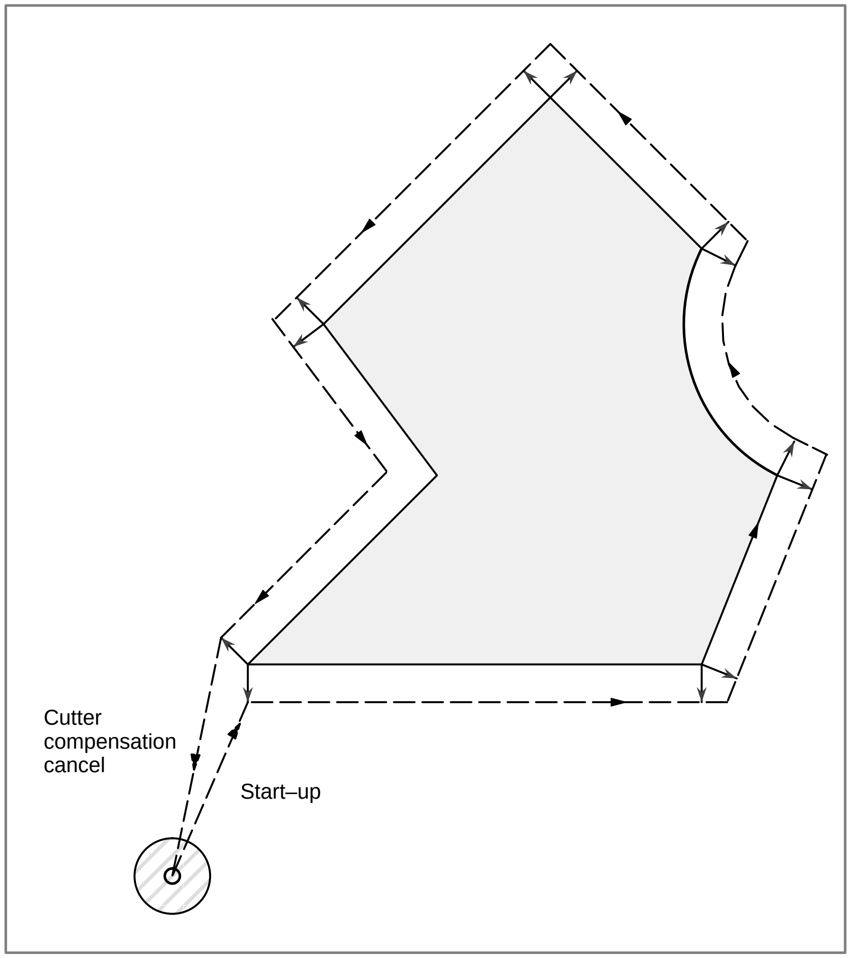


Fig. 10.4 (b) Outline of cutter compensation C

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5001						OFH		

[Data type] Bit

OFH Offset number of tool length compensation, cutter compensation and tool offset

0 : Specifies the tool length compensation using an H code, and cutter compensation C using a D code
Tool offset conforms to TPH in parameter TPH (No. 5001#5).

1 : Specifies the tool length compensation, cutter compensation and tool offset using H codes

	#7	#6	#5	#4	#3	#2	#1	#0
5003						CCN	SUV	SUP

[Data type] Bit

SUP Start up or cancel in cutter compensation C

0 : Type A
1 : Type B

SUV When G40, G41, and G42 are specified independently,

0 : The start up and cancel operation conforms to the standard specification.

1 : Moves by a distance corresponding to the offset vector which is vertical to the next block movement.

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.

1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during cutter compensation C

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of the corner during cutter compensation C.

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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.14.4	Cutter compensation B
		II.14.5	Cutter compensation C
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.14.4	Cutter compensation C

10.4.2 Tool Nose Radius Compensation (T Series)

General

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

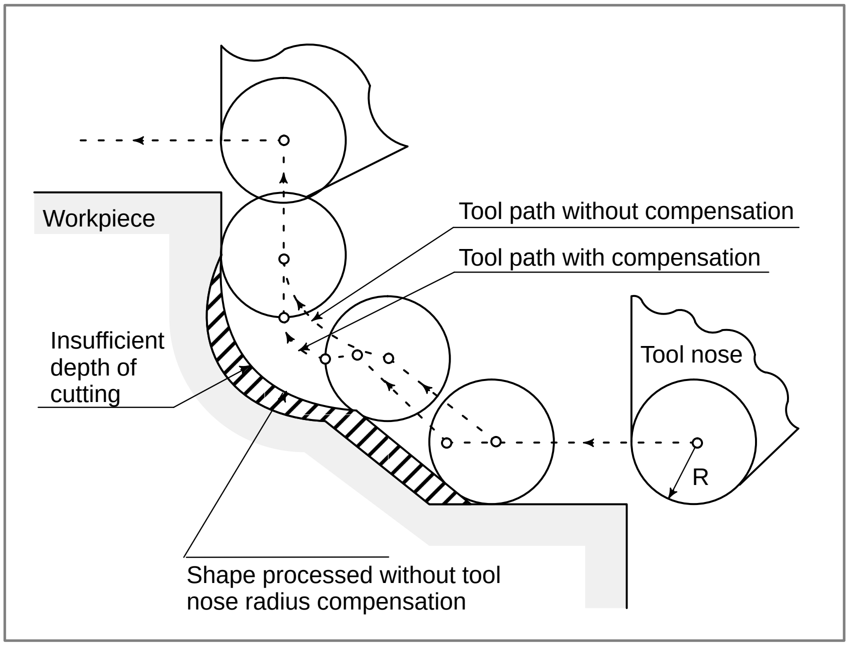


Fig. 10.4.2 Tool path of tool nose radius compensation

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5002	WNP							

[Data type] Bit

WNP Imaginary tool tip direction used for tool nose radius compensation, 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
5003						CCN		

[Data type] Bit

CCN When automatic reference position return (G28) is specified in the cutter compensation C mode (M series) or in tool nose radius compensation (T series):

0 : The cutter compensation vector is cancelled in movement to an intermediate position.

1 : The cutter compensation vector is not cancelled in movement to an intermediate position, but is cancelled in movement to the reference position.

5010	Limit value that ignores the vector when a tool moves on the outside of a corner during tool nose radius compensation

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 16383

This parameter sets the limit value that ignores a slight movement occurring when a tool moves on the outside of a corner during tool nose radius compensation.

Alarm and message

Number	Message	Description
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.14.2	Tool Nose Radius Compensation
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.14.2	Tool Nose Radius Compensation

11 PROGRAM COMMAND

11.1 DECIMAL POINT PROGRAMMING/ POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING

General

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, F .... M series
X, Y, Z, U, V, W, A, B, C, I, J, K, R, F. .... T series

There are two types of decimal point notation: calculator-type notation and standard notation.
When calculator-type decimal point notation is used, a value without decimal point is considered to be specified in millimeters, inches or degree. When standard decimal point notation is used, such a value is considered to be specified in least input increments. Select either calculator-type or standard decimal point notation by using the DPI bit (bit 0 of parameter 3401). Values can be specified both with and without decimal point in a single program.

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401								DPI

[Data type] Bit

DPI When a decimal point is omitted in an address that can include a decimal point

0 : The least input increment is assumed.
1 : The unit of mm, inches, degree, or s is assumed. (Pocket calculator type decimal point programming)

Alarm and message

Number	Message	Description
007	ILLEGAL USE OF DECIMAL POINT	Decimal point “ · ” input error (A decimal point was input after an address with which it can not be used. Or multiple decimal points were input.) Modify the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.8.4	Decimal point programming
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.8.3	Decimal point programming
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.8.4	Decimal point programming
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.8.3	Decimal point programming

11.2 G CODE SYSTEM (T SERIES)

General

There are three G code systems : A,B, and C (Table 11.2). Select a G code system using parameter GSC (No. 3401#7) and parameter GSB (No. 3401#6).

Table 11.2 G code list (1/3)

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or Helical interpolation CW
G03	G03	G03		Circular interpolation CW or Helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High speed cycle cutting
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G10	G10	G10		Data setting
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Reference position return
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point 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

Table 11.2 G code list (2/3)

G code A	G code B	G code C	Group	Function
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
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 cut mode
G69	G69	G69	04	Mirror image for double turrets OFF or balance cut 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
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscillation grinding cycle (for grinding machine)
G74	G74	G75		Oscillation direct constant–dimension grinding cycle (for grinding machine)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring

Table 11.2 G code list (3/3)

G code A	G code B	G code C	Group	Function
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		End face 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
—	G99	G99		Return to R point level

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3401	GSC	GSB						

[Data type] Bit

GSB, GSC The G code system is set.

GSC	GSB	G code
0	0	G code system A
0	1	G code system B
1	0	G code system C

	#7	#6	#5	#4	#3	#2	#1	#0
3402		CLR			G91			G01

[Data type] Bit

G01 Mode entered when the power is turned on or when the control is cleared

0 : G00 mode (positioning)
1 : G01 mode (linear interpolation)

G91 When the power is turned on or when the control is cleared

0 : G90 mode (absolute command)
1 : G91 mode (incremental command)

CLR Reset button on the CRT/MDI panel, external reset signal, reset and rewind signal and emergency stop signal

0 : Cause reset state.
1 : Cause clear state.

For the reset and clear states, refer to APPENDIX E of operator's manual.

Alarm and message

Number	Message	Description
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.

Note

NOTE

1 If the CNC enters the clear state (see bit 6 (CLR) of parameter 3402) when the power is turned on or the CNC is reset, the modal G codes change as follows.
  (1) G codes marked with in Table 11.2 are enabled.
  (2) When the system is cleared due to power-on or reset, whichever specified, either G20 or G21, remains effective.
  (3) Bit 7 (G23) of parameter No. 3402 is used to specify whether G22 or G23 is to be selected upon power–on. The selection of G22 or G23 is not, however, changed when the CNC is cleared upon a reset.
  When the system is cleared due to reset, whichever specified, either G22 or G23, remains effective.
  (4) Setting bit 0 (G01) of parameter 3402 determines which code, either G00 or G01, is effective.
  (5) Setting bit 3 (G91) of parameter 3402 determines which code, either G90 or G91, is effective.

2 G codes of group 00 except G10 and G11 are single-shot G codes.

3 Alarm 010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.

4 G codes of different groups can be specified in the same block.
  If G codes of the same group are specified in the same block, the G code specified last is valid.

5 If a G code of group 01 is specified in a canned cycle, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.

6 When G code system A is used for a canned cycle, only the initial level is provided at the return point.

7 G codes are displayed for each group number.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.3	PREPARATORY FUNCTION (G FUNCTION)
		APPENDIX E	STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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

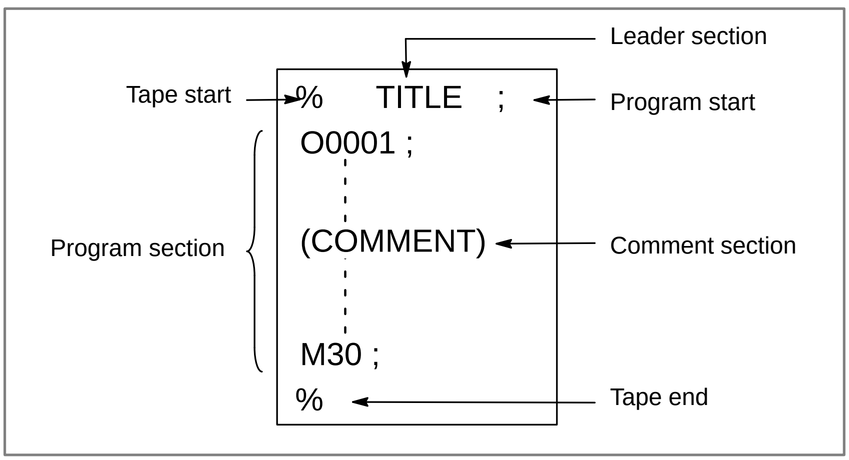


Fig. 11.3 Program configuration

Parameter

0100	#7	#6	#5	#4	#3	#2	#1	#0
							CTV	

Setting entry is acceptable.

[Data type] Bit

CTV: Character counting for TV check in the comment section of a program.

0 : Not performed
1 : Performed

3201	#7	#6	#5	#4	#3	#2	#1	#0
		NPE	N99					

[Data type] Bit

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:

0 : Completed
1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:

0 : Completed
1 : Not completed

3404	#7	#6	#5	#4	#3	#2	#1	#0
		EOR				SBP		

[Data type] Bit

SBP Address P of the block including M198 in the subprogram call function

0 : Indicating a file number
1 : Indicating a program number

EOR When the end-of-record mark (%) is read during program execution:

0 : P/.S alarm No. 5010 occurs.
(Automatic operation is stopped, and the system enters the alarm state.)
1 : No alarm occurs.
(Automatic operation is stopped, and the system is reset.)

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

When the subprogram call function is used, this parameter sets the M code for calling a program in a file stored on the external input/output device.

NOTE
The M code is judged to be M198 when zero is specified as the setting value.

Alarm and message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input).
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective (when TVC, bit 0 of setting parameter 0000 is set to 1).
5010	END OF RECORD	The end of record (%) was specified.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.12	Program Configuration
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.12	Program Configuration
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.12	Program Configuration
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.12	Program Configuration

11.4 INCH/METRIC CONVERSION

General

Either inch or metric input can be selected by G code.

Signal

Inch input signal INCH<F002#0>

[Classification] Output signal

[Function] This signal indicates that inch input mode is in progress.

[Output condition] “1”indicates that the inch input mode (G20) is in progress, and “0” indicates that metric input mode (G21) is in progress.

This signal changes to the corresponding state when modes are switched using the setting data display on the MDI panel.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F002								INCH

Parameter

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

Setting entry is acceptable.

[Data type] Bit

INI Unit of input

0: In mm
1: In inches

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

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

[Data type] Bit

INM Least command increment on the linear axis

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

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

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

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

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

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

[Data type] Bit

ZPI Coordinates at the reference position when a coordinate system is set automatically

0 : Value set in parameter No. 1250 is used.

1 : For input in mm, the value set in parameter 1250 is used, or for input in inches, the value set in parameter No. 1251 is used.

1250	Coordinate value of the reference position used when automatic coordinate system setting is performed

[Data type] Two–word axis

[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.

1251	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 axis

[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 ZPI in parameter 1201#1 is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
1403								MIF

[Data type] Bit

MIF Cutting feedrates at feed per minute is specified by F commands

0 : In units of 1 mm/min for millimeter machines or 0.01 inches/min for inch machines.

1 : In unit of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

NOTE
M series are not equipped with this parameter. Cutting feedrates are specified by F commands in units of 0.001 mm/min for millimeter machines or 0.00001 inches/min for inch machines.

	#7	#6	#5	#4	#3	#2	#1	#0
3104								MCN

[Data type] Bit

MCN Machine position is:

0 : Not displayed according to the unit of input.
(Regardless of whether input is made in mm or inches, the machine position is displayed in mm for millimeter machines, or in inches for inch machines.)

1 : Displayed according to the unit of input.
(When input is made in mm, the machine position is displayed in mm, and when input is made in inches, the machine position is displayed in inches accordingly.)

	#7	#6	#5	#4	#3	#2	#1	#0
3405								AUX

[Data type] Bit

AUX The least increment of the command of the second miscellaneous function specified with a decimal point

0 : Assumed to be 0.001

1 : Depending on the input increment.
(For input in mm, 0.001 is assumed, or for input in inches, 0.0001 is assumed.)

	#7	#6	#5	#4	#3	#2	#1	#0
5006								OIM

[Data type] Bit

OIM When the unit is switched between the inch and metric systems, automatic tool compensation value conversion is:

0 : Not performed

1 : Performed

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

[Data type] Bit

PIM When only the axes controlled by the PMC are used, the linear axis is:

0 : Influenced by inch/millimeter input.

1 : Not influenced by inch/millimeter input.

Warning

WARNING

When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re–set according to the least input increment.

However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re–set.

Note

NOTE

1 When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
2 Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.
3 The inch and metric input can also be switched using settings.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.8.3	Inch/metric conversion (G20, G21)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.8.2	Inch/metric conversion (G20, G21)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.8.3	Inch/metric conversion (G20, G21)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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).

> **NOTE**
> This function cannot be used for two-path control.

• 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

Six axes maximum can be controlled. Six axes can be controlled simultaneously.

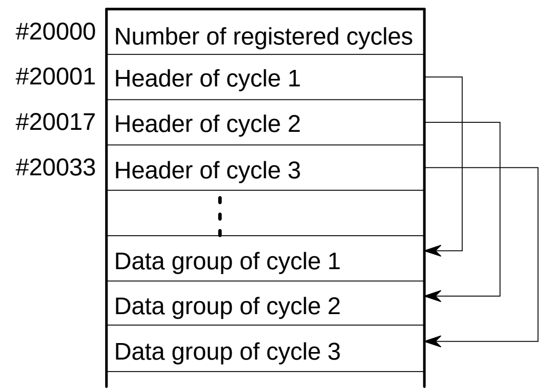
• Pulse distribution

Set the number of pulses per cycle in parameter 7501#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
–	–	r6	r5	r4	r3	r2	r1	–	–	t6	t5	t4	t3	t2	t1

The bits from t1 to t6, 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 r6, corresponding to the 1st to 6th 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 6th axes**

·**Constant data**
When the corresponding data type bit (t6 to t1) is 0, specify “distribution data value”.

·**Variable data**
When the corresponding data type bit (t6 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 (t6 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 25000 as the data assignment variable.

- **Total number of fixed data items for the first to 6th 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 (t6 to t1) is 1, the data is regarded as a variable data.

Parameter

7501	#7	#6	#5	#4	#3	#2	#1	#0
	IPC	IT2	IT1	IT0				CSP
	IPC	IT2	IT1	IT0				

[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

IPC 0 : The system does not monitor whether a distribution process is stopped while high-speed cutting (G05) is performed with high-speed remote buffer A or B or in a high-speed cycle cutting.

1 : The system monitors whether a distribution process is stopped while high-speed machining (G05) is performed with high-speed remote buffer A or B or in a high-speed cycle cutting.
(Alarms 179 and 000 are simultaneously issued if the distribution process is stopped. In this case, the power must be turned off then on again.)

NOTE
The distribution process stops, when the host cannot send data with the high-speed remote buffer by the specified time.

7505	#7	#6	#5	#4	#3	#2	#1	#0
							HUNx	HSCx
								HSCx

NOTE
After setting this parameter, the power must be turned off then on again.

[Data type] Bit axis

HSCx Specifies whether each axis is used for high-speed distribution in a high-speed cycle cutting or with a high-speed remote buffer.

0 : Not used for high-speed distribution
1 : Used for high-speed distribution

HUNx Specifies whether the unit of data to be distributed during cutting in a high-speed cycle is ten times the least input increment.

0 : The unit of data is the same as the least input increment.
1 : The unit of data is ten times the least input increment.

NOTE
This parameter is used when a data item to be distributed exceeds a word in terms of the least input increment or the maximum travel speed.
CNC distributes ten times the value for cutting in a high–speed cycle for the axes in which HUNx of this parameter is set to 1. Therefore, set a value one tenth the value to be distributed for cutting in a high-speed cycle along the specified axes.

7510	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 6
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.

Number	Message	Description
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 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

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

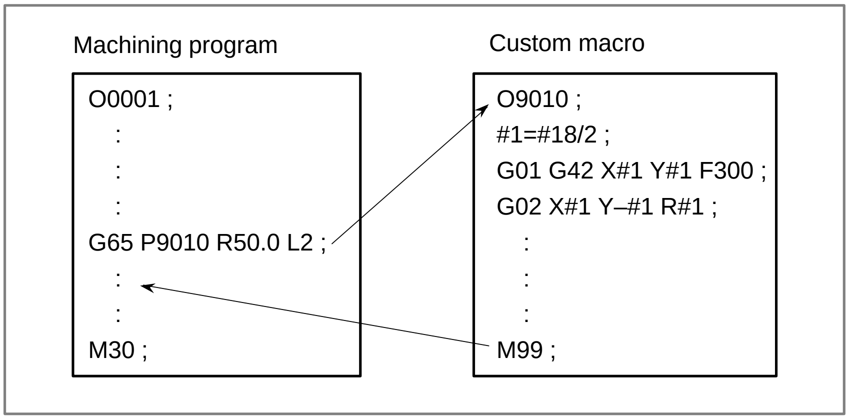
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.19.1	High speed cycle cutting
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.18.1	High speed cycle cutting

11.6 CUSTOM MACRO

11.6.1 Custom Macro

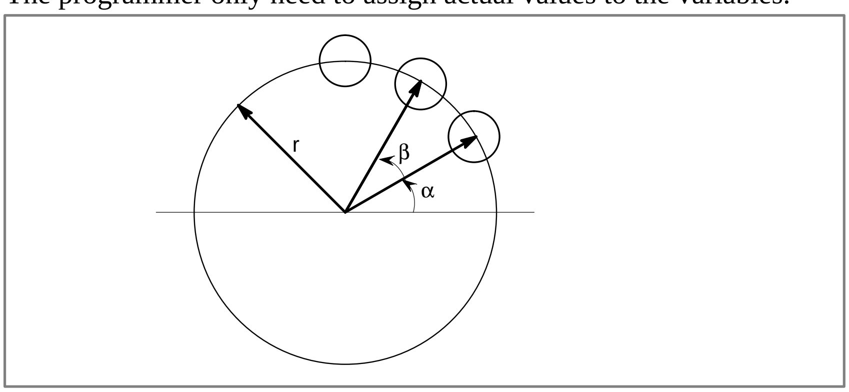
General

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

Signal

Custom Macro Input Signal UI000 to UI015 <G054, G055>

[Classification] Input signal

[Function] No function is provided for the control unit. These signals can be read by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UI000 UI001 UI002 UI003 : UI014 UI015	1 1 1 1 : 1 1	#1000 #1001 #1002 #1003 : #1014 #1015	"0" at "0" and "1" at "1"
UI000 to UI015	16	#1032	16–bit binary code *1

*1 Variable value $\#1032 = \sum_{i=0}^{15} \{\#[1000 + i] \times 2^i\}$

Custom Macro Output Signal UI000 to UI015 <F054, F055> UO100 to UO131 <F056 to F059>

[Classification] Output signal

[Function] No function is provided for the control unit. These signals can be read or written by a custom macro as a type of system variable, and are used for interface signals between custom macros and the PMC.
These signals correspond to system variables as indicated below.

Signals	Q'ty	Variables	Correspondence of values
UO000 UO001 UO002 UO003 : UO014 UO015	1 1 1 1 : 1 1	#1100 #1101 #1102 #1103 : #1114 #1115	"0" at "0" and "1" at "1"
UO000AUO015 UO100AUO115	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 H0Iand Vi=1 when UO1i is H1I

These system variables can be used on the left side of an assignment statement as well as on the right side.
The value assigned to the system variable used on the left side last is used for the value of the system variable to be assigned on the right side.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008

	#7	#6	#5	#4	#3	#2	#1	#0
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124

Parameter

- **Setting for single block stop**

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					

[Data type] Bit

SBM Custom macro statement

0 : Not stop the single block
1 : Stops the single block

- **Other settings**

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV	TCS	CRO	PV5		PRT	

[Data type] Bit

PRT Reading zero when data is output using a DPRNT command

0 : Outputs a space
1 : Outputs no data

PV5 Custom macro common variables

0 : Outputs custom macro common variables #500 through #599.
1 : Outputs custom macro common variables #100 through #199 and #500 through #599.

CRO ISO code output using a BPRNT command or a DPRNT command

0 : Outputs only LF after data is output
1 : Outputs LF and CR after data is output

TCS Custom macro (subprogram)

0 : Not called using a T code
1 : Called using a T code

CCV Custom macro’s common variables Nos. 100 through 149

0 : Cleared to “vacant” by reset
1 : Not cleared by reset

CLV Custom macro’s local variables #1 through #33

0 : Cleared to “vacant” by reset
1 : Not cleared by reset

- **Setting when macro statement is input/output with EIA code**

	#7	#6	#5	#4	#3	#2	#1	#0
6010	*7	*6	*5	*4	*3	*2	*1	*0
6011	=7	=6	=5	=4	=3	=2	=1	=0
6012	#7	#6	#5	#4	#3	#2	#1	#0
6013	[7	[6	[5	[4	[3	[2	[1	[0
6014	]7	]6	]5	]4	]3	]2	]1	]0

[Data type] Bit

These parameters are used to input/output macro statements with EIA code.

The numeral of a suffix indicates the bit position in a code.

***0 to *7** Set the hole pattern of an EIA code indicating *.

=0 to =7 Set the hole pattern of an EIA code indicating =.

#0 to#7 Set the hole pattern of an EIA code indicating #.

[0 to [7 Set the hole pattern of an EIA code indicating [.

]0 to]7 Set the hole pattern of an EIA code indicating].

0 :Corresponding bit is 0

1 :Corresponding bit is 1.

- **Setting an M code that calls a program entered in a file**

6030	M code that calls the program entered in file

[Data type] Byte

[Valid data range] 0, and 1 to 255

When the subprogram call function is used, this parameter sets the M code for calling a program in a file stored on the external input/output device.

> **NOTE**
> The M code is judged to be M198 when zero is specified as the setting value.

- **Setting G codes that call custom macros of program Nos.9010 to 9019**

6050	G code that calls the custom macro of program number 9010
6051	G code that calls the custom macro of program number 9011
6052	G code that calls the custom macro of program number 9012
6053	G code that calls the custom macro of program number 9013
6054	G code that calls the custom macro of program number 9014
6055	G code that calls the custom macro of program number 9015
6056	G code that calls the custom macro of program number 9016
6057	G code that calls the custom macro of program number 9017
6058	G code that calls the custom macro of program number 9018
6059	G code that calls the custom macro of program number 9019

[Data type] Word

[Valid data range] 1 to 999

These parameters set the G codes that call the custom macros of program numbers 9010 through 9019.

> **NOTE**
> Setting value 0 is invalid. No custom macro can be called by G00.

- **Setting M codes that call subprograms of program Nos.9001 to 9009**

6071	M code that calls the subprogram of program number 9001
6072	M code that calls the subprogram of program number 9002
6073	M code that calls the subprogram of program number 9003
6074	M code that calls the subprogram of program number 9004
6075	M code that calls the subprogram of program number 9005
6076	M code that calls the subprogram of program number 9006
6077	M code that calls the subprogram of program number 9007
6078	M code that calls the subprogram of program number 9008
6079	M code that calls the subprogram of program number 9009

[Data type] Two–word

[Valid data range] 1 to 99999999

These parameters set the M codes that call the subprograms of program numbers 9001 through 9009.

NOTE
Setting value 0 is invalid. No custom macro can be called by M00.

- **Setting M codes that call custom macros of no.9020 to 9029**

6080	M code that calls the custom macro of program number 9020
6081	M code that calls the custom macro of program number 9021
6082	M code that calls the custom macro of program number 9022
6083	M code that calls the custom macro of program number 9023
6084	M code that calls the custom macro of program number 9024
6085	M code that calls the custom macro of program number 9025
6086	M code that calls the custom macro of program number 9026
6087	M code that calls the custom macro of program number 9027
6088	M code that calls the custom macro of program number 9028
6089	M code that calls the custom macro of program number 9029

[Data type] Two–word

[Valid data range] 1 to 99999

These parameters set the M codes that call the custom macros of program numbers 9020 through 9029.

NOTE
Setting value 0 is invalid. No custom macro can be called by M00.

- **ASCII codes that call subprogram of program No. 9004 and 9005**

6090	ASCII code that calls the subprogram of program number 9004
6091	ASCII code that calls the subprogram of program number 9005

[Data type] Byte

[Valid data range] 65 (A:41H) to 90 (Z:5AH)

These parameters set the ASCII codes that call subprograms in decimal.
Addresses that can be used are as follows:
T series : A, B, F, H, I, K, M, P, Q, R, S, T
M series: A, B, D, F, H, I, J, K, L, M, P, Q, R, S, T, X, Y, Z

NOTE
Set 0 when no subprogram is called

Alarm and message

Number	Message	Description
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro, or the header contents are improper in a high–speed cycle cutting. 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 – 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.

Number	Message	Description
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦ 3 is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
199	MACRO WORD UNDE FINED	Undefined macro word was used. Modify the custom macro.

Caution

CAUTION

Machine tool builders: You are requested to attach your custom macro program tape or program list to the CNC unit without fail.

If it is necessary to replace part program storage memory due to a failure, FANUC servicemen or end user in charge of maintenance should know the contents of your custom macro for the purpose of repairing the trouble immediately.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.15	Custom macro
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.15	Custom macro
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.15	Custom macro
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.15	Custom macro

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.

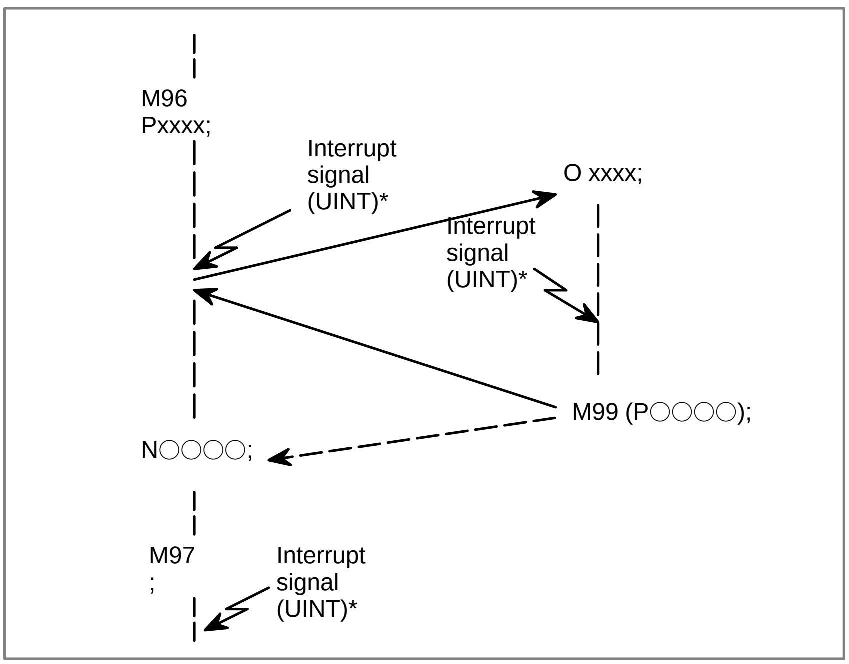


Fig 11.6.2 Interruption type custom macro function

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<G053#3>

[Classification] Input signal

[Function] This signal calls and executes a program in memory. During execution, a program in automatic operation is suspended.
To enable this signal to be accepted, a particular miscellaneous function must be specified in a command program for automatic operation. In addition, automatic operation must already be started to accept this signal.
The particular miscellaneous function code is set by parameter 6003, 6033 and 6034.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053					UINT			

Parameter

- **Various Setting for Custom Macro**

	#7	#6	#5	#4	#3	#2	#1	#0
6003	MUS	MCY	MSB	MPR	TSE	MIN	MSK	

[Data type] Bit

MSK Absolute coordinates at that time during custom macro interrupt

0 : Not set to the skip coordinates (system variables #5061 and later)
1 : Set to the skip coordinates (system variables #5601 and later)

MIN Custom macro interrupt

0 : Performed by interrupting an in–execution block (Custom macro interrupt type I)
1 : Performed after an in–execution block is completed (Custom macro interrupt type II)

TSE Custom macro interrupt signal UINT

0 : Edge trigger method (Rising edge)
1 : Status trigger method

MPR Custom macro interrupt valid/invalid M code

0 : M96/M97
1 : M code set using parameters (Nos. 6033 and 6034)

MSB Interrupt program

0 : Uses a dedicated local variable (Macro–type interrupt)
1 : Uses the same local variable as in the main program (Subprogram–type interrupt)

MCY Custom macro interrupt

0 : Not performed during cycle operation
1 : Performed during cycle operation

MUS Interrupt–type custom macro

0 : Not used
1 : Used

- **Setting M code that makes interruption effective and ineffective**

6033	M code that validates a custom macro interrupt
6034	M code that invalidates a custom macro interrupt

[Data type] Byte type

[Valid data range] 0 to 255

These parameters set the custom macro interrupt valid/invalid M codes.

> **NOTE**
> These parameters can be used when MPR, #4 of parameter No. 6003, is 1. M96 is used as a valid M code and M97 is used as an invalid M code when MPR is 0, irrespective of the state of this parameter.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.15.11	Interruption type custom macro
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.15.11	Interruption type custom macro
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.15.11	Interruption type custom macro
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.15.11	Interruption type custom macro

11.6.3 Custom Macro Variables Common to Two–path Control (Two–path Control)

General

With two–path control, common variables are provided separately for each path; variable #n used with a path 1 is different from variable #n used with path 2. By parameter setting (No. 6036 and No. 6037), however, some or all of common variables #100 to # 149 and #500 to #531 can be made usable commonly by path 1 and path 2 so that such variables can be written or read for either path. Such variables are referred to as custom macro variables common between two paths.

Parameter

- **Setting the no. of custom macro variables common between two paths**

6036	Number of custom macro variables common between two paths (#100's)

[Data type] Byte

[Unit of data] Number of custom macro variables

[Valid data range] 0 to 50

The parameter specifies the number of variables commonly used for both paths 1 and 2 (custom macro variables common between two paths) that are included in custom macro variables 100 to 149.
The custom macro variables common between two paths can be written or read for either of the paths.

Example) When this parameter is set to 10, the custom macro variables are specified as follows:
Custom macro variables 100 to 109: Used commonly between two paths
Custom macro variables 110 to 149: Used independently for each path

> **NOTE**
> 1 This parameter is dedicated to the 2–path control.
> 2 When this parameter is set to 0, custom macro variables 100 to 149 are not used commonly between two paths.

- **Setting the No. of custom macro variables common between two paths**

6037	Number of custom macro variables common between two paths (#500's)

[Data type] Byte

[Unit of data] Number of custom macro variables

[Valid data range] 0 to 32

This parameter specifies the number of variables commonly used for both paths 1 and 2 (custom macro variables common between two paths) that are part of custom macro variables 500 to 531.
The custom macro variables common to tool posts can be written or read for either of the tool posts.

Example) When this parameter is set to 10, the custom macro variables are specified as follows:
Custom macro variables 500 to 509: Used commonly between two paths
Custom macro variables 510 to 531: Used independently for each path

NOTE
1 This parameter is dedicated to the 2–path control.
2 When this parameter is set to 0, custom macro variables 500 to 531 are not used commonly between two paths.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.21.3	Custom macro variables common to tool posts
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.20.5	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.

Explanations

A canned cycle consists of a sequence of six operations.

Operation 1	Positioning a hole position
Operation 2	Rapid traverse up to point R level
Operation 3	Hole machining
Operation 4	Operation at the bottom of a hole
Operation 5	Retraction to point R level
Operation 6	Rapid traverse up to the initial point

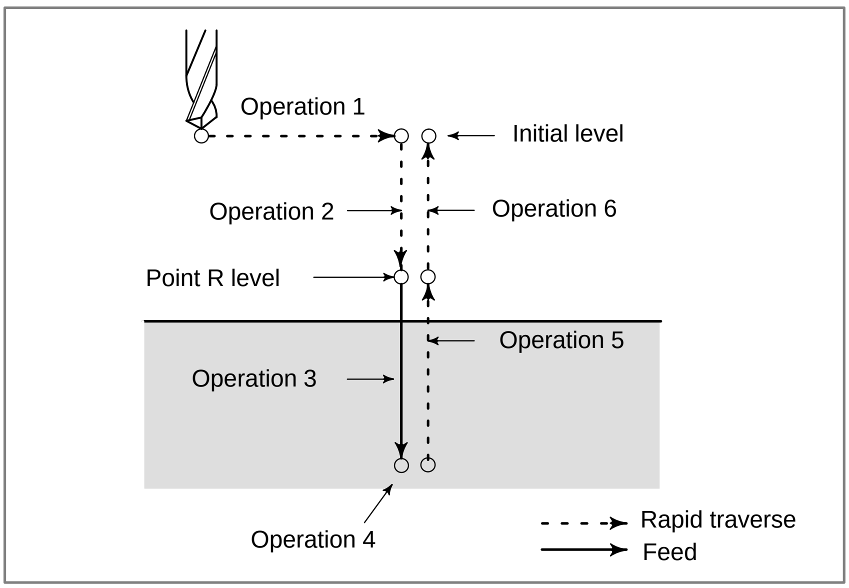


Fig. 11.7 (a) Canned cycle operation sequence

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:

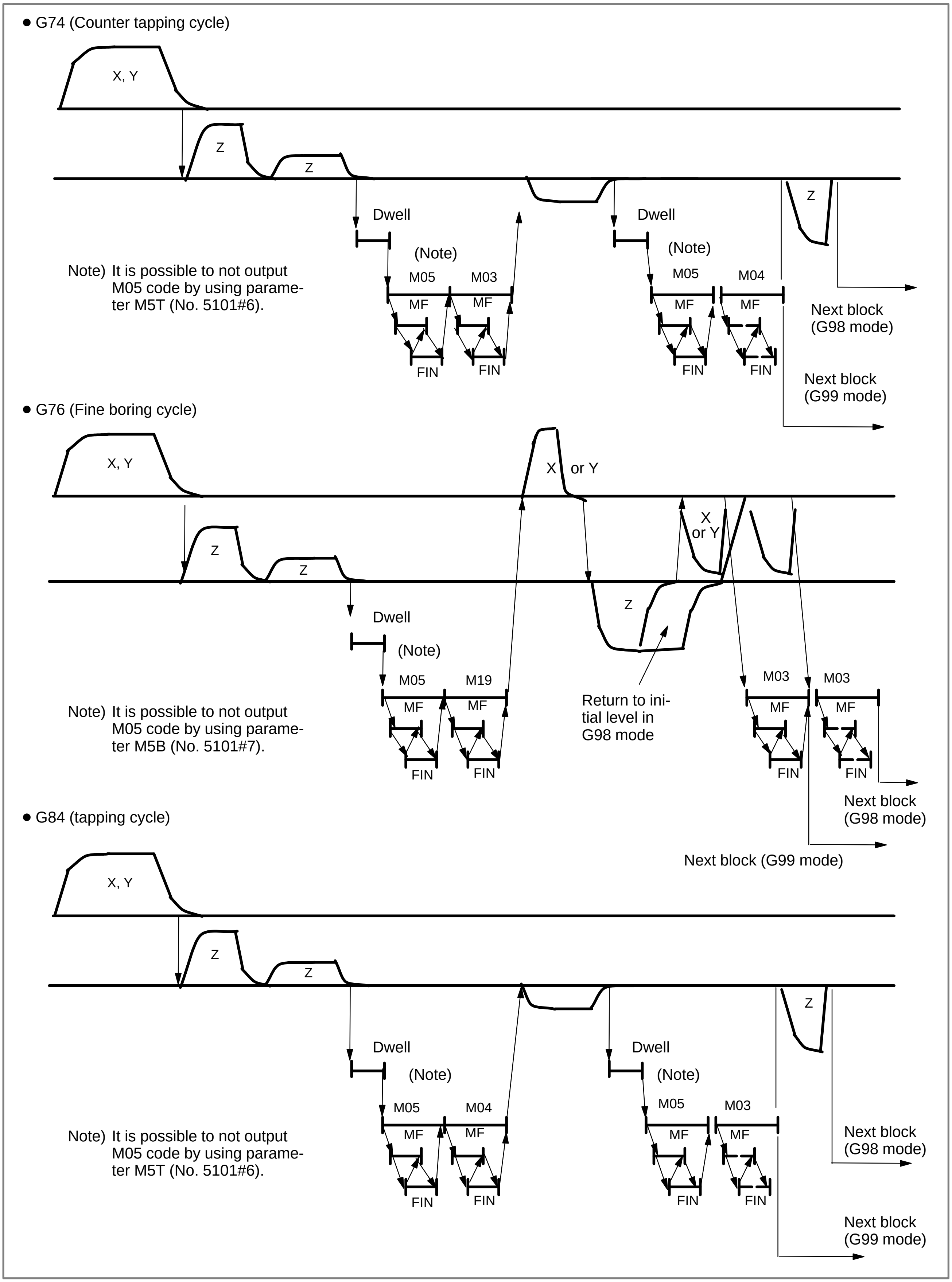


Fig. 11.7 (b) Canned cycle for M series (1/2)

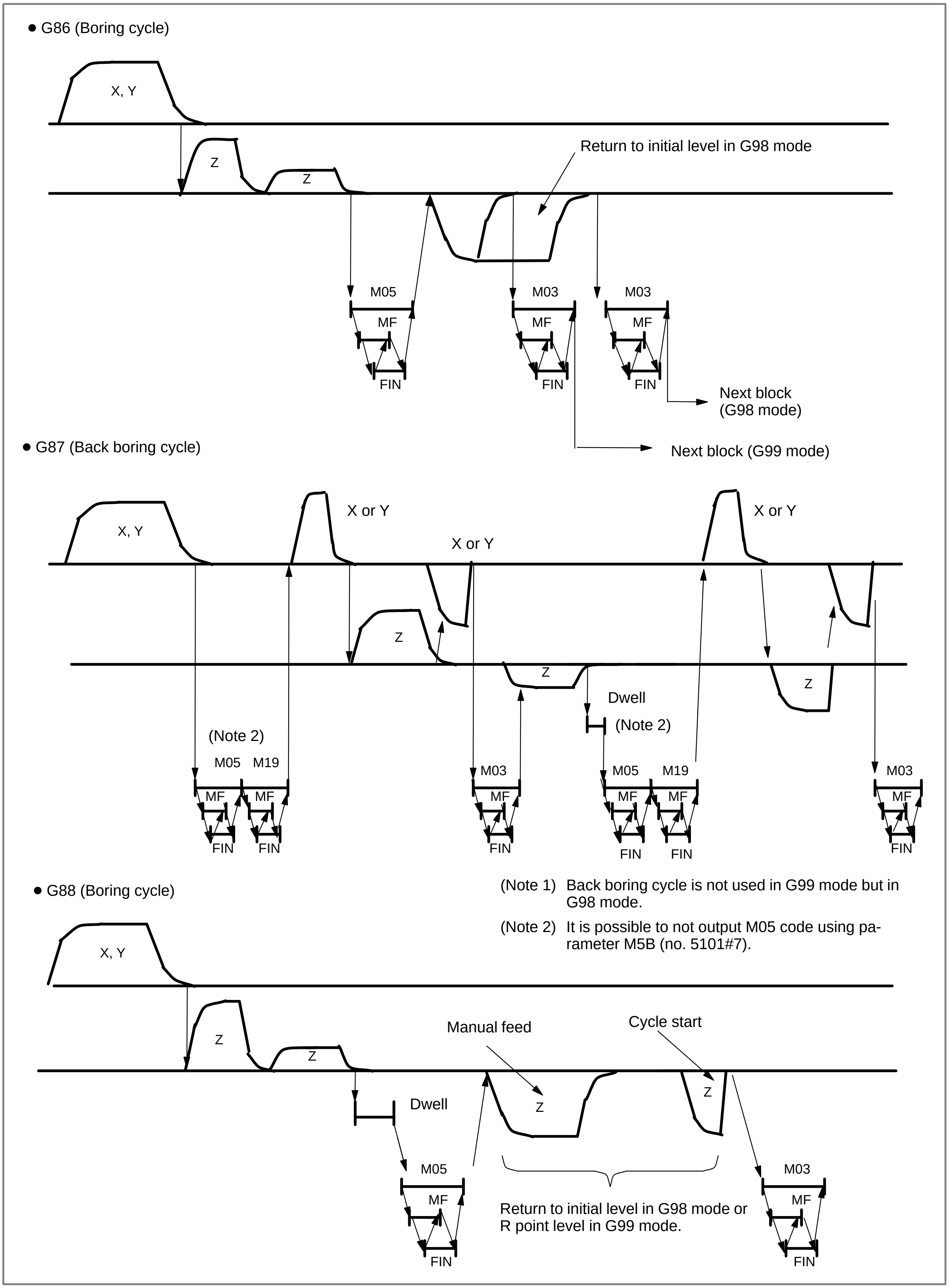
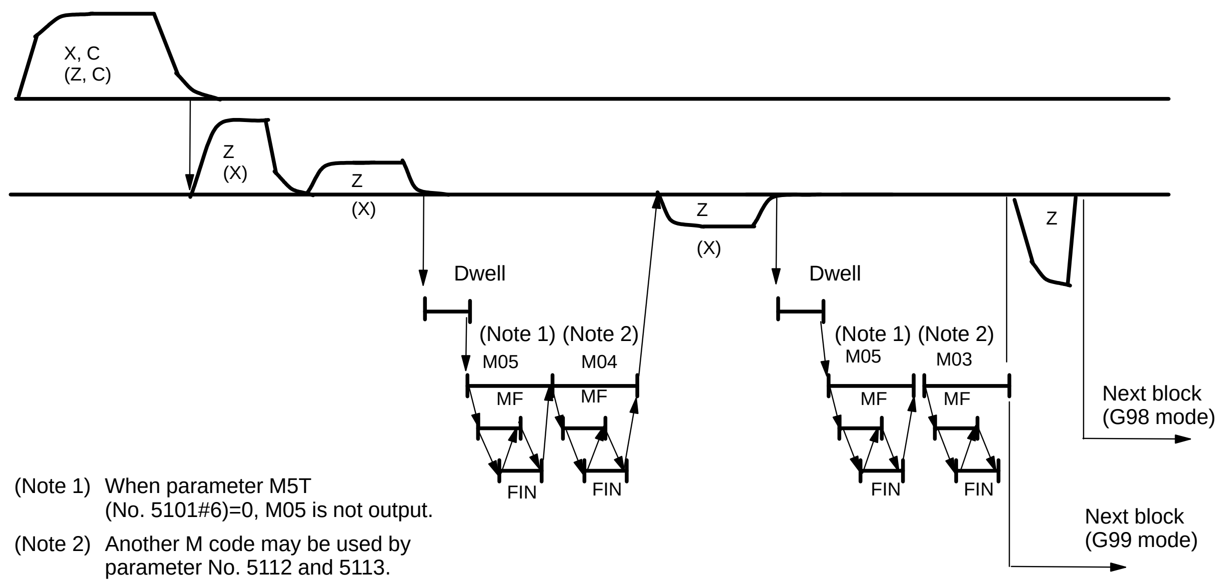


Fig. 11.7 (c) Canned cycle for M series (2/2)

- G84 (Face tapping cycle)
  G88 (Side tapping cycle)



(Note 1) When parameter M5T (No. 5101#6)=0, M05 is not output.

(Note 2) Another M code may be used by parameter No. 5112 and 5113.

Fig 11.7 (d) Canned cycle for T series

- **M code used for C–axis clamp/unclamp (T series)**

When an M code specified in parameter No.5110 for C–axis clamp/unclamp is 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. 5111.

Tapping signal

During a tapping cycle, the tapping signal is output. The tapping signal is also output while the G code of the tapping cycle is valid.

Override

During tapping, cutting feedrate override is always set to 100%.

Feed hold

When the feed hold key is pressed during tapping, the movement is not stopped immediately but the movement is stopped when the tool is returned to level R.

Dry run

The TDR bit (bit 5 of parameter No. 1401) specifies whether dry run is valid during tapping.

Signal

Tapping signal TAP <F001#5>

[Classification] Output signal

[Function] Reports that the system is in tapping mode.

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

– The system is in tapping cycle mode.
  G74, G84: M series
  G84, G88: T series
– The system is in tapping mode.
  G63: M series

The signal is set to 0 when:

– The system is in neither tapping cycle mode nor tapping mode.
– A reset or emergency stop is specified.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F001			TAP					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101		M5T			ILV	RTR		FXY
	M5B	M5T	RD2	RD1			EXC	FXY

[Data type] Bit

FXY The drilling axis in the drilling canned cycle is:

0 : Always the Z–axis
1 : The axis selected by the program

> **NOTE**
> In the case of the T system, this parameter is valid only for the drilling canned cycle in the Series 15 format.

EXC G81

0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

RTR G83 and G87

0 : Specify a high–speed peck drilling cycle
1 : Specify a peck drilling cycle

ILV Initial point position in drilling canned cycle

0 : Not updated by reset
1 : Updated by reset

RD2, RD1 Set the axis and direction in which the tool in drilling canned cycle G76 or G87 is got free. RD2 and RD1 are set as shown below by plane selection.

RD2	RD1	G17	G18	G19
0	0	+X	+Z	+Y
0	1	–X	–Z	–Y
1	0	+Y	+X	+Z
1	1	–Y	–X	–Z

M5T When a spindle rotates from the forward to the reverse direction and vice versa in tapping cycles G84 and G74 for M series (G84 and G88 for T series), befor M04 or M03 is output:
For T series

0 : Not output M05
1 : Outputs M05

For M series

0 : Outputs M05
1 : Not output M05

M5B In drilling canned cycles G76 and G87:

0 : Outputs M05 before an oriented spindle stops
1 : Not output M05 before an oriented spindle stops

	#7	#6	#5	#4	#3	#2	#1	#0
5102	RDI	RAB						

[Data type] Bit

RAB The R command for the drilling canned cycle in the Series 15 format is:

0 : Regarded as an incremental command
1 : Regarded as:
An absolute command in the case of G code system A
An absolute command in the case of G code system B or C when the G90 mode is specified.
An incremental command in the case of G code system B or C when the G91 mode is specified.

RDI The R command for the drilling canned cycle in the Series 15 format:

0 : Is regarded as the specification of a radius
1 : Follows the specification of a diameter/radius for the drilling axis

	#7	#6	#5	#4	#3	#2	#1	#0
5103								
								SIJ

[Data type] Bit

SIJ A tool shift value for the drilling canned cycle G76 or G87 is specified by:

0 : Address Q
1 : Address I, J, or K

5110	C–axis clamp M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 99

This parameter sets the C–axis clamp M code in a drilling canned cycle.

5111	Dwell time when C–axis unclamping is specified in drilling canned cycle

[Data type] Word

[Unit of data] ms

[Valid data range] 0 to 32767

This parameter sets the dwell time when C–axis unclamping is specified in a drilling canned cycle.

5112	Spindle forward–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle forward–rotation M code in a drilling canned cycle.

NOTE
M03 is output when “0” is set.

5113	Spindle reverse–rotation M code in drilling canned cycle

[Data type] Byte

[Valid data range] 0 to 255

This parameter sets the spindle reverse–rotation M code in a drilling canned cycle.

NOTE
M04 is output when “0” is set.

5114	Return or clearance value of drilling canned cycle G83
	Return value of high–speed peck drilling cycle G73

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 32767

For M series, this parameter sets the return value in high–speed peck drilling cycle G73 (G83 for T series).

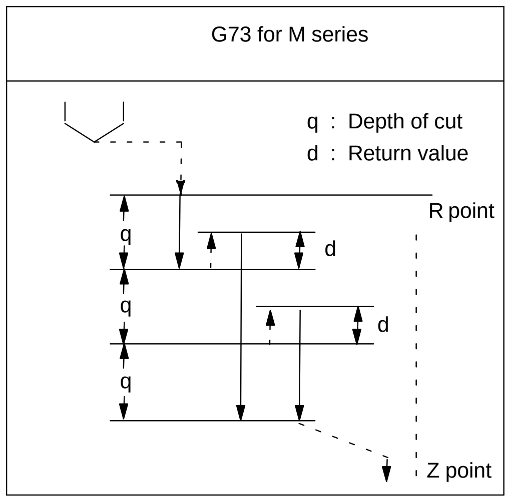


Fig. 11.7 (e) High–speed peck drilling cycle (G73) for M series

For T series, this parameter sets the return or clearance value in drilling canned cycle G83.

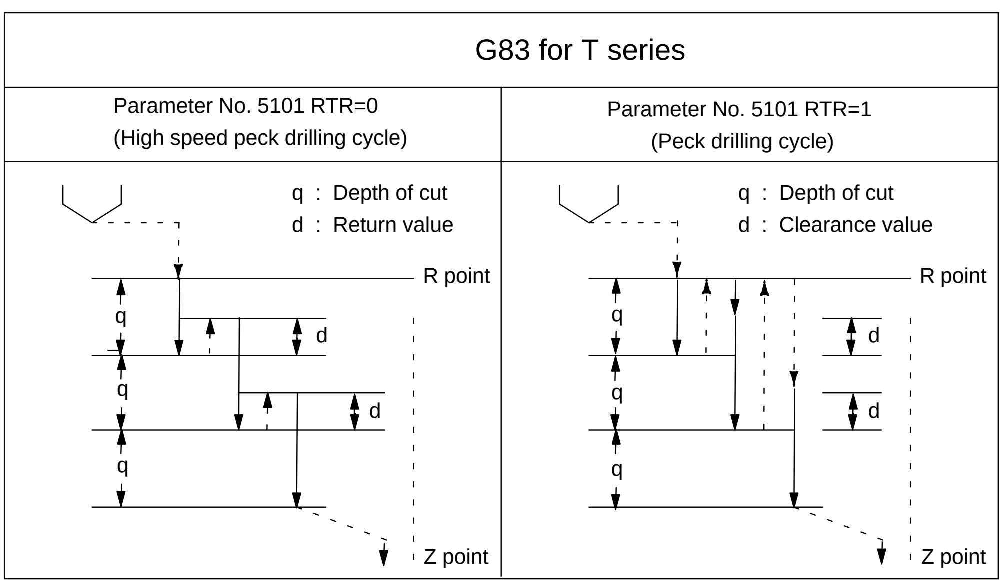


Fig. 11.7 (f) Drilling canned cycle (G83) for T series

5115	
	Clearance canned cycle G83

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.001	mm
Inch input	0.001	0.0001	0.0001	inch

[Valid data range] 0 to 32767

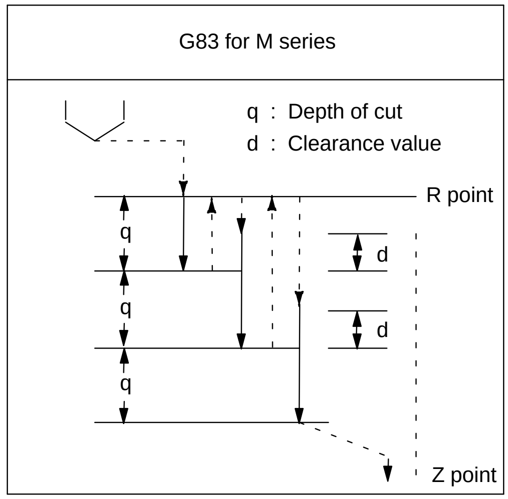


Fig. 11.7 (g) Peck drilling cycle (G83) for M series

Alarm and message

Number	Message	Description
044	G27–G30 NOT ALLOWED IN FIXED CYCLE (M series)	One of G27 to G30 is commanded in a canned cycle mode. Modify the program.

Note

NOTE
A parameter FXY (No. 5101#0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.13.1	Canned cycle
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.13.3	Canned cycle for hole machining
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.13.1	Canned cycle
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.13.3	Canned cycle for hole machining

11.8 EXTERNAL MOTION FUNCTION (M SERIES)

General

Upon completion of positioning in each block in the program, an external operation function signal can be output to allow the machine to perform specific operation.

G81 IP_ ; (The IP_ is axis move command)

Every time positioning for the IP_ move command is completed, the CNC sends an external operation signal to the machine. An external operation signal is output for each positioning operation until canceled by G80 or a group 01 G code.

No external operation signals are output during execution of a block that contains neither X nor Y.

Basic procedure

1 Once positioning for a move command has been completed, the CNC sets the external operation signal EF to 1.

2 When the EF signal is set to 1, the PMC executes drilling or another operation. Once the operation has been completed, the PMC sets completion signal FIN to 1.

3 The CNC resets the EF signal to 0 upon the elapse of the time (TFIN) specified in parameter No. 3011 after the FIN signal is set to 1.

4 When the EF signal is set to 0, the PMC resets the FIN signal to 0.

5 The CNC starts executing the next block.

The timing diagram is shown below:

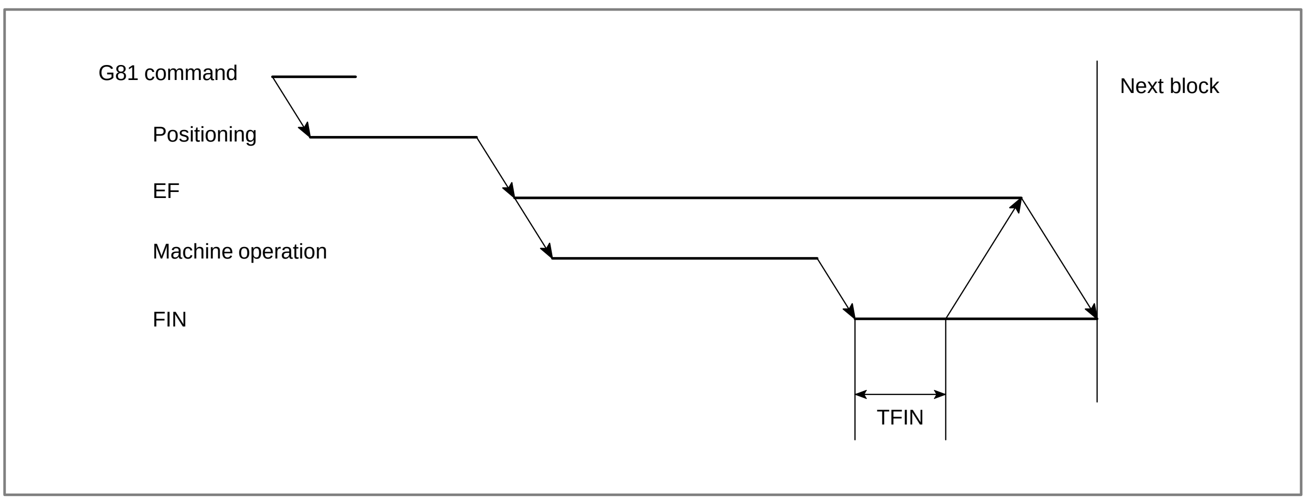


Fig. 11.8 (a) Timing diagram of basic procedure

Signal

External Operation Signal EF<F008#0>

[Classification] Output signal

[Function] Reports that the positioning of G81 has been completed in the external motion function, and that a special external operation is required.

[Output condition] For details of the output condition and procedure, see the ”basic procedure”, described previously.
For details of completion signal FIN, see section 8.1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F008								EF

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5101							EXC	

[Data type] Bit

EXC G81:

0 : Specifies a drilling canned cycle
1 : Specifies an external operation command

Caution

CAUTION

1 When this function is used, canned cycles (G73, G74, G76, and G82 to G89) cannot be used.
2 When the high–speed M, S, T, or B interface is used, the signals used by this function are transferred in high–speed mode. See Section 8.4.

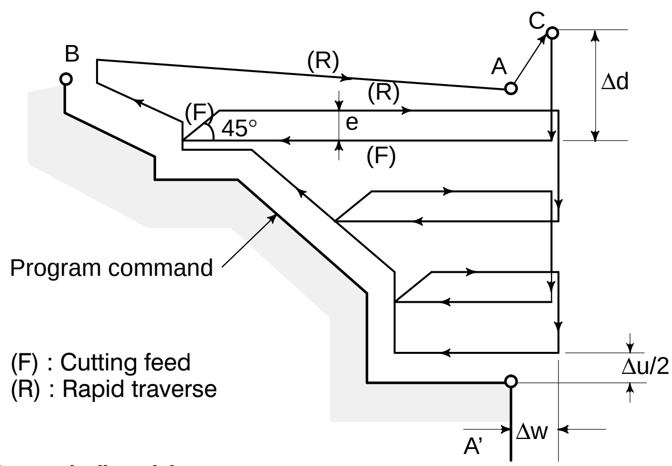
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.13.8	External operation function (G81)
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.13.8	External operation function (G81)

11.9 CANNED CYCLE (T SERIES)/MULTIPLE REPETITIVE CANNED CYCLE (T 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. 5132), and the parameter is changed by the program command.
- e : Escaping amount
  This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.
- ns : Sequence number of the first block for the program of finishing shape.
- nf : Sequence number of the last block for the program of finishing shape.
- Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).
- Δw : Distance and direction of finishing allowance in Z direction.
- f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Signal

Chamfering signal CDZ<G053#7>

[Classification] Input signal

[Function] Executes chamfering in a threading cycle. Specify the chamfering distance in parameter No. 5130.

[Operation] When the signal is set to 1, chamfering is not executed in the threading cycle.
When the signal is set to 0, chamfering is executed in the threading cycle.

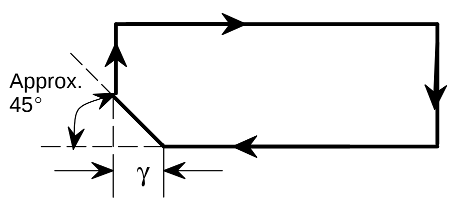


Fig. 11.9 (a) Straight thread cutting cycle

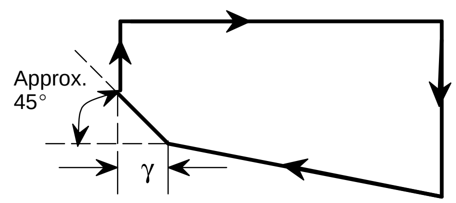


Fig. 11.9 (b)Taper thread cutting cycle

Set the chamfering distance γ to the parameter No. 5130. 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

	#7	#6	#5	#4	#3	#2	#1	#0
G053	CDZ							

Parameter

- **Various setting for multiple repetitive canned cycle**

	#7	#6	#5	#4	#3	#2	#1	#0
5102						QSR	MRC	

[Data type] Bit

MRC When a target figure other than a monotonically increasing or monotonically decreasing figure is specified in a multiple repetitive turning canned cycle (G71, G72):

0 : No alarm occurs.
1 : P/S alarm No. 064 is occurs.

> **NOTE**
> This parameter is valid for multiple repetitive turning canned cycle type I.

QSR Before a multiple repetitive canned cycle (G70 to G73) is started, a check to see if the program contains a block that has the sequence number specified in address Q is:

0 : Not made.
1 : Made. (If the sequence number specified in address Q cannot be found, an alarm occurs and the canned cycle is not executed.)

• Chamfering distance in thread cutting cycles G76 and G92

5130	Chamfering distance in thread cutting cycles G76 and G92

[Data type] Byte

[Unit of data] 0.1

[Valid data range] 0 to 127

This parameter sets the chamfering distance in thread cutting cycles G76 and G92.

• Depth of cut in multiple repetitive canned cycles G71 and G72

5132	Depth of cut in multiple repetitive canned cycles G71 and G72

[Data type] Two–word

[Unit of data]

Increment system	IS–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 depth of cut in multiple repetitive canned cycles G71 and G72.

• Escape in multiple repetitive canned cycles G71 and G72.

5133	Escape in multiple repetitive canned cycles G71 and G72.

[Data type] Two–word

[Unit of data]

Increment system	IS–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 escape in multiple repetitive canned cycles G71 and G72.

- **Escape in multiple repetitive canned cycles G73**

5135	Escape in multiple repetitive canned cycle G73 in X–axis direction
5136	Escape in multiple repetitive canned cycle G73 in Z–axis direction

[Data type] Two–word

[Unit of data]

Increment system	IS–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 escape in multiple repetitive canned cycle G73 of an X, then Z axis.

- **Division count in multiple repetitive canned cycle G73**

5137	Division count in multiple repetitive canned cycle G73

[Data type] Two–word

[Unit of data]

[Valid data range] 0 to 99999999

This parameter sets the division count in multiple repetitive canned cycle G73.

- **Return in multiple canned cycles G74 and G75**

5139	Return in multiple canned cycles G74 and G75

[Data type] Two–word

[Unit of data]

Increment system	IS–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 return in multiple repetitive canned cycles G74 and G75.

- **Minimum depth of cut in multiple repetitive canned cycle G76**

5140	Minimum depth of cut in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–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 multiple repetitive canned cycle G76.

- **Finishing allowance in multiple repetitive canned cycle G76**

5141	Finishing allowance in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data]

Increment system	IS–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 multiple repetitive canned cycle G76.

- **Repetition count of final finishing in multiple repetitive canned cycle G76**

5142	Repetition count of final finishing in multiple repetitive canned cycle G76

[Data type] Two–word

[Unit of data] Cycle

[Valid data range] 1 to 99999999

This parameter sets the repetition count in multiple repetitive canned cycle G76.

- **Tool nose angle in multiple repetitive canned cycle G76.**

5143	Tool nose angle in multiple repetitive canned cycle G76.

[Data type] Two–word

[Unit of data] Degree

[Valid data range] 0 to 120 when 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.

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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.13.1 II.13.2	Canned cycle Multiple repetitive canned cycle
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.13.1 II.13.2	Canned cycle Multiple repetitive canned cycle

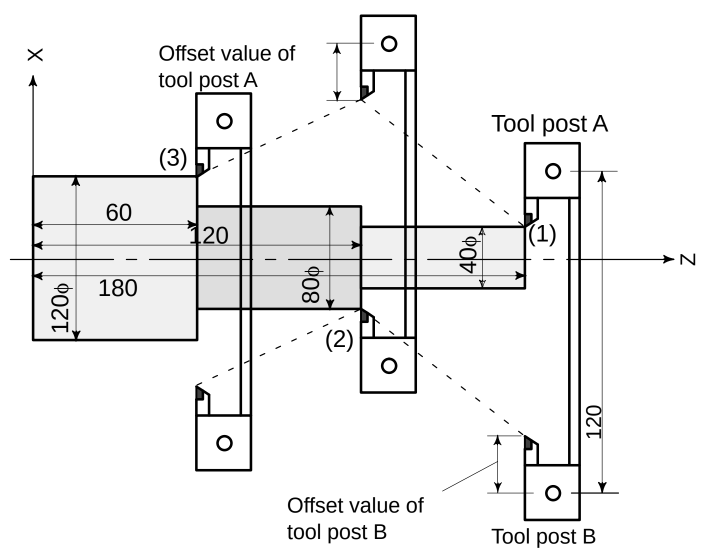
11.10 MIRROR IMAGE FOR DOUBLE TURRETS (T SERIES)

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. 1290).
Program example for double turrets.



X40.0 Z180.0 T0101 ;	Position turret A at (1)
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position turret B at (2)
G69 ;	Shift the coordinate system by the distance B to A, and cancel mirror image.
X120.0 Z60.0 T0101 ;	Position turret A at (3)

Parameter

- **Distance between two turrets**

1290	Distance between two turrets 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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.13.6	Mirror image for double turrets
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.13.5	Mirror image for double turrets

11.11 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 <F061#0> 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 *BEUCL <G038#6> turns to "0".

(4) The CNC then turns the B axis unclamp signal BUCLP to "0", to indicate it received the *BEUCL signal.

(5) When the PMC is notified that BUCLP has been turned to "0", the PMC should turn *BEUCL 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<F061#1>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 <G038#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.

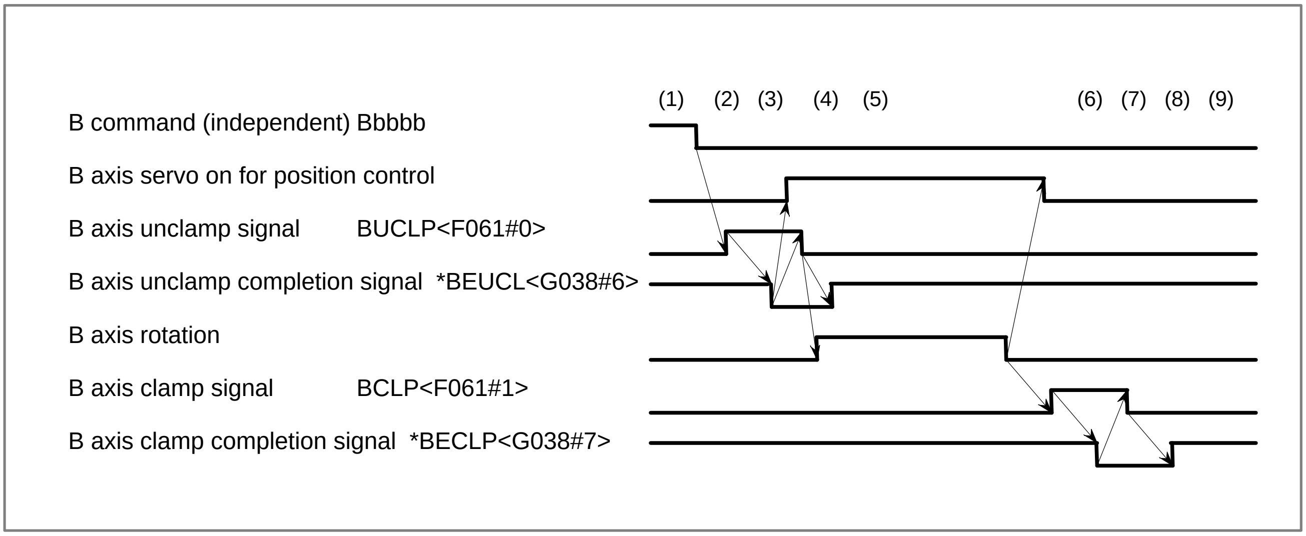


Fig. 11.11 (a) Time chart for positioning index table (type A)

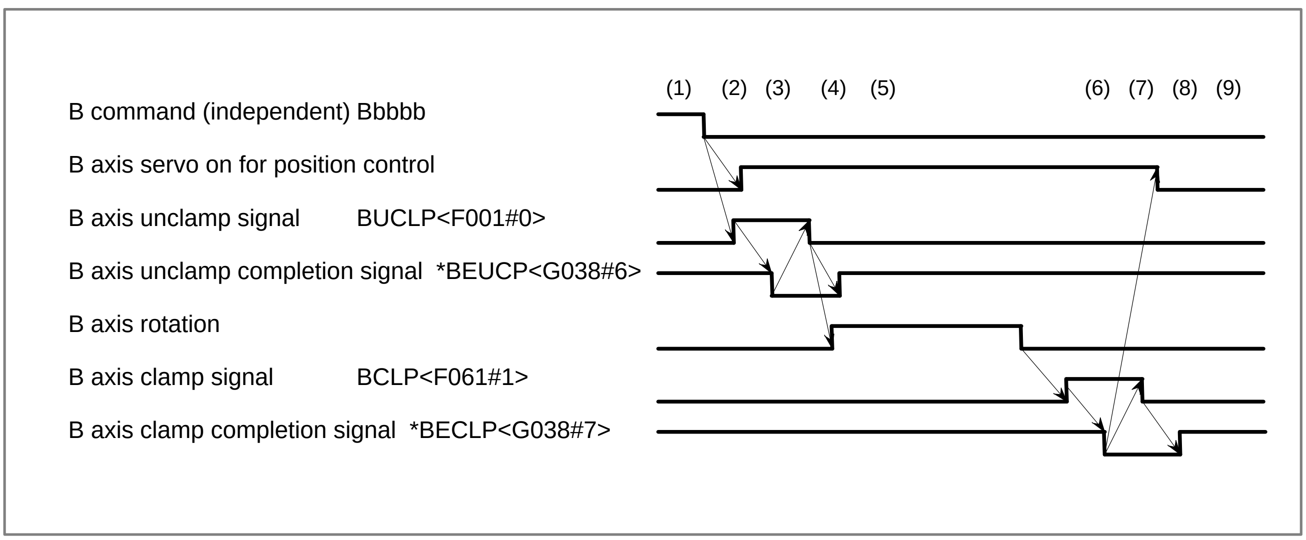


Fig. 11.11 (b) Time chart for positioning index table (type B)

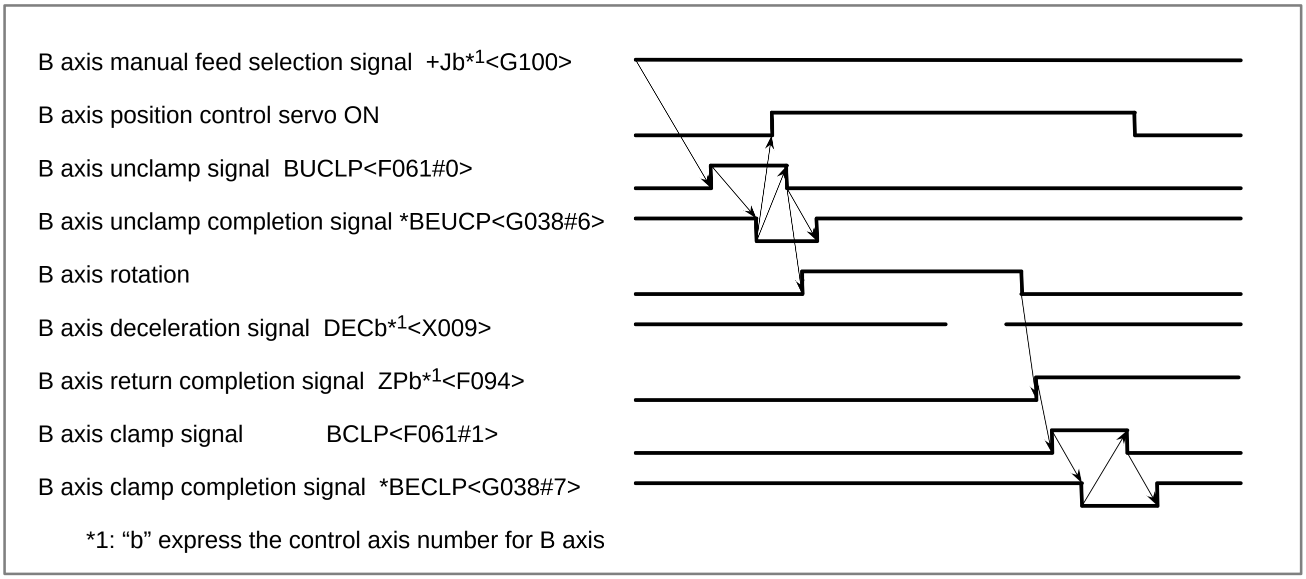


Fig. 11.11 (c) Manual reference position return of B axis time chart (type A)

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. 5512. 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 (INC, bit 3 of parameter No. 5500)
– 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. 5511).

Absolute/incremental programming

Setting G90, bit 4 of parameter No. 5500, specifies absolute programming, irrespective of G90/G91 mode.

Signal

B axis clamp signal BCLP<F061#1>

[Classification] Output signal

[Function] Instructs the PMC side to clamp the B axis mechanically with a clutch or shot pin.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis clamp completion signal BECLP<G038#7>

[Classification] Input signal

[Function] Notifies the CNC of completion of the B axis clamp operation.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp signal BUCLP<F061#0>

[Classification] Output signal

[Function] Instructs the PMC side to release the B axis from the mechanical clamp.

[Output condition] The output condition and procedure are the same as those described in the basic procedure for positioning the index table.

B axis unclamp completion signal *BEUCP<G038#6>

[Classification] Input signal

[Function] Notifies the CNC of completion of the release of the B axis from the mechanical clamp.

[Operation] The operation and procedure are the same as those described in the basic procedure for positioning the index table.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G038	*BECLP	*BEUCP						

	#7	#6	#5	#4	#3	#2	#1	#0
F061							BCLP	BUCLP

Parameter

- **Setting linear or rotation axis**

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

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

[Data type] Bit axis

ROTx, ROSx Setting linear or rotation axis

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

- **Various setting for index table indexing**

	#7	#6	#5	#4	#3	#2	#1	#0
5500	IDX			G90	INC	ABS	REL	DDP

[Data type] Bit

DDP 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)

REL Relative position display of index table indexing axis

0 : Not rounded by 360 degrees
1 : Rounded by 360 degrees

ABS Displaying absolute coordinate value of index 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.

INC Rotation in the G90 mode when negative–direction rotation command M code (parameter No. 5511) is not set

0 : Not set to the shorter way around the circumference
1 : Set to the shorter way around the circumference (Set ABS, #2 of parameter No. 5500, to 1.)

G90 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

IDX Index table indexing sequence

0 : Type A
1 : Type B

- **Negative direction rotation command M code**

5511	Negative0direction 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 (INC, #3 of parameter No. 5500).
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.

NOTE
Set ABS, #2 of parameter No. 5500, to 1.

• Unit of index table indexing angle

5512	Unit of index table indexing angle

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm

[Valid data range] 0 to 360000

This parameter sets the unit of index table indexing angle. 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	ILLEGAL ANGLE COMMAND	The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.
136	ILLEGAL AXIS COMMAND	In index table indexing , another control axis was instructed together with the B axis. Modify the program.

Caution

CAUTION
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. (Set 1 in the ROTx bit (bit 0 of parameter No. 1006).)
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 should be performed.
7 No movement can be performed by automatic return from the reference position (G29), return to the second reference position (G30), or selection of the machine coordinate system (G53).
8 Only the fourth axis can be used as the index table indexing axis.

Reference Item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.13.11	Index table indexing function
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.13.5	Index table indexing function

11.12 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. 5400#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. 5411) 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	**X_Y_Z_** :	Absolute command for center coordinate value of scaling
⋮	Scaling is effective. (Scaling mode)	**P_** :	Scaling magnification
G50 ;	Scaling cancel		

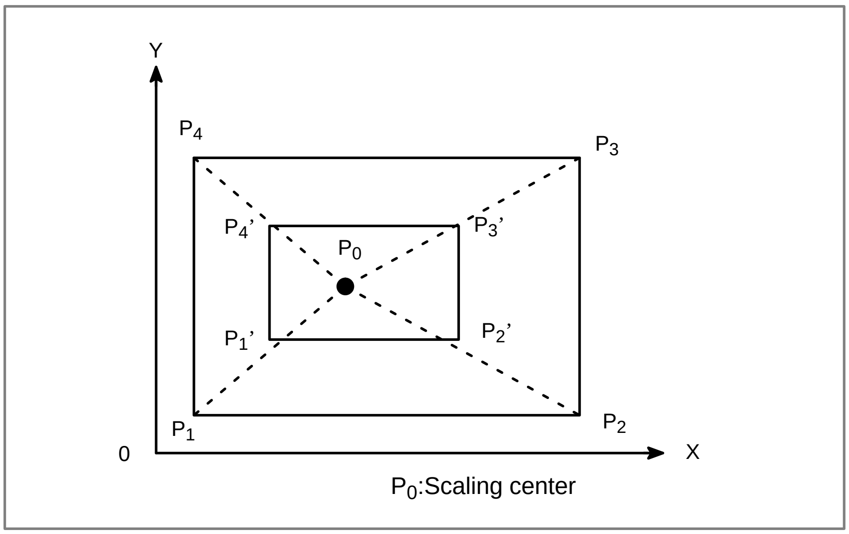


Fig. 11.12 (a) Scaling (P_1 P_2 P_3 P_4→P_1'P_2'P_3'P_4')

- **Scaling of each axis, programmable mirror image (negative magnification)**

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter XSC (No. 5400#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 5401#0) to enable scaling along each axis.
Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001(set parameter SCR (No. 5400#7)).
Magnification is set to parameter 5421 within the range ±0.00001 to ±9.99999 or ±0.001 to ±9.999.
If a negative value is set, mirror image is effected.
If magnification I, J or K is not commanded, a magnification value set to parameter (No. 5421) 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
⋮	Scaling is effective. (Scaling mode)	**I_J_K_:**	Scaling magnification for X axis, Y axis and Z axis respectively
G50	Scaling cancel		

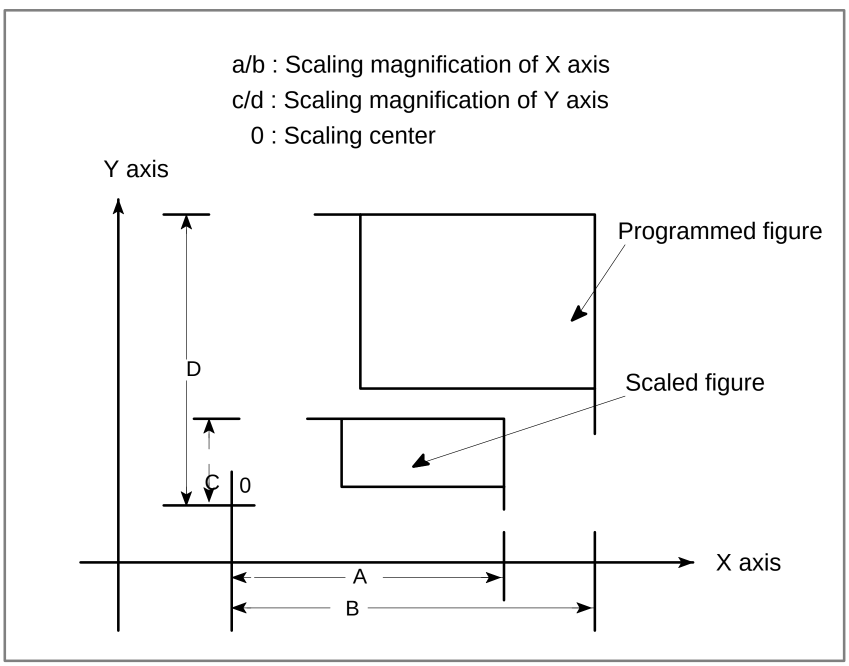


Fig. 11.12 (b) Scaling of each axis

Parameter

• Setting valid/invalid and magnification of scaling

	#7	#6	#5	#4	#3	#2	#1	#0
5400	SCR	XSC						

[Data type] Bit

XSC Axis scaling and programmable mirror image

0 : Invalidated (The scaling magnification is specified by P.)
1 : Validated

SCR Scaling magnification unit

0 : 0.00001 times (1/100,000)
1 : 0.001 times

• Valid/invalid setting to each axis scaling

	#7	#6	#5	#4	#3	#2	#1	#0
5401								SCLx

[Data type] Bit axis

SCLx Scaling for every axis

0 : Invalidated
1 : Validated

• Magnification used when scaling magnification is not specified

5411	Magnification used when scaling magnification is not specified

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] 1 to 999999

This parameter sets the scaling magnification. This setting value is used when a scaling magnification (P) is not specified in the program.

> **NOTE**
> Parameter No. 5421 becomes valid when scaling for every axis is valid. (XSC, #6 of parameter No. 5400 is “1”.)

- **Scaling magnification for every axis**

5421	Scaling magnification for every axis

[Data type] Two–word axis

[Unit of data] 0.001 or 0.00001 times (Selected using SCR, #7 of parameter No. 5400)

[Valid data range] $-999999 \sim -1$, $1 \sim 999999$

This parameter sets the scaling magnification for every axis.

Alarm and message

Number	Message	Description
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 P_p, or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.14.9	Scaling (G50, G51)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.14.9	Scaling (G50, G51)

11.13 COORDINATE SYSTEM ROTATION

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.

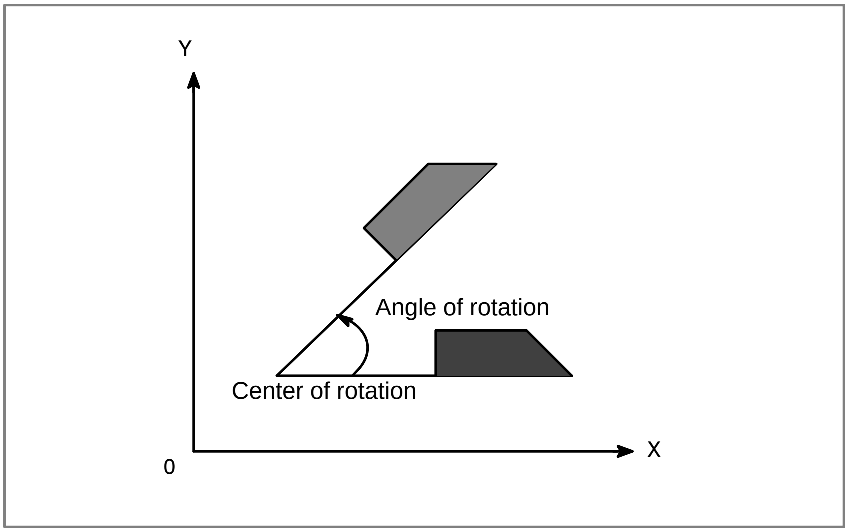


Fig. 11.13 (a) Coordinate system rotation

FORMAT	
{G17 / G18 / G19} **G68 (G68.1)**α_β_**R_** ; Start rotation of a coordinate system.	
	Coordinate system rotation mode (The coordinate system is rotated.)
G69 (G69.1);	Coordinate system rotation cancel command
Note:G68/G69 for M series, G68.1/G69.1 for T series.	

MEANING OF COMMAND

G17 (G18 or G19) : Select the plane 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 5400#0 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg
Valid data range : –360.000 to 360.000

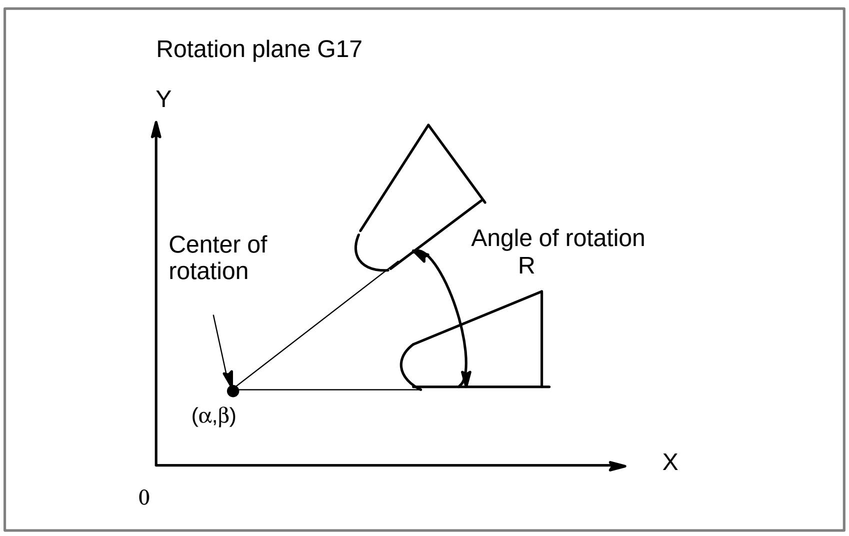


Fig. 11.13 (b) Coordinate system rotation

Parameter

- **Angle specification method of coordinate system rotation**

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Data type] Bit

RIN Coordinate rotation angle command (R)

0 : Specified by an absolute method
1 : Specified by G90 or G91

- **Angular displacement used when no angular displacement is specified for coordinate system rotation**

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation

[Data type] Two–word

[Unit of data] 0.001 degrees

[Valid data range] –360000 to 360000

This parameter sets the angular displacement for coordinate system rotation. When the angular displacement 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 SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.

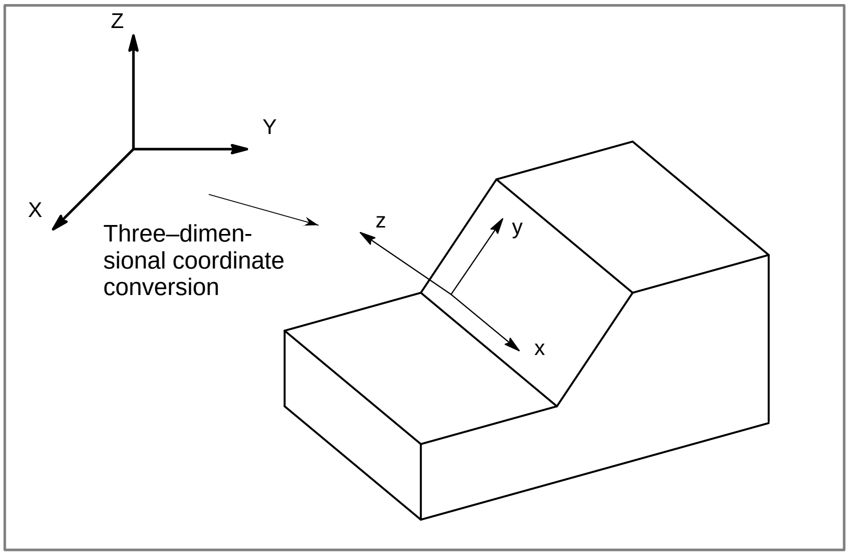
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.14.10	Coordinate system rotation (G68, G69)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.14.7	Coordinate system rotation (G68.1, G69.1)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.14.8	Coordinate system rotation (G68, G69)

11.14 THREE–DIMENSIONAL COORDINATE CONVERSION (M SERIES)

General

The coordinate system can be rotated about an axis by specifying the center of rotation, direction of the axis of rotation, and angular displacement. This coordinate conversion function is quite useful for three–dimensional machining using a diesinking machine. By applying three–dimensional coordinate conversion to a program generated for machining on XY plane, identical machining can be executed on a desired plane.



Parameter

- **Setting relative position and absolute position**

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				

[Data type] Bit

DRL Relative position

0 : The actual position displayed takes into account tool length offset.
1 : The programmed position displayed does not take into account tool length offset.

DRC Relative position

0 : The actual position displayed takes into account cutter compensation.
1 : The programmed position displayed does not take into account cutter compensation.

DAL Absolute position

0 : The actual position displayed takes into account tool length offset.
1 : The programmed position displayed does not take into account tool length offset.

DAC Absolute position

0 : The actual position displayed takes into account cutter compensation.
1 : The programmed position displayed does not take into account cutter compensation.

To execute absolute position display with three–dimensional coordinate conversion, the DRL, DRC, DAL, and DAC bits must be set to 1.

- **Setting absolute coordinates in the three–dimensional coordinate conversion mode**

	#7	#6	#5	#4	#3	#2	#1	#0
3106		DAK						

[Data type] Bit

DAK When absolute coordinates are displayed in the three–dimensional coordinate conversion mode:

0 : Coordinates in the program coordinate system are displayed.
1 : Coordinates in the workpiece coordinate system are displayed.

This parameter is significant only in three–dimensional coordinate conversion mode.

Alarm and message

Number	Message	Description
047	ILLEGAL AXIS SELECT	For startup of three–dimensional tool compensation or three–dimensional coordinate conversion, two or more axes were specified in the same direction (basic and parallel axes.)
048	BASIC 3 AXIS NOT FOUND	For startup of three–dimensional tool compensation or three–dimensional coordinate conversion, the three basic axes used when X_p, Y_p, and Z_p are omitted were not specified in parameter No. 1022.
5043	TOO MANY G68 NESTING	G68, three–dimensional coordinate conversion, was specified three times or more.
5044	G68 FORMAT ERROR	A format error occurred in a block including G68. The alarm is issued when any of the following takes place. (1) In a block including G68, any of I, J, and K is not specified (the option for coordinate rotation is not provided). (2) In a block including G68, all of I, J, and K is 0. (3) In a block including G68, R is not specified.

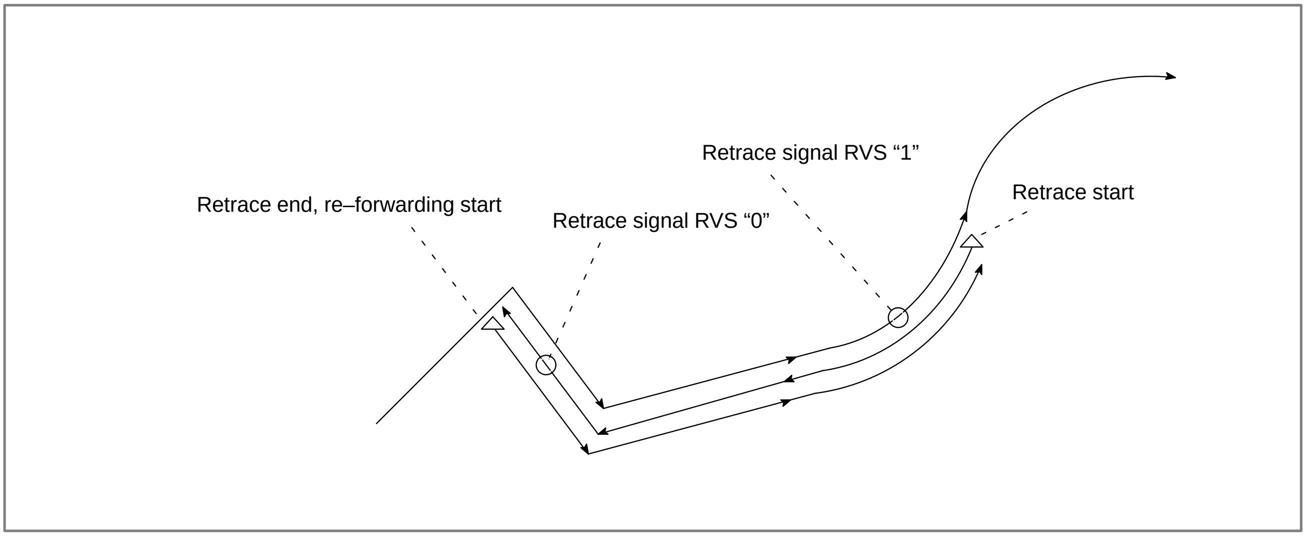
Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.13.10	Three–dimensional coordinate conversion (G68, G69)

11.15 RETRACE (M SERIES)

General

A tool can retrace the tool path along which the tool has moved. This operation is referred to as retrace. In addition, a tool can move forward again along the path that the tool has retraced. This operation is referred to as re–forward. The tool can then resume machining according to the program when it returns to the position where it started retrace.



By using the retrace signal RVS, which is an input signal from the PMC to the control unit, retrace and re–forward can be performed. Retrace is performed when the retrace signal RVS turns to “1”. Re–forward operation is performed when the retrace signal turns to “0”. In retrace, a tool can retrace blocks that have been executed in the automatic operation mode (memory command, tape command, manual data input), but a limit is imposed on the number of blocks that can be retraced. For detailed information, refer to the operator’s manual.
Whether to use the same federate as specified for normal operation or use a feedrate dedicated to retrace can be determined by setting parameter No. 1414.

Signal

Retrace signal (RVS) <G007#0>

[Classification] Input signal

[Function] Directs the control unit to retrace the tool along the path which the tool was moved in automatic operation (memory command, tape command, manual data input).

[Operation] When RVS turns to "1" while the tool is being moved forward (hereafter referred to as forward), the tool retraces the tool path along which it was moved. Note, however, that the tool does not start retrace immediately after this signal turns to "1" ; the tool starts retrace after the block performing the current forward operation is executed. When RVS turns to "0" while retrace is in progress, the tool switches from retrace operation to re–forward. In this case also, the tool does not start re–forward to resume machining immediately after this signal turns to "0" retrace; the tool starts re–forward after retracing all commands of the block currently subject to retrace.

The movement of a tool can be immediately switched from forward to retrace or from retrace to re–forward. For this purpose, first turn the feed hold signal *SP to "0" to stop automatic operation. Then, change the state of RVS after the automatic operation start in–progress signal STL turns to "0" and the automatic operation stop state is set. And the feed hold lamp SPL turns to "1". Next, turn feed hold signal *SP the automatic operation start signal ST from "1" to "0" to start automatic operation. Then, the tool can switch its movement to retrace or re–forward during execution of a block.

Retrace–in–progress signal (RVSL) <F082#2>

[Classification] Output signal

[Function] Notifies the PMC that retrace is in progress.

[Output condition] This signal turns to "1"when:

· Tool is in retrace with the retrace signal RVS turned to "1".

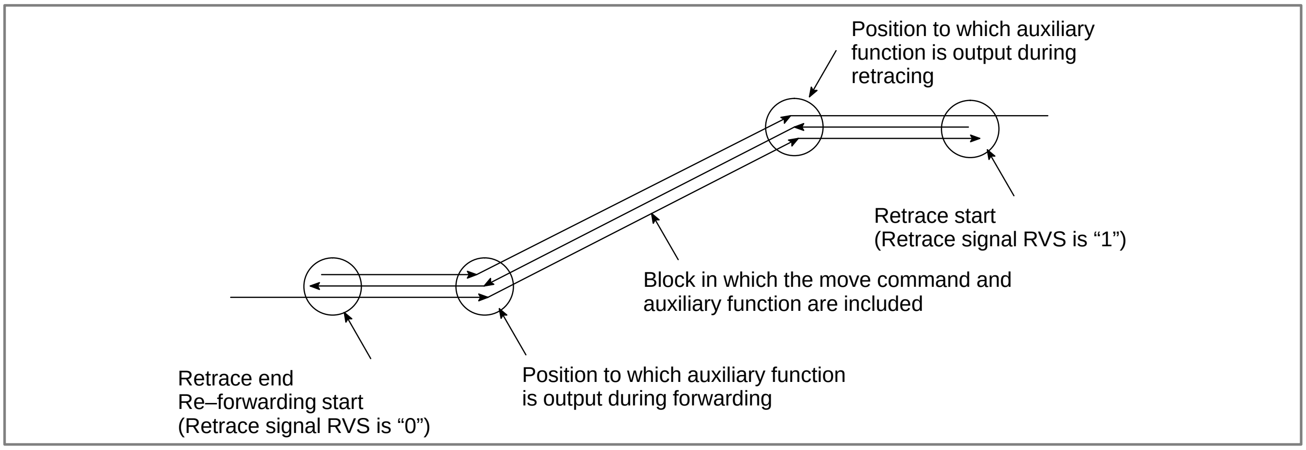
This signal turns to "0"

· The tool is in forward or re–forward with the retrace signal RVS turned to "0".

· The tool is at stop because of no block to retrace during retracing.

When the tool is in retrace, the M functions, S functions, T functions, and second auxiliary functions are executed in the same way as when the tool is moving forward. During retrace, this signal can be used on the PMC, if required, to prevent these functions from being executed in the same way as when the tool is moving forward.

Particularly when an M function, S function, T function, or second auxiliary function is specified in a block containing a move command, the positions where the code signals and strobe signal are output differ, depending on whether the tool is in forward (or re–forward) or retrace. So, take action on the PMC by using this signal and distribution end signal DEN as required.



Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G007								RVS

	#7	#6	#5	#4	#3	#2	#1	#0
F082						RVSL		

Parameter

1414	Feedrate for retrace

[Data type] Two–word

This parameter sets the feedrate for retrace when the retrace function is used.

(1) For rapid traverse

[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
Rotation axis	1 deg/min	6 to 240000	6 to 100000

NOTE
When 0 is set in this parameter, the rapid traverse rate that is set in parameter No. 1420 is used for retrace.

(2) For cutting feed
When a value other than 0 is specified in this parameter, the same feedrate as an F command specified using the value without a decimal point is set and is used for retrace. When 0 is specified in this parameter, the programmed feedrate (F command) is used for retrace.

Alarm and message

While a tool is in retrace, the retrace–in–progress signal RVSL is sent, and the character string RVRS blinks on the CRT screen to signal that the tool is currently in retrace. When a tool is in re–forward, the character string RTRY blinks to signal that the tool is currently in re–forward. The indication of RTRY continues until the tool returns to the block where retrace was started, that is, until forward movement is resumed. When there are no more retraceable blocks, the character string RVED blinks to signal that no further retrace can take place.

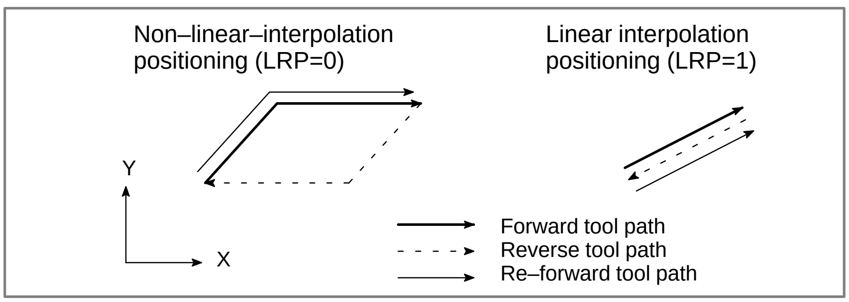
Warning

WARNING

Positioning (G00)

If non–linear–interpolation positioning is executed (the LRP bit (bit 1 of parameter No. 1401) is set to 0), the retrace tool path will not agree with the forward tool path. The re–forward tool path agrees with the forward tool path.

If the linear interpolation positioning is executed (the LRP bit (bit 1 of parameter No. 1401) is set to 1), the retrace tool path agrees with the forward tool path.



Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.4.11	Retrace function

11.16 MACRO COMPILER/ 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 FLASH ROM module, 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 FLASH ROM, 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 FLASH ROM, 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–61803E–1)

11.17 SMALL HOLE PECK DRILLING CYCLE (M SERIES)

General

This is a repetition of a peck drilling cycle in which when an overload torque detection signal (skip signal is used) is received with an arbor having an overload torque detection function during drilling, the tool is retracted, and the spindle speed and cutting feedrate are changed to retry machining.

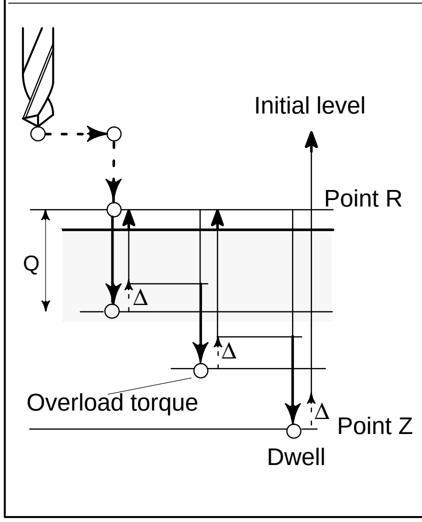
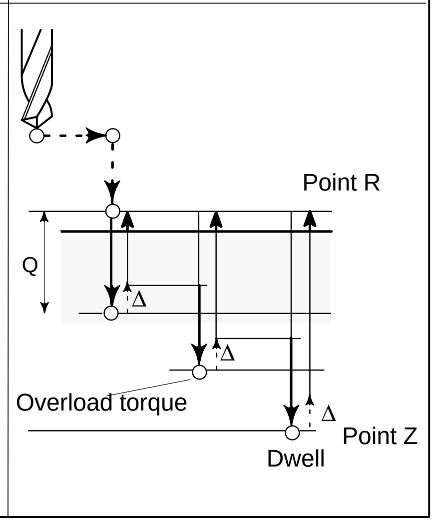
The cycle is realized by:

* X– and Y–axis positioning
* Positioning at point R along the Z–axis
* Cutting along the Z–axis (first time, depth of cut Q, incremental)

Repeated until point Z is reached:

Retracting
(bottom of hole → minimum clearance Δ , incremental)
Retracting
(→ to point R, absolute)
Forwarding
(point R → to point with hole bottom + clearance Δ, absolute)
Cutting
(second and subsequent times, cut of depth Q + Δ, incremental)

* Dwell
* Return to point R along the Z–axis (or initial point) = end of cycle

G83(G98)	G83(G99)
	

Change of cutting conditions

The cutting conditions are changed at each pecking operation (forwarding → cutting → retracting) during one G83 cycle. (Appropriate setting of bits 1 and 2 of parameter No. 5160 can specify that the cutting conditions are not to be changed.)

• Changing of cutting feedrate

The cutting feedrate programmed with the F word is changed during each of the second and subsequent cutting operations. Parameter Nos. 5166 and 5167 specify the ratio of change for a case in which a skip signal was received during the previous cutting operation and a case in which no skip signal was received during the previous cutting operation, respectively.

$$\boxed{\text{Cutting feed rate} = F \times \alpha}$$

(First time) $\alpha = 1.0$

(Second and subsequent times) $\alpha = \alpha \times \beta \div 100$
(where β is the ratio of change for the first time)

Skip during the previous cutting: $\beta = b1\%$ (parameter No. 5166)
No skip during the previous cutting: $\beta = b2\%$ (parameter No. 5167)

When the ratio α at which the cutting feedrate is changed becomes less than the ratio specified in parameter No. 5168, changing the cutting feedrate is discontinued. The upper limit to the newly specified cutting feedrate is the maximum cutting feedrate.

• Changing of spindle speed

The spindle speed programmed with the S word is changed during each of the second and subsequent cutting operations. Parameter Nos. 5164 and 5165 specify the ratio of change for a case in which a skip signal was received during the previous cutting operation and a case in which no skip signal was received during the previous cutting operation, respectively.

$$\boxed{\text{Spindle speed} = S \times \gamma}$$

(First time) $\gamma = 1.0$

(Second and subsequent times) $\gamma = \gamma \times \delta \div 100$
(where δ is the ratio of change for the first time)

Skip during the previous cutting: $\delta = d1\%$ (parameter No. 5164)
No skip during the previous cutting: $\delta = d2\%$ (parameter No. 5165)

If the cutting feedrate is clamped at the lower limit, the spindle speed is not changed. The upper limit to the newly specified spindle speed is the maximum S analog data.

Signal

Overload torque signal SKIP<X004#7>

[Classification] Input signal

[Function] Retracts a tool if an overload torque is applied it.

[Operation] When this signal becomes “1”, the control unit operates as follows:

- Assuming that an overload torque is applied to a tool, retracts it, and changes the spindle speed and cutting feedrate, then retries machining.
- This signal is valid, when the drill axis is between points R and Z and is moving forward or in cutting operation (so that the tool can retract).

> **NOTE**
> This signal is used also as a skip signal. (See Section 14.3.)

Small–diameter peck drilling in progress signal PECK2<F066#5>

[Classification] Output signal

[Function] Indicates whether small–diameter peck drilling is in progress.

[Output condition] This signal becomes “1” under the following conditions.

- When the tool returns from drill axis point R positioning to R point/initial level, after it was positioned at the hole position along a nondrill axis.
- The signal does not become “1” during positioning at the hole position.

Signal address

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

	#7	#6	#5	#4	#3	#2	#1	#0
F066			PECK2					

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

[Data type] Bit

OLS When an overload torque signal is received in a peck drilling cycle of a small diameter, the feed and spindle speed are

0 : Not changed.
1 : Changed.

NOL When the depth of cut per action is satisfied although no overload torque signal is received in a peck drilling cycle of a small diameter, the feed and spindle speed are:

0 : Not changed.
1 : Changed.

5163	M code that specifies the peck drilling cycle mode of a small diameter

[Data type] Two–word

[Unit of data]

[Valid data range] 1 to 99999999

This parameter sets an M code that specifies the peck drilling cycle mode of a small diameter.

5164	Percentage of the spindle speed to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed when the tool is retracted because the overload torque signal is received in a peck drilling cycle of a small diameter.

$S2 = S1 \times d1 \div 100$

S1: Spindle speed to be chaged
S2: Spindle speed changed

d1 is set as a percentage.

5165	Percentage of the spindle speed to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed when the tool is retracted without the overload torque signal received in a peck drilling cycle of a small diameter.

$S2 = S1 \times d2 \div 100$

S1: Spindle speed to be chaged
S2: Spindle speed changed

d2 is set as a percentage.

5166	Percentage of cutting feedrate to be changed when the tool is retracted after an overload torque signal is received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed when the tool is retracted because the overload torque signal is received in a peck drilling cycle of a small diameter.

$F2 = F1 \times b1 \div 100$
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b1 is set as a percentage.

5167	Percentage of the cutting feedrate to be changed when the tool is retracted without an overload torque signal received

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate tot be changed when the tool is retracted without the overload torque signal received in a peck drilling cycle of a small diameter.

$F2 = F1 \times b2 \div 100$
F1: Cutting feedrate to be changed
F2: Changed cutting feedrate
b2 is set as a percentage.

5168	Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Data type] Byte

[Unit of data] %

[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly in a peck drilling cycle of a small diameter to the specified cutting feedrate.

$FL = F \times b3 \div 100$
F: Specified cutting feedrate
FL: Changed cutting feedrate
Set b3 as a percentage.

5170	Number of the macro variable to which the total number of retractions during cutting is output

[Data type] Word

[Valid data range] 100 to 149

This parameter sets the number of the macro variable to which the total number of times the tool is retracted during cutting in a peck drilling cycle mode of a small diameter is output. The total number cannot be output to common variables 500 to 599.

5171	Number of the macro variable to which the total umber of retractions because of an overload signal is output

[Data type] Word

[Valid data range] 100 to 149

This parameter sets the common variable number of the custom macro to which the number of times the tool is retracted after the overload signal is received during cutting in a peck drilling cycle mode of a small diameter is output. The total number cannot be output to common variables 500 to 599.

5172	Speed of retraction to point R when no address I is issued

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 400

This parameter sets the speed of retraction to point R when no address I is issued in a peck drilling cycle of a small diameter.

5173	Speed of advancing to the position just before the bottom of a hole when no address I is issued

[Data type] Word

[Unit of data] mm/min

[Valid data range] 0 to 400

This parameter sets the speed of advancing to the position just before the bottom of a previously machined hole when no address I is issued in a peck drilling cycle of a small diameter.

5174	Clearance in a peck drilling cycle of a small diameter

[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Linear axis (millimeter input)	0.01	0.001	0.0001	mm
Linear axis (inch input)	0.001	0.0001	0.00001	inch

[Valid data range] 0 to 32767

This parameter sets the clearance in a peck drilling cycle of a small diameter.

Alarm and message

● Diagnostic display

520	Total number of retract operations during cutting since G83 was issued

521	Total number of retract operations due to reception of the overload torque signal since G83 was issued

The indications of DGN Nos. 520 and 521 are cleared by G80.

522	Coordinates at which the drill axis started retracting (least input command)

523	Difference between the previous and current coordinates at which the drill axis started retracting (least input increment: previous – current)

Warning

WARNING

Forwarding or retracting is not performed by rapid traverse positioning. Instead, it is performed with the same interpolation as for cutting feed. This means exponential acceleration/deceleration is performed; however, the tool life management function does not count the tool life during forwarding or retracting.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II. 13.1.7	Small hole peck drilling cycle
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II. 13.1.7	Small hole peck drilling cycle

11.18 HIGH–SPEED CYCLE MACHINING RETRACTING

General

When high–speed cycle machining is to be interrupted, retracting can be performed by setting the high–speed cycle machining retract signal HSRT to 1.
It is also possible to specify and execute a special type of escape after retracting by previously registering a retract cycle in the high–speed cycle header.

Retracting

To use retracting during high–speed cycle machining, it is necessary to previously set up an escape direction and speed for each axis using parameter No. 7514 and an amount of retracting (based on the specified distribution count) using parameter No. 7515. If the high–speed cycle machining retract signal HSRT becomes 1 during high–speed cycle machining, a parameter–specified escape operation is performed simultaneously with the current cycle operation. This composite operation enables the tool to escape while protecting the workpiece and tool from damage.

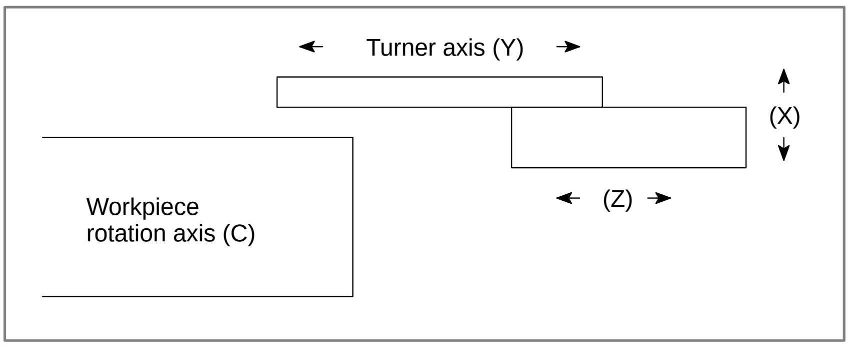


Fig. 11.18 Retracting performed on lead machine

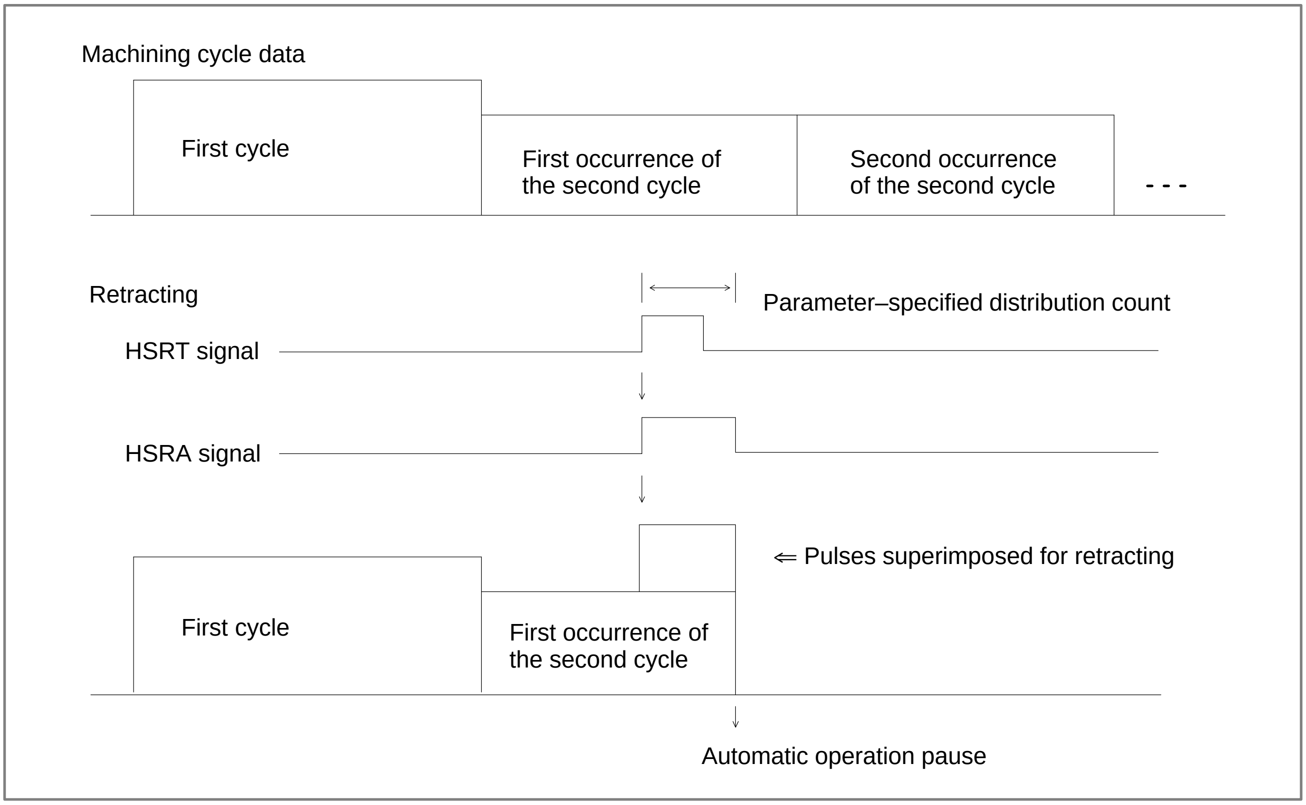
If a positive escape direction and speed (fx) for the X–axis are previously specified using parameters, turning on the retract signal enables the workpiece to retract along the X–axis simultaneously with operation on the turner axis and rotation around the C–axis. In addition, if an escape direction and speed for the Z–axis are specified previously, a simultaneous operation like an X–Z escape operation can also be performed.

Retracting begins in a distribution period during which the retract signal HSRT <G065#3> becomes 1. It ends when a retracting distribution count specified in parameter No. 7515 expires, or when distribution for the current cycle is completed. In this case, the cycle connection information and the repetition cycle count are ignored, and retract operation ends at the end of the current one cycle. Therefore, the actual distribution count used for retracting simultaneously with usual movement can range from 1 to the distribution count specified in parameter No. 7515 or a distribution count that is applied until the end of the corresponding cycle.
If no retract cycle is registered, the CNC causes automatic operation to pause after retracting ends. If a retract cycle is registered, the CNC enters a retract cycle state.

When retracting begins, the retracting signal HSRA <F062#2> becomes 1. Once retracting begins, resetting the retract signal HSRT to 0 does not stop retract operation.

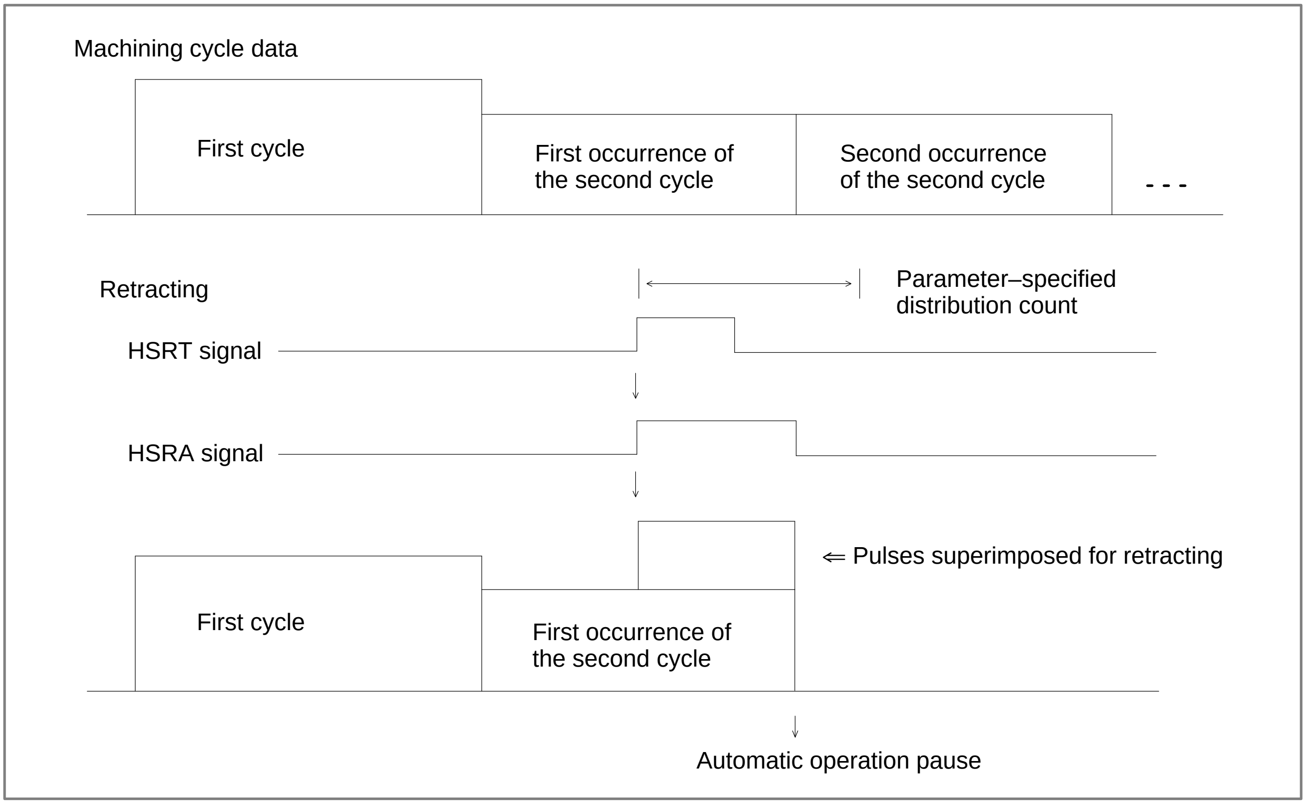
[Example]

(1) Parameter No. 7515 setting < remaining distribution count for the current cycle
A number of pulses for retracting based on the parameter–specified distribution count are superimposed on a usual operation. When the operation ends, the CNC shifts to an automatic operation pause or retract cycle execution.



If the parameter No. 7515 setting is 0, the distribution count is assumed to be infinite, allowing retracting to continue until the current cycle ends.

(2) Parameter No. 7515 setting >= remaining distribution count for the current cycle
A number of pulses for retracting until the end of the current cycle are superimposed on a usual operation. When retracting ends, the CNC shifts to an automatic operation pause or retract cycle execution.



The escape direction and speed for each axis are specified in parameter No. 7514 (axis type parameter). An algebraic sign prefixed to the escape speed represents the escape direction.

(Example) Parameter No. 7514: X = 10 (for metric machine)
Y = –20
Z = 0

Retracting occurs at a speed of 10 mm/min in the positive direction along the X–axis and 20 mm/min in the negative direction along the Y–axis, but does not occur along the Z–axis (0 mm/min).

A reset interrupts retracting.

Retract cycle

Cycle data can be registered with the cycle header. This data is used to move the tool to a safer position or to decelerate high–speed motion about a rotation axis and high–speed travel along other axes to a stop after retracting ends.

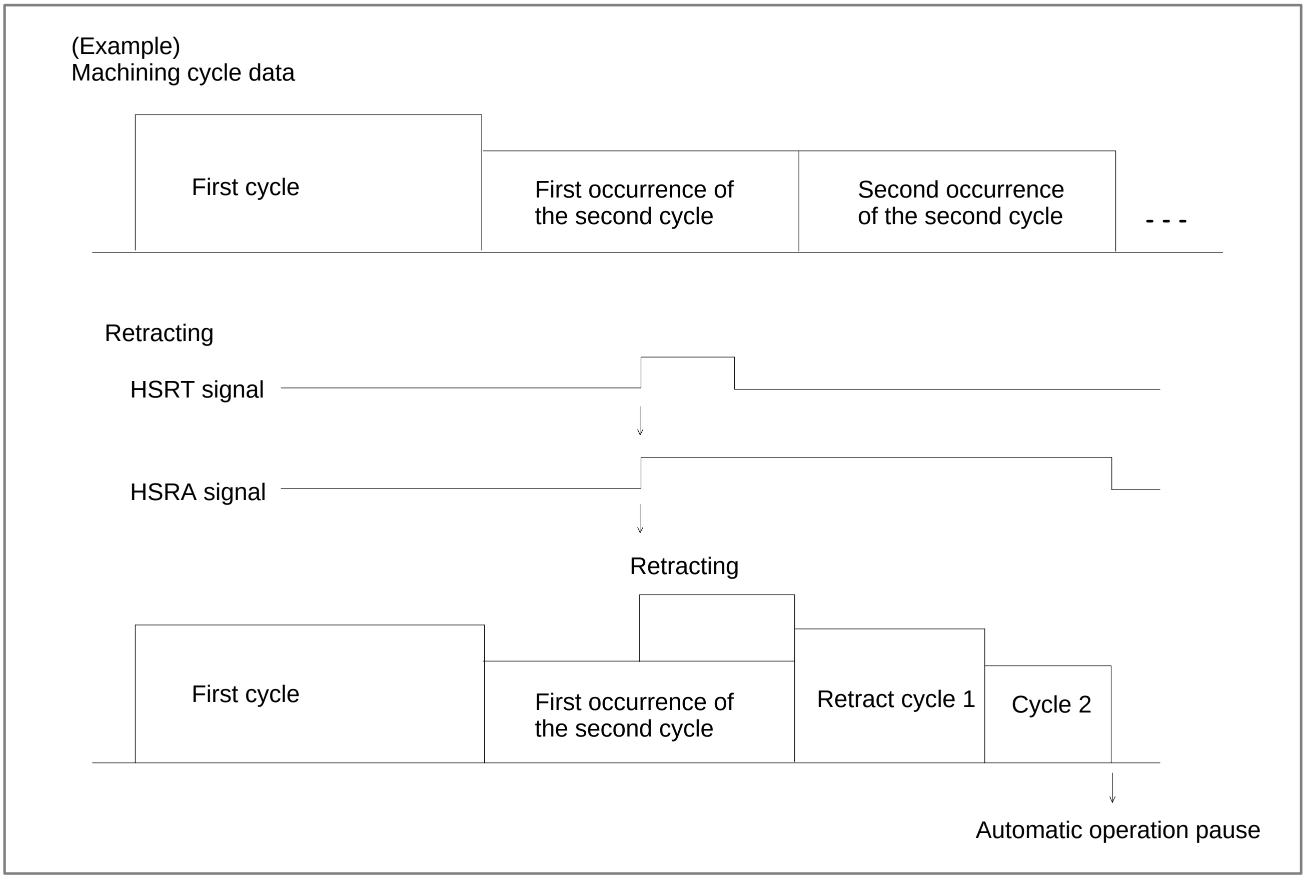
Whether to enable retracting can be selected using a data type specification variable in the cycle header. If retracting is disabled, the retract signal is ignored. In a connection cycle, retracting begins with the first enabled–retracting cycle to appear.

If no retract cycle is registered, the CNC shifts to an automatic operation pause (see Section 5.1) after retracting. If a retract cycle is registered, shifting to an automatic operation pause occurs after the retract cycle is executed.

Previously registering a retract cycle in the high–speed cycle header causes the retract cycle to be executed after retracting. If the machining cycle data is cycle connection information indicating that connection of more than one machining cycle is attempted, it is possible to register different retract cycles for each machining cycle or the same retract cycle for all machining cycles.

A retract cycle is created using the same specification methods (including the cycle connection, cycle repetition count, and data specification methods) as for ordinary cycles.

The retracting signal HSRA <F062#2> remains 1 during retracting.



Resetting parameter No. 7514 for retracting to all 0s makes it possible to execute a retract cycle without retracting (additional retract movement = 0).

A reset interrupts retracting.

Learning control

When retracting begins, a command is issued to disable learning control for all axes.

High–speed cycle header

Bit 7 of the data type specification variable in the high–speed cycle header specifies whether to enable retracting. If this bit is off, the retract signal HSRT is ignored; so neither retracting nor retract cycle is executed.

Data type specification variable

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
–	–	r6	r5	r4	r3	r2	r1	RT	–	t6	t5	t4	t3	t2	t1

RT: Retracting selection flag
- = 0 : Retracting is not to be performed.
- = 1 : Retracting is to be performed.

If retracting is enabled (RT = 1), the sixth–axis data specification variable works as the retract cycle connection information. This variable specifies a retract cycle number (1 to 999). If the variable is 0, it is assumed that there is no retract cycle.

NOTE
When the high–speed cycle machining retract function is used, up to five axes can be controlled for high–speed cycle machining.

Header configuration when retracting is enabled

Variable	Contents		
#20001/20017/20033.. (#200001/200017/200033..)	Cycle repetition count		
#20002/20018/20034.. (#200002/200018/200034..)	Cycle connection information		
#20003/20019/20035.. (#200003/200019/200035..)	Data count		
#20004/20020/20036.. (#200004/200020/200036..)	Data type	RT=1	
#20005/20021/20037.. (#200005/200021/200037..)	1st–axis data specification variable		
#20006/20022/20038.. (#200006/200022/200038..)	2nd–axis data specification variable		
#20007/20023/20039.. (#200007/200023/200039..)	3rd–axis data specification variable		
#20008/20024/20040.. (#200008/200024/200040..)	4th–axis data specification variable		
#20009/20025/20041.. (#200009/200025/200041..)	5th–axis data specification variable		
#20010/20026/20042.. (#200010/200026/200042..)	Retract cycle connection information		
#20011/20027/20043.. (#200011/200027/200043..)	1st–axis fixed–data count		
#20012/20028/20044.. (#200012/200028/200044..)	2nd–axis fixed–data count		
#20013/20029/20045.. (#200013/200029/200045..)	3rd–axis fixed–data count		
#20014/20030/20046.. (#200014/200030/200046..)	4th–axis fixed–data count		
#20015/20031/20047.. (#200015/200031/200047..)	5th–axis fixed–data count		
#20016/20032/20048.. (#200016/200032/200048..)	(Not used)		

NOTE
P–code variable numbers enclosed in parentheses apply to data variable addition A/B.

Signal

High–speed cycle machining retract signal HSRT<G065#3>

[Classification] Input signal

[Function] Starts high–speed cycle machining retracting.

[Operation] When this signal becomes 1, the control unit behaves as follows:

- If high–speed cycle machining is under way, and retracting is enabled for the current machining cycle, the control unit begins high–speed cycle machining retracting.

High–speed cycle machining retracting signal HSRA<F062#2>

[Classification] Output signal

[Function] Indicates that high–speed cycle machining retracting is under way.

[Output condition] This signal becomes 1 during:

- High–speed cycle machining retracting or retract cycle.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G065					HRST			

	#7	#6	#5	#4	#3	#2	#1	#0
F062						HSRA		

Parameter

7514	Escape direction and speed for high–speed cycle machining retracting

[Data type] Two–word axis

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	–30 ~ –240000 30 ~ 240000	–30 ~ –100000 30 ~ 100000
Inch machine	0.01 inch/min	–30 ~ –96000 30 ~ 96000	–30 ~ –4800 30 ~ 4800

This parameter specifies the escape direction and speed for each axis. An algebraic sign prefixed to the escape speed represents the escape direction.

7515	Retracting distribution count for high–speed cycle machining retracting

[Data type] Two–word

This parameter specifies a retracting distribution count for high–speed cycle machining retracting. If the machining cycle ends before the specified distribution count expires, retracting ends immediately. If this parameter is 0, it is assumed that the retracting distribution cycle is indefinite; so retracting continues until the current cycle ends.

Reference item

CONNECTION MANUAL (This manual)	11.5	High–speed cycle machining

12 DISPLAY/SET/EDIT

12.1 DISPLAY/SET

12.1.1 Clock Function

General

Time is displayed in the hour/minute/second format on each display screen. Some screens allows display of the year, month, and day.
The custom macro system variable can be used to read the time. 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, Janualy 23, 1997 is represented as 19970123.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time

12.1.2 Displaying Operation History

General

This function displays a history of the key and signal operations, performed by the CNC operator, upon the occurrence of a failure or CNC alarm. The history can also be displayed for previously generated CNC alarms.

The following history data is recorded:

(1) MDI key operation sequences, performed by the CNC operator
Example: A ~Z, <POS>, <PAGE ↑>, [SF1]

(2) On/off status transitions of selected input and output signals
Example: G0000.7↑ , SBK ↑

(3) CNC alarm information
Example: P/S0010

(4) Time (date, time) stamp
Example: 92/01/20
09:15:30

The history data can be output to an input/output device, connected via the reader/punch interface. Previously output history data can be input from an input/output device.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3106	OHS			OHD				

[Data type] Bit

OHD The operation history screen is:
0 : Not displayed.
1 : Displayed.

OHS Operation history sampling is:
0 : Performed.
1 : Not performed.

	#7	#6	#5	#4	#3	#2	#1	#0
3112			OPH					

NOTE
To put this parameter setting in effect, switch the power off then back on again.

OPH Specifies whether to enable the operation history function.
0 : Enabled.
1 : Disabled.

3122	Time interval used to record time data in operation history

[Data type] Word

[Unit of data] Minutes

[Valid data range] 0 to 1439

Time data is recorded in operation history at set intervals. When 0 is specified in this parameter, 10 minutes is assumed as the default. However, note that time data is not recorded if there is no data to be recorded at the specified time.

Note

NOTE

1 While the operation history screen is displayed, history data is not recorded.

2 When the duration of the on/off state of an input signal is 16 msec or shorter, that state is not recorded in the history. In addition, note that some signals are not recorded in the history.

3 Once the memory area becomes full, the oldest data is deleted to allow new data to be recorded. The memory area can store information corresponding to approximately 8000 key strokes.

4 Recorded history data is held in memory even after the power is turned off. Note, however, that the history data is erased by performing an entire–memory clear operation.

5 For the operation history function, sampling is disabled when bit 7 (OHS) of parameter No. 3106 is set to 1.

6 Be careful to set the date and time correctly.
When the time is recorded at regular intervals, but no data is recorded within a defined period, the time for that period is not recorded.
To input and output operation history data, the reader/punch interface option is required.

12.1.3 Help Function

General

The help function displays on the screen detailed information about alarms issued in the CNC and about CNC operations. The following information is displayed.

- **Detailed information of alarms**

When the CNC is operated incorrectly or an erroneous machining program is executed, the CNC enters the alram state. The help screen displays detailed information about the alarm that has been issued and how to reset it. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

When setting or referring to a system parameter, if you are not sure of the number of the parameter, the help screen displays a list of parameter Nos. for each function.

Note

NOTE
The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.13	HELP FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.13	HELP FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.13	HELP FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.13	HELP FUNCTION

12.1.4 Displaying Alarm History

General

Up to 25 of the most recent CNC alarms are stored and displayed on the screen.

The following information items are displayed.

(1) The date the alarm was issued

(2) Alarm No.

(3) Alarm message (some contains no message)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.7.2	ALARM HISTORY DISPLAY
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.7.2	ALARM HISTORY DISPLAY
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.7.2	ALARM HISTORY DISPLAY
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.7.2	ALARM HISTORY DISPLAY

12.1.5 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
3111								SVS

[Data type] Bit

SVS Servo tuning screen

0 : Not displayed

1 : Displayed

Reference item

Series 16*i*/160*i*/18*i*/180*i*	MAINTENANCE MANUAL (B–63005EN)	5.2	Servo Tuning Screen
Series 21*i*/210*i*	MAINTENANCE MANUAL (B–63085EN)	5.2	Servo Tuning Screen

12.1.6 Spindle Setting and Tuning Screen

General

On the spindle setting and 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
3111						SVP	SPS	

[Data type] Bit type

SPS Spindle setting and tuning screen

0 : Not displayed
1 : Displayed

SVP Synchronization errors displayed on the spindle setting and tuning screen

0 : Instantaneous values are displayed.
1 : Peak–hold values are displayed.

Reference item

MAINTENANCE MANUAL (B–63005EN)	6.1.2	Spindle setting and tuning screen

12.1.7 Waveform Diagnosis Display

General

Waveform diagnosis is classified into two main types.

(1) One–shot type

One–shot waveform diagnosis provides graphs of waveforms to illustrate changes in the following data.

In one–shot waveform diagnosis, the start of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the adjustment of servo motors and spindle motors.

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data

b. Combined feedrate for the first, second, and third axes

c. Spindle motor speed, load meter reading, and position deviation, converted to spindle position

d. On/off status of the machine signal, specified by a signal address

(2) Stored type

In stored waveform diagnosis, changes in the following data are recorded. When a servo alarm is issued, the recorded data can be read and displayed graphically as a waveforms.

In stored waveform diagnosis, the end of data collection can be triggered by the rising or falling edge of a machine signal.

This function is designed to facilitate the determination of a faulty location.

Stored data can be output via the reader/punch interface.

a. Servo motor error value along each axis, number of distributed pulses, torque, feedrate, current, and thermal simulation data

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112								SGD

[Data type] Bit

SGD Servo waveform

0 : Not displayed
1 : Displayed

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

3120	Time from the output of an alarm to the termination of sampling (waveform diagnosis function)

[Data type] Word

[Unit of data] ms

[Valid data range] 1 to 32760

This parameter specifies the time allowed from the time a servo alarm condition occurs until data collection and recording are stopped in the waveform diagnosis function; data collection can be continued additionally for the specified time.

Note

NOTE

1 Once the memory area becomes full, the oldest data is deleted to allow new data to be recorded (stored type).
2 Waveform diagnosis data is held in memory even after the power is turned off (stored type).
3 Be careful to set the date and time correctly.
4 To output stored type waveform data, the reader/punch interface option is required.
5 Waveform diagnosis is enabled when bit 0 (SGD) of parameter No. 3112 is set to 1. To perform waveform diagnosis, a graphics card is required.
Usual graphic display function can not be used when waveform diagnosis function is enable.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	MAINTENANCE MANUAL (B–63005EN)	1.10	Waveform diagnostic display
Series 21*i*/210*i*	MAINTENANCE MANUAL (B–63085EN)	1.10	Waveform diagnostic display

12.1.8 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 CNC 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 CNC 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 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 CNC can be displayed on the screen.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.7.3	CHECKING BY SELF–DIAGNOSTIC SCREEN

12.1.9 Display of Hardware and Software Configuration

General

The required hardware/software configuration for CNC maintenance can be displayed on the screen.

The system configuration screen displays the following information:

(1) Printed circuit board configuration

The type and function of the printed circuit board mounted in each slot are displayed. When a CPU is mounted on a printed circuit board, the software series and edition are also displayed.

(2) Software configuration

The series and editions of installed software, including the CNC software, servo software, PMC management software, and ladder programs, are displayed. For the CNC software, assembly information is also displayed.

(3) Module configuration

The configuration of the modules or hardware mounted on each printed circuit board (such as the type of a module or the hardware, and whether it is mounted) is displayed.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	MAINTENANCE MANUAL (B–63005EN)	1.3	System Configuration Screen
Series 21*i*/210*i*	MAINTENANCE MANUAL (B–63085EN)	1.3	System Configuration Screen

12.1.10 Position Display Neglect

General

Indication of the current position can be suppressed by setting bit 0 (NDPx) of parameter No. 3115, or by issuing the position indication ignore signal.

Bit 1 (NDAx) of parameter No. 3115 enables the display of positions in the machine coordinate system only.

Signal

Position Display Neglect Signal NPOS1~NPOS8<G198>

[Classification] Input signal

[Function] Disables the display of the current position.
A separate signal is provided for each controlled axis. The number at the end of each signal name denotes the controlled axis number.

NPOS1

- 1···1st axis current position display
- 2···2nd axis current position display
- 3···3rd axis current position display
- 4···4th axis current position display
- 5···5th axis current position display
- 6···6th axis current position display
- 7···7th axis current position display
- 8···8th axis current position display

[Operation] While a signal is set to 0, the current position in the corresponding axis is displayed. While the signal is set to 1, the current position along the corresponding axis is not displayed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G198	NPOS8	NPOS7	NPOS6	NPOS5	NPOS4	NPOS3	NPOS2	NPOS1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3115							NDAx	NDPx

[Data type] Bit axis

NDPx Display of the current position for each axis

0 : The current position is displayed.
1 : The current position is not displayed.

NDAx Position display using absolute coordinates and relative coordinates is:

0 : Performed.
1 : Not performed. (Machine coordinates are displayed.)

12.1.11 Run Hour and Parts Count Display

General

This function displays the integrated power–on time, the integrated cycle operation time, the integrated cutting time and timer (started by an input signal from PMC) on the screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, 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 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

Target part count reached signal PRTSF<F062#7>

[Classification] Output signal

[Function] Reports to the PMC that the specified number of parts have been machined.

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

· Machining of the specified number of parts has been completed.

When 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.

General–purpose integrating meter start signal TMRON <G053#0>

[Classification] Input signal

[Function] The CNC has an integrating meter which is started by an input signal from the PMC, as well as integrating meters for counting the automatic operation time and counting cutting time. The count for this integrating meter can be displayed on the screen. The count can be preset by MDI operation.

[Operation] When the signal is set to 1, the integrating meter starts counting.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G053								TMRON

	#7	#6	#5	#4	#3	#2	#1	#0
F062	PRTSF							

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6700								PCM

[Data type] Bit

PCM M code that counts the total number of machined parts and the number of machined parts

0 : M02, or M30, or an M code specified by parameter No. 6710
1 : Only M code specified by parameter No. 6710

6710	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.

6711	Number of machined parts

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

The number of machined parts is counted (+1) together with the total number of machined parts when the M02, M30, or a M code specified by parameter No. 6710 is executed.

> **NOTE**
> When bit 0 (PCM) of parameter No. 6700 is set to 1, the number of parts is not counted with M02 and M30.

6712	Total number of machined parts

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One piece

[Valid data range] 0 to 99999999

This parameter sets the total number of machined parts.

The total number of machined parts is counted (+1) when M02, M30, or an M code specified by parameter No. 6710 is executed.

> **NOTE**
> When bit 0 (PCM) of parameter No. 6700 is set to 1, the number of parts is not counted with M02 and M30.

6713	Number of required parts

Setting entry is acceptable.

[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.

6750	Integrated value of power–on period

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of power–on period.

6751	Operation time (integrated value of time during automatic operation)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6752	Operation time (integrated value of time during automatic operation)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of time during automatic operation (neither stop nor hold time included).

The actual operation time is the sum of the values set in parameter Nos. 6751 and 6752.

6753	Integrated value of cutting time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 1 to 60000

6754	Integrated value of cutting time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of a cutting time that is performed in cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03).

The actual cutting time is the sum of the values set in parameter Nos. 6753 and 6754.

6755	Integrated value of general–purpose integrating meter drive signal (TMRON) ON time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6756	Integrated value of general–purpose integrating meter drive signal (TMRON) ON time

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the integrated value of a time while input signal TMRON from PMC is on.

The actual integrated time is the sum of the values set in parameter Nos. 6755 and 6756.

6757	Operation time (integrated value of one automatic operation time)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One ms

[Valid data range] 0 to 60000

6758	Operation time (integrated value of one automatic operation time)

Setting entry is acceptable.

[Data type] Two–word

[Unit of data] One minute

[Valid data range] 0 to 99999999

This parameter displays the one automatic operation drive time (neither stop nor hold state included). The actual operation time is the sum of the values set in parameter Nos. 6757 and 6758. The operation time is automatically preset to 0 during the power–on sequence and the cycle start from the reset state.

Reference item

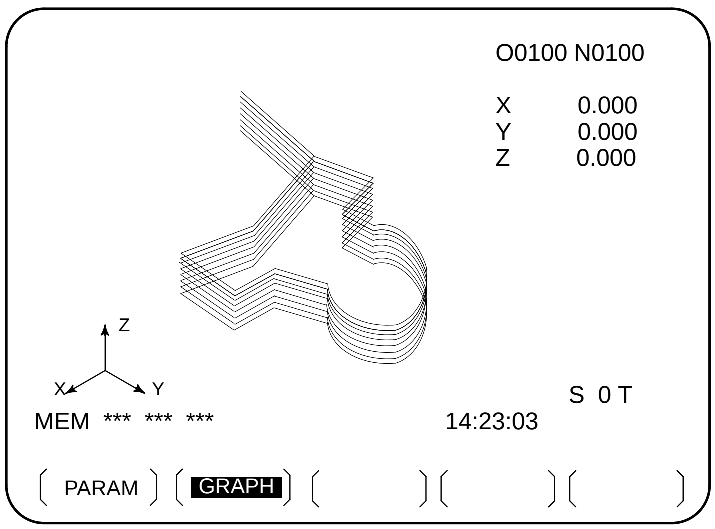
Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.4.5	Displaying and Setting Run Time,Parts Count, and Time
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.4.9	Displaying and Setting Run Time,Parts Count, and Time

12.1.12 Graphic Display/ Dynamic Graphic Display/Background Graphic

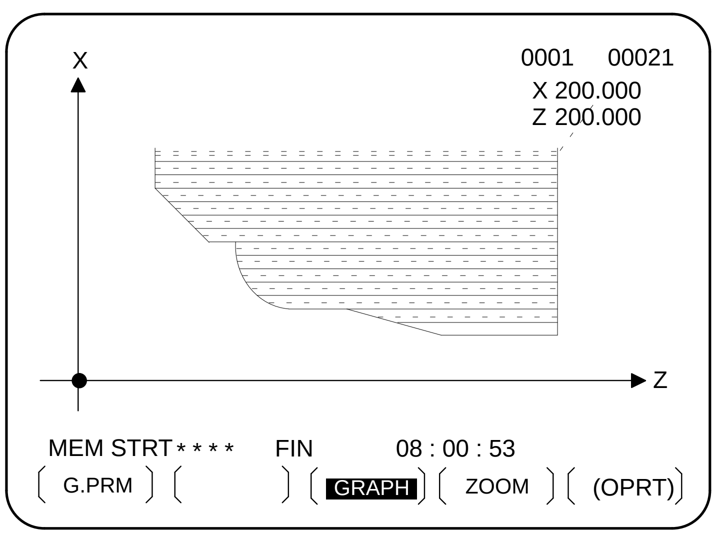
General

Graphic Display

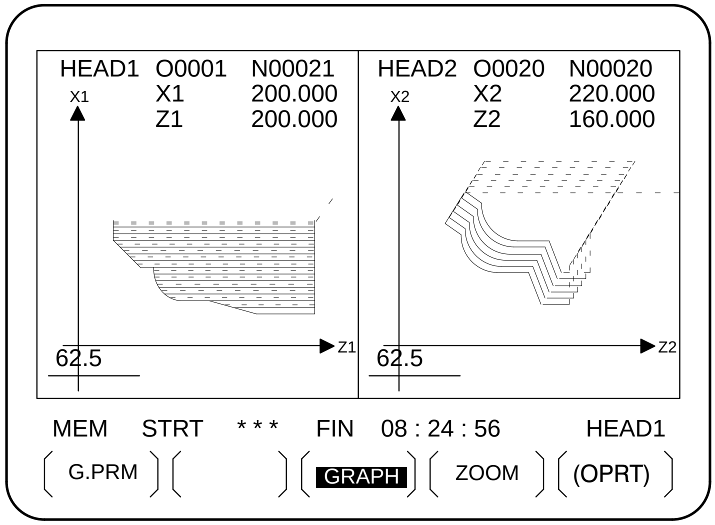
It is possible to draw the programmed tool path on the screen, which makes it possible to check the progress of machining, while observing the path on the screen.
In addition, it is also possible to enlarge/reduce the screen.
The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.
With T series (Two–path control) the tool paths of two tool posts are displayed on the same screen, one on the right and the other on the left.



M series



T series



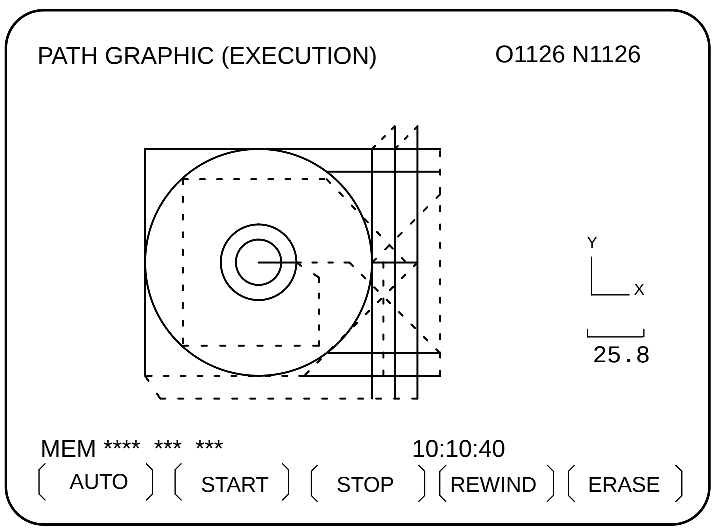
T series (Two–path control)

Dynamic graphic display (M series)

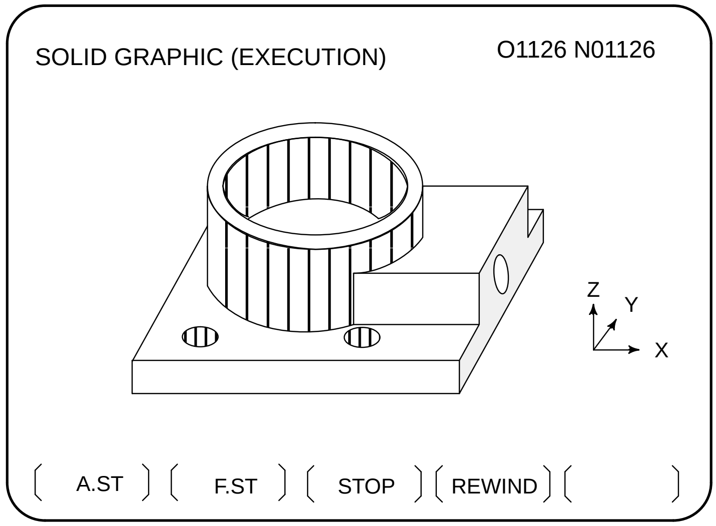
There are the following two functions in Dynamic Graphics.

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

Background graphic (M series)

The background graphic function allows a programmed tool path to be drawn on the screen while machining is being performed by another program.
When the background graphic mode is set, the previously selected program remains selected. The tool path of a desired program can be drawn on the background graphic screen.
The parameters used in actual machining are also used in background graphic mode. Setting and drawing with the background graphic function are the same as with the dynamic graphic display function.

Signal

Check drawing–under–way signal CKGRP <F062#5>

[Classification] Output signal

[Function] Indicates that a dynamic graphics display (animated graphics display) is being drawn for check purposes.

[Operation] The signal becomes 1 when:

- Check drawing begins.

The signal becomes 0 when:

- A reset occurs after check drawing has ended.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F062			CKGRP					

Parameter

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

[Data type] Bit

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

0 : Output
1 : Not output

	#7	#6	#5	#4	#3	#2	#1	#0
3109								
		BGO						

BGO Response when the <OFFSET> function key is pressed on the background graphic screen

0 : Returns the display to the machining screen.
1 : Displays the background graphic offsets, offsets in the workpiece coordinate system, and macro variables.
(In this case, ”BGGRP” appears in the bottom right corner of the screen. Using this screen, the background graphic data can be checked.)

	#7	#6	#5	#4	#3	#2	#1	#0
6500		NZM			DPA	GUL	SPC	GRL
			DPO					

[Data type] Bit

GRL Graphic display (two–path control lathe)

0 : Path 1 is displayed on the left, and path 2 is displayed on the right.
1 : Tool post 1 is displayed on the right, and tool post 2 is displayed on the left.

SPC Graphic display (two–path control lathe) is done

0 : on two spindles and two tool posts
1 : on one spindle and two tool posts

GUL 0 : The positions of X1– and X2–axes are not replaced with each other in the coordinate system specified with parameter 6509.
1 : The positions of X1– and X2–axes are replaced with each other in the coordinate system specified with parameter 6509. (2–path control)

NOTE
This parameter is used for two–path control lathe.

DPA Current position display on the graphic display screen

0 : Displays the actual position to ensure tool nose radius compensation
1 : Displays the programmed position

DPO Current position on the solid drawing (machining profile drawing) or tool path drawing screen

0 : Not appear
1 : Appears
When the background graphic function is used, modal information items F, S, and T are displayed, together with the current position. When the [POS] soft key is selected in dynamic graphic display mode, however, F, S, and T are not displayed.

NZM 0 : The screen image is not enlarged by specifying the center of the screen and magnification. (Screen image enlargement by a conventional method is enabled.)
1 : The screen image is enlarged by specifying the center of the screen and magnification. (Screen image enlargement by the conventional method is disabled.)

	#7	#6	#5	#4	#3	#2	#1	#0
6501			CSR					
			CSR	FIM	RID	3PL	TLC	ORG

[Data type] Bit

ORG Movement when coordinate system is altered during drawing
0 : Draws in the same coordinate system
1 : Draws in the new coordinate system (only for the path drawing)

TCL In solid drawing
0 : Not compensate the tool length
1 : Compensates the tool length

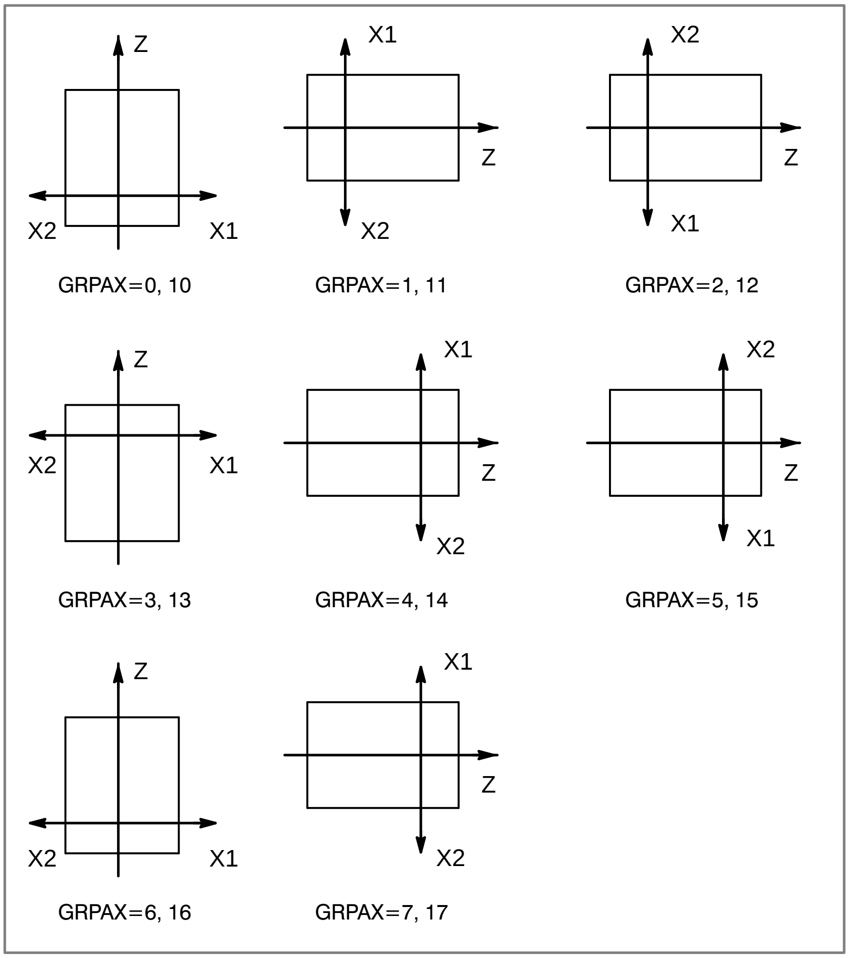
3PL Tri–plane drawing in solid drawing
0 : Drawn by the first angle projection
1 : Drawn by the third angle projection

RID In solid drawing
0 : Draws a plane without edges.
1 : Draws a plane with edges.

FIM Machining profile drawing in solid drawing
0 : Displayed in the coarse mode
1 : Displayed in the fine mode

CSR While the screen image is enlarged, the shape of the graphic cursor is:
0 : A square. (■)
1 : An X. (×)

6509	Coordinate system for drawing a single spindle (2–path control)



[Data type] Byte

[Valid data range] 0 to 7 and 10 to 17 (However, 0 to 7 are the same settings as 10 to 17.)

This parameter sets the coordinate system for drawing a single spindle (bit 1 of parameter 6500 = 1) for 2–path control.

The following shows the relationship between the settings and the drawing coordinate systems:

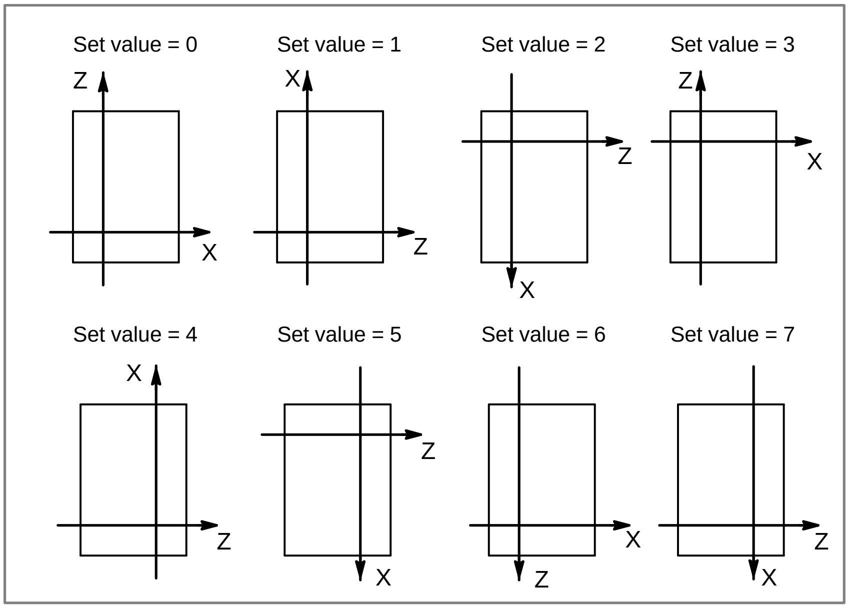
6510	Drawing coordinate system

[Data type] Byte

[Valid data range] 0 to 7

This parameter specifies the drawing coordinate system for the graphic function.

The following show the relationship between the set values and the drawing coordinate systems.



NOTE

This parameter is specified for each path in the two path control. A different drawing coordinate system can be selected for each tool post.

6511	
	Right margin in solid drawing

6512	
	Left margin in solid drawing

6513	
	Upper margin in solid drawing

6514	
	Lower margin in solid drawing

[Data type] Word

[Unit of data] Dot

These parameters set the machining profile drawing position in margins on the screen. The unit is a dot.

Parameter No.	Margin area	Standard set value			
		DPO=0		DPO=1	
		7.2″/8.4″ LCD	9.5″/10.4″ LCD	7.2″/8.4″ LCD	9.5″/10.4″ LCD
6511	Right	0	0	200	100
6512	Left	0	0	0	0
6513	Upper	25	32	25	32
6514	Lower	0	10	0	10

Set DPO with parameter No. 6500#5.

6515	
	Change in cross–section position in tri–plane drawing

[Data type] Byte type

[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.

	#7	#6	#5	#4	#3	#2	#1	#0
8100								
	NWP							RST

[Data type] Bit type

RST When the reset key on the MDI panel is pressed:

0 : The reset is effective for both paths. Alternatively, the reset is effective for both the machining side and the background graphic side (M series).

1 : The reset key is effective only for the path selected with the path select signal. Alternatively, the reset key cannot be used to stop machining in background graphic mode (M series).

NWP When the background graphic function is being used, this bit must be set to 1.

Note

NOTE
When the dynamic graphics function is used, the graphics function cannot be used. (M series)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY
		III.12.3	BACKGROUND GRAPHIC
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.12.1	GRAPHICS FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.12.1	GRAPHICS FUNCTION
		III.12.2	DYNAMIC GRAPHIC DISPLAY
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.12.1	GRAPHICS FUNCTION

12.1.13 Displaying Operating Monitor

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 up to three servo axes by setting parameters 3151 to 3153.

- **Display of the spindle axes**

When serial spindles are used, the reading on the load meter and speedmeter can be displayed only for the main serial spindle.

- **Speedmeter**

Although the speedmeter normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3111		OPS	OPM					

[Data type] Bit

OPM Operating monitor

0 : Not displayed
1 : Displayed

OPS The speedmeter on the operating monitor screen indicates:

0 : Spindle motor speed
1 : Spindle speed

3151	Number of the axis for which the first load meter for the servo motor is used
3152	Number of the axis for which the second load meter for the servo motor is used
3153	Number of the axis for which the third load meter for the servo motor is used
3154	Number of the axis for which the 4th load meter for the servo motor is used
3155	Number of the axis for which the 5th load meter for the servo motor is used
3156	Number of the axis for which the 6th load meter for the servo motor is used
3157	Number of the axis for which the 7th load meter for the servo motor is used
3158	Number of the axis for which the 8th load meter for the servo motor is used

[Data type] Byte

[Valid data range] 0, 1, . . . , the number of control axes

Set the numbers of the axes for which measurement values on the load meters for maximum eight servo motors are displayed. Set 0 for those axes for which a load meter need not be displayed.

2086	Rated current parameter (RTCURR)

[Data type] Word axis

4127	Load meter displayed value for maximum output

[Data type] Word axis

Note

> **NOTE**
> The reading on the load meter depends on servo parameter 2086 and spindle parameter 4127.
> These parameters are set by the automatic setting.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.1.8	Operating Monitor Display
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.1.8	Operating Monitor Display
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.1.7	Operating Monitor Display
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.1.7	Operating Monitor Display

12.1.14 Stamping the Machining Time

General

When a program is executed, its main program machining time is displayed on the program machining time display screen. The machining time can be displayed, in hours, minutes, and seconds format, for up to 10 main programs. Upon the execution of the eleventh and subsequent programs, previous data is cleared, starting from the oldest.

The time between the first start operation after a reset being performed in memory operation mode, until another reset is performed, is counted. When no reset is performed during operation, the time from the start of operation until M02 (M30) is encountered is counted. While the duration of the operation stop state is not being counted, the duration of the wait for completion of an M, S, T, or B function is counted.

The displayed machining time can be inserted (stamped) as a comment for a program stored in memory. The machining time is placed as a comment after the program number.

The machining time inserted after a program number can be displayed on the program directory screen (by setting bit 0 (NAM) of parameter No. 3107 to 1). Using this screen, the user can determine the machining time required for each program. This data is useful for on–site process planning.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3107								NAM

[Data type] Bit

NAM Program list

0 : Only program numbers are displayed.
1 : Program numbers and program names are displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3404			M02					

[Data type] Bit

M02 When M02 is specified in memory operation

0 : M02 is sent to the machine, and the program is positioned at its beginning automatically. As completion signal FIN is returned without resetting or without resetting and rewinding, the program is restarted from the beginning.
1 : M02 is only sent to the machine. The program is positioned at its beginning by the reset and rewind signal.

Note

NOTE
When M02 does not reset the control unit, and completion signal FIN is sent to continuously reexecute the program from the beginning (when bit 5 (M02) of parameter No. 3404 is set to 0), counting of machining time is terminated by completion signal FIN.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.11.2.6	Stamping the machining time
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.11.2.6	Stamping the machining time

12.1.15 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 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.7200) can be used to select whether the control switches on the machine operator's panel or soft switches on the MDI of the control unit are used 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 override, feedrate override, rapid traverse override

Group5 :Optional block skip, single block, machine lock, dry run

Group6 :Protect key

Group7 :Feed hold

Group8 :General purpose

The states of all soft switches are 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 <F073#0> MD2O <F073#1> MD4O <F073#2> ZRNO <F073#4>	MD1 MD2 MD4 ZRN
2	Jog feed axis select	+J10 – +J40 –J10 – –J40 <F081>	+J1 – +J4 –J1 – –J4
	Manual rapid traverse	RTO <F077#6>	RT
3	Handle feed	HS1AO <F077#0> HS1BO <F077#1> HS1CO <F077#2> HS1DO <F077#3>	HS1A HS1B HS1C HS1D
	Handle feed magnification	MP1O <F076#0> MP2O <F076#1>	MP1 MP2
4	Jog feed rate override	*JV0O – *JV15O <F079, F080>	*JV0 – *JV15
	Feedrate override	*FV0O – *FV7O <F078>	*FV0 – *FV7
	Rapid traverse override	ROV1O <F076#4> ROV2O <F076#5>	ROV1 ROV2
5	Optional block skip	BDTO <F075#2>	BDT
	Single block	SBKO <F075#3>	SBK
	Machine lock	MLKO <F075#4>	MLK
	Dryrun	DRNO <F075#5>	DRN
6	Protect key	KEYO <F075#6>	KEY1 – KEY4
7	Feed hold	SPO <F075#7>	*SP
8	General purpose (Switch from 1st line to the 8th line)	OUT0 – OUT7 <F072>	

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O			MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	:JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	–J4O	+J4O	–J3O	+J3O	–J2O	+J2O	–J1O	+J1O

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
7200		OP7	OP6	OP5	OP4	OP3	OP2	OP1

[Data type] Bit

OP1 Mode selection on software operator’s panel

0 : Not performed
1 : Performed

OP2 JOG feed axis select and manual 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 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

7210	Jog–movement axis and its direction on software operator's panel "↑"
7211	Jog–movement axis and its direction on software operator's panel "↓"
7212	Jog–movement axis and its direction on software operator's panel "→"
7213	Jog–movement axis and its direction on software operator's panel "←"
7214	Jog–movement axis and its direction on software operator's panel "↙"
7215	Jog–movement axis and its direction on software operator's panel "↗"
7216	Jog–movement axis and its direction on software operator's panel "⤴"
7217	Jog–movement axis and its direction on software operator's panel "↷"

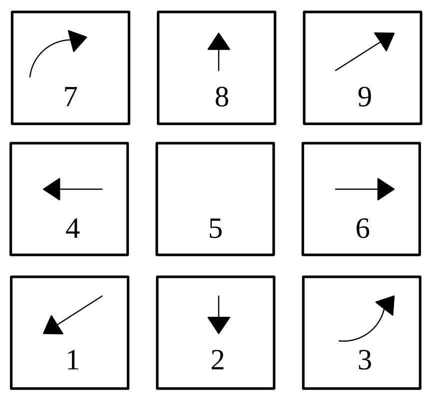
[Data type] Byte

[Valid data range] 0 to 8

On software operator's panel, set a feed axis corresponding to an arrow key on the MDI panel when jog feed is performed.

Set value	Feed axis and direction
0	Not moved
1	First axis, positive direction
2	First axis, negative direction
3	Second axis, positive direction
4	Second axis, negative direction
5	Third axis, positive direction
6	Third axis, negative direction
7	Fourth axis, positive direction
8	Fourth axis, negative direction

Arrow keys on the MDI panel



Example

Under X, Y, and Z axis configuration, to set arrow keys to feed the axes in the direction specified as follows, set the parameters to the values given below. [8↑] to the positive direction of the Z axis, [2↓] to the negative direction of the Z axis, [6→] to the positive direction of the X axis [4←] to the negative direction of the X axis, [1↙] to the positive direction of the Y axis, [9↗] to the negative direction of the Y axis

Parameter No. 7210 = 5 (Z axis, positive direction)
Parameter No. 7211 = 6 (Z axis, negative direction)
Parameter No. 7212 = 1 (X axis, positive direction)
Parameter No. 7213 = 2 (X axis, negative direction)
Parameter No. 7214 = 3 (Y axis, positive direction)
Parameter No. 7215 = 4 (Y axis, negative direction)
Parameter No. 7216 = 0 (Not used)
Parameter No. 7217 = 0 (Not used)

7220	Name of general–purpose switch on software operator’s panel
⋮	⋮
7283	Name of general–purpose switch on software operator’s panel

[Data type] Byte

Example

These parameters set the names of the general–purpose switches (SIGNAL 1 through SIGNAL 8) on the software operator’s panel 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. 7220 to 7283.

Parameter No. 7220:

Sets the character code (083) corresponding to S of SIGNAL 1.

Parameter No. 7221:

Sets the character code (073) corresponding to I of SIGNAL 1.

Parameter No. 7222:

Sets the character code (071) corresponding to G of SIGNAL 1.

Parameter No. 7223:

Sets the character code (078) corresponding to N of SIGNAL 1.

Parameter No. 7224:

Sets the character code (065) corresponding to A of SIGNAL 1.

Parameter No. 7225:

Sets the character code (076) corresponding to L of SIGNAL 1.

Parameter No. 7226:

Sets the character code (032) corresponding to (space) of SIGNAL 1.

Parameter No. 7227:

Sets the character code (049) corresponding to 1 of SIGNAL 1.

Parameter Nos. 7228 to 7235:

Set the character codes of SIGNAL 2 shown in the figure above.

Parameter Nos. 7236 to 7243:

Set the character codes of SIGNAL 3 shown in the figure above.

Parameter Nos. 7244 to 7251:

Set the character codes of SIGNAL 4 shown in the figure above.

Parameter Nos. 7252 to 7259:

Set the character codes of SIGNAL 5 shown in the figure above.

Parameter Nos. 7260 to 7267:

Set the character codes of SIGNAL 6 shown in the figure above.

Parameter Nos. 7268 to 7275:

Set the character codes of SIGNAL 7 shown in the figure above.

Parameter Nos. 7276 to 7283:

Set the character codes of SIGNAL 8 shown in the figure above.

The character codes are shown in character code list on the following page.

Character to Code Correspondence Table

Char-acter	Code	Com-ment	Char-acter	Code	Comment	Char-acter	Code	Com-ment	Char-acter	Code	Comment
A	065		6	054		ア	177		ム	209	
B	066		7	055		イ	178		メ	210	
C	067		8	056		ウ	179		モ	211	
D	068		9	057		エ	180		ヤ	212	
E	069			032	Space	オ	181		ユ	213	
F	070		!	033	Exclamation mark	カ	182		ヨ	214	
G	071		”	034	Quotation marks	キ	183		ラ	215	
H	072		#	035	Number	ク	184		リ	216	
I	073		$	036	Dollar mark	ケ	185		ル	217	
J	074		%	037	Percent	コ	186		レ	218	
K	075		&	038	Ampersand	サ	187		ロ	219	
L	076		’	039	Apostrophe	シ	188		ワ	220	
M	077		(	040	Left parenthesis	ス	189		ヲ	166	
N	078		)	041	Right parenthesis	セ	190		ン	221	
O	079		*	042	Asterisk	ソ	191		ァ	167	
P	080		+	043	Positive sign	タ	192		ィ	168	
Q	081		,	044	Comma	チ	193		ゥ	169	
R	082		–	045	Negative sign	ツ	194		ェ	170	
S	083		.	046	Period	テ	195		ォ	171	
T	084		/	047	Slash	ト	196		ャ	172	
U	085		:	058	Colon	ナ	197		ュ	173	
V	086		;	059	Semicolon	ニ	198		ョ	174	
W	087		<	060	Left angle bracket	ヌ	199		ッ	175	
X	088		=	061	Equal sign	ネ	200		゛	222	Dakuten
Y	089		>	062	Right angle bracket	ノ	201		゜	223	Han dakuten
Z	090		?	063	Question mark	ハ	202		。	161	Full stop
0	048		@	064	Commercial at mark	ヒ	203		「	162	Left quotation mark
1	049		[	091	Left square bracket	フ	204		」	163	Right quotation mark
2	050		¥	092	Yen mark	ヘ	205		、	164	Comma
3	051		]	093	Right square bracket	ホ	206		・	165	Centered dot
4	052		^	094		マ	207			000	Space
5	053		_	095	Underline	ミ	208				

NOTE

The “dakuten” and “han dakuten” in Katakana also correspond to one character.

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 Only one soft switch is available for protection key. However, four input signals are available for protection key (KEY1, KEY2, KEY3 and KEY4). Generally, four input signals are simultaneously turned to "1" or "0" according to the state of the soft switch for protection keys.

3 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".

NOTE

4 The following table lists the override values which can be selected by soft switches for jog feedrate.

	*JV00 – *JV150 (*JV0 – *JV150) 15 12 8 4 0 bit				Override values (%)
0	1 1 1 1	1 1 1 1	1 1 1 1	1 1 1 1	0
1	1 1 1 1	1 1 1 1	1 1 1 1	0 1 0 1	0.1
2	1 1 1 1	1 1 1 1	1 1 1 1	0 0 0 1	0.14
3	1 1 1 1	1 1 1 1	1 1 1 0	1 0 1 1	0.2
4	1 1 1 1	1 1 1 1	1 1 1 0	0 1 0 0	0.27
5	1 1 1 1	1 1 1 1	1 1 0 1	1 0 1 0	0.37
6	1 1 1 1	1 1 1 1	1 1 0 0	1 0 1 1	0.52
7	1 1 1 1	1 1 1 1	1 0 1 1	0 1 1 1	0.72
8	1 1 1 1	1 1 1 1	1 0 0 1	1 0 1 1	1.0
9	1 1 1 1	1 1 1 1	0 1 1 1	0 0 1 1	1.4
10	1 1 1 1	1 1 1 1	0 0 1 1	0 1 1 1	2.0
11	1 1 1 1	1 1 1 0	1 1 1 1	0 0 0 1	2.7
12	1 1 1 1	1 1 1 0	1 0 0 0	1 1 0 1	3.7
13	1 1 1 1	1 1 0 1	1 1 1 1	0 1 1 1	5.2
14	1 1 1 1	1 1 0 1	0 0 1 0	1 1 1 1	7.2
15	1 1 1 1	1 1 0 0	0 0 0 1	0 1 1 1	10.0
16	1 1 1 1	1 0 1 0	1 0 0 0	0 1 1 1	14.0
17	1 1 1 1	1 0 0 0	0 0 1 0	1 1 1 1	20.0
18	1 1 1 1	0 1 0 1	0 1 1 1	0 0 1 1	27.0
19	1 1 1 1	0 0 0 1	1 0 0 0	1 0 1 1	37.0
20	1 1 1 0	1 0 1 1	1 0 1 0	1 1 1 1	52.0
21	1 1 1 0	0 0 1 1	1 1 0 1	1 1 1 1	72.0
22	1 1 0 1	1 0 0 0	1 1 1 0	1 1 1 1	100.0
23	1 1 0 0	1 0 0 1	0 1 0 0	1 1 1 1	140.0
24	1 0 1 1	0 0 0 1	1 1 0 1	1 1 1 1	200.0

NOTE

5 The following table lists the override values which can be selected by soft switches for feedrate override.

	*FV0O – *FV7O (*FV0 – *FV7) 7 4 0 ↓ ↓ ↓	Override values (%
0	1 1 1 1 1 1 1 1	0
1	1 1 1 1 0 1 0 1	10
2	1 1 1 0 1 0 1 1	20
3	1 1 1 0 0 0 0 1	30
4	1 1 0 1 0 1 1 1	40
5	1 1 0 0 1 1 0 1	50
6	1 1 0 0 0 0 1 1	60
7	1 0 1 1 1 0 0 1	70
8	1 0 1 0 1 1 1 1	80
9	1 0 1 0 0 1 0 1	90
10	1 0 0 1 1 0 1 1	100
11	1 0 0 1 0 0 0 1	110
12	1 0 0 0 0 1 1 1	120
13	0 1 1 1 1 1 0 1	130
14	0 1 1 1 0 0 1 1	140
15	0 1 1 0 1 0 0 1	150
16	0 1 0 1 1 1 1 1	160
17	0 1 0 1 0 1 0 1	170
18	0 1 0 0 1 0 1 1	180
19	0 1 0 0 0 0 0 1	190
20	0 0 1 0 0 1 1 1	200

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.10	Displaying and Setting the Software Operator's Panel
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.13	Displaying and Setting the Software Operator's Panel
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.4.10	Displaying and Setting the Software Operator's Panel
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.4.13	Displaying and Setting the Software Operator's Panel

12.1.16 Multi–language Display

General

The LCD screens are displayed in a parameter–set language.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3102		SPN	HNG	ITA	CHI	FRN	GRM	JPN

	#7	#6	#5	#4	#3	#2	#1	#0
3119							POR	DAN
							POR	

[Data type] Bit type

> **NOTE**
> When this parameter is set, turn off the power once.

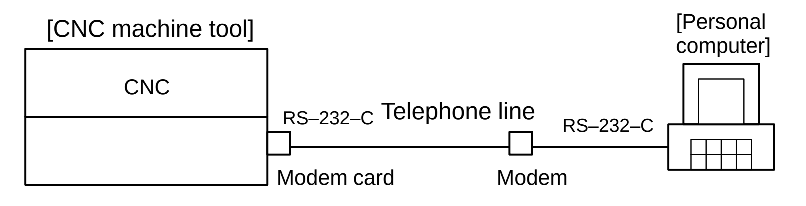
These bits select the language to be used for the display.

POR	DAN	SPN	HNG	ITA	CHI	FRN	GRM	JPN	language
0	0	0	0	0	0	0	0	0	English
0	0	0	0	0	0	0	0	1	Japanese
0	0	0	0	0	0	0	1	0	German
0	0	0	0	0	0	1	0	0	French
0	0	0	0	0	1	0	0	0	Chinese (Taiwanese)
0	0	0	0	1	0	0	0	0	Italian
0	0	0	1	0	0	0	0	0	Korean
0	0	1	0	0	0	0	0	0	Spanish
0	1	0	0	0	0	0	0	0	Danish
1	0	0	0	0	0	0	0	0	Portuguese

12.1.17 Remote Diagnosis

General

The remote diagnosis function allows CNC status monitoring and modification to CNC data to be performed remotely by menu–based operation. The remote diagnosis function, operating under MS–DOS, is installed on a standard personal computer, connected as a service terminal to the CNC via the RS–232–C interface, over a telephone line, and so on.



The remote diagnosis terminal software is sold separately.

The remote diagnosis function provides the following capabilities:

a. CNC programs

a–1 Computer → CNC

(1) CNC command data for verification
(2) Searching for a specified program
(3) Part program
(4) Deleting a specified program
(5) Deleting all programs

a–2 CNC → computer

(1) Part program
(2) Displaying a program directory
(3) Program number of a program being executed
(4) Sequence number of a sequence being executed

b. Computer → CNC

(1) Parameter
(2) Pitch error data
(3) Tool offset value
(4) Custom macro variable
(5) Selecting a display screen
(6) Memory contents
(7) PMC data
(8) Displaying a specified message
(9) All parameters

c. CNC → computer
   (1) Alarm information
   (2) Machine position
   (3) Absolute position
   (4) Skip position
   (5) Servo delay
   (6) Acceleration/deceleration delay
   (7) Diagnosis
   (8) Parameter
   (9) Tool life management data
   (10) Display screen status
   (11) Modal information
   (12) Pitch error data
   (13) Tool offset value
   (14) Custom macro variable
   (15) Memory contents
   (16) Ladder program
   (17) Actual feedrate
   (18) Status
   (19) A/D conversion data
   (20) PMC data
   (21) Screen character data
   (22) Printed circuit board information
   (23) Ladder title
   (24) Series and edition of PMC/ladder
   (25) All parameters

d. File function selection
   (1) Listing files
   (2) Referencing a file
   (3) Deleting a file
   (4) Copying a file
   (5) Renaming a file
   (6) Linking a file
   (7) Changing the current directory
   (8) Creating a directory
   (9) Deleting a directory

NOTE
An arrow "→" indicates the direction of data flow.

12.1.18 External Operator Message Logging and Display

General

External operator messages can be logged in a history file.

These messages can be displayed on the external operator message history screen.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3112						OMH		

[Data type] Bit

OMH The external operator message history screen is:

0 : Not displayed.
1 : Displayed.

	#7	#6	#5	#4	#3	#2	#1	#0
3113	MS1	MS0						MHC

MHC External operator message history data:

0 : Cannot be cleared.
1 : Can be cleared.
(Such data can be cleared using the [CLEAR] soft key.)

MS0, MS1 A combination of the number of characters preserved as external operator message history data and the number of history data items is set according to the table below.

MS1	MS0	Number of history data characters	Number of history data items
0	0	255	8
0	1	200	10
1	0	100	18
1	1	50	32

CAUTION
When the values of MS0 and MS1 are changed, all preserved external operator message history data is cleared.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.7.1	External operator message logging and display
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.7.1	External operator message logging and display
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.7.1	External operator message logging and display
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.7.1	External operator message logging and display

12.1.19 Erase Screen Display/Automatic Erase Screen Display

General

Displaying the same characters in the same positions on the screen causes a LCD to degrade relatively quickly. To help prevent this, the screen can be cleared by pressing specific keys. It is also possible to specify the automatic clearing of the screen if no keys are pressed during a period specified with a parameter.

Erase screen display

Holding down the CAN key and pressing an arbitrary function key clears the screen.

Hold down the CAN key and press an arbitrary function key (such as POS and PROG).

Press an arbitrary function key, when the screen is displayed again.

Automatic erase screen display

The CNC screen is automatically cleared if no keys are pressed during the period (in minutes) specified with a parameter (No. 3123). The screen is restored by pressing any key.

- **Clearing the screen with automatic erase screen display**

The CNC screen is cleared once the period (minutes) specified with parameter No. 3123 has elapsed, provided the following conditions are satisfied:

Conditions for clearing the CNC screen

- Automatic erase screen display cacel signal *CRTOF is “0”.
- Parameter No. 3123 is set to other than 0.
- None of the following keys have been pressed:
  MDI keys
  Soft keys
  External input keys
- No alarm has been issued.

- **Restoring the screen with automatic erase screen display**

The cleared CNC screen is restored once at least one of the following conditions is satisfied:

Conditions for restoring the CNC screen

- Automatic erase screen display cancel signal *CRTOF is “1”.
- Any of the following keys has been pressed:
  MDI keys
  Soft keys
  External input keys
- An alarm has been issued.

- **Automatic erase screen display cancel signal**

Automatic erase screen display cancel signal *CRTOF (G062#1) is valid only for the signals for path 1. This signal is invalid for the signals of path 2 (G1062#1) and those of the loader (G062#1).

- **Clearing the screen using CAN + function key**

If parameter No. 3123 is set to 0, clearing of the screen using the CAN key and a function key is disabled.

- **Specified period**

The period specified with parameter No. 3123 is valid only for path 1.

- **Alarm for another path**

The screen is not cleared if an alarm is issued for path 1 or 2 or the loader before the specified period elapses.

WARNING
Pressing any key while the screen is being cleared restores the screen. In such a case, however, the function assigned to the pressed key is initiated. Do not press the DELETE, INSERT, or ALTER key to restore the screen, therefore.

Signal

Automatic erase screen display cancel signal *CRTOF <G062#1>

[Classification] Input signal

[Function] Enables or disables the automatic screen erase function. This signal is used to switch the control mode.

[Operation] When this signal is set to 0, the control unit:

· Enables the automatic screen erase function.

When this signal is set to 1, the control unit:

· Disables the automatic screen erase function, displays the screen, and initializes the timer.

This signal is valid only for path 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G062							*CRTOF	

Parameter

3123	Time until erase screen display is applied

[Data type] Bytes

[Unit of data] Minutes

[Valid data range] 1 to 255

This parameter specifies the period that must elapse before the erase screen display is applied. If 0 is set, the screen is not cleared.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.8	Cleaning the screen
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.8	Cleaning the screen
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.8	Cleaning the screen
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.8	Cleaning the screen

12.1.20 Touch Pad

General

A touch pad can be used on the LCD display, as follows:

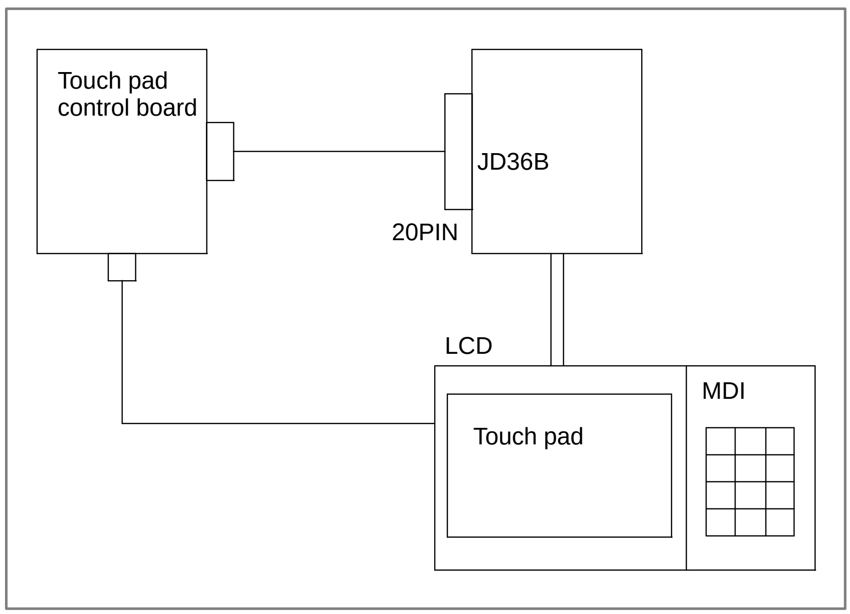
(1) The soft keys (F0 to F9, FR, and FL) in the lower section on the 10.4–inch color LCD/MDI panel are changed to those for the touch pad.

(2) Touch pad operation substitutes for cursor control on the 10.4–inch color LCD.

(3) A touch pad can be used to create C executor–based applications such as touch pad–based software versions of a machine operator's panel and a calculator.

Hardware connection

The CNC uses RS–232C serial port 2 (JD36B) on the motherboard to connect the touch pad.



Hardware characteristics

The hardware characteristics of the touch pad are briefly described below for reference purposes. The user need not be aware of them when using the touch pad.

When a point on the touch pad is pressed, the touch pad controller sends the coordinates of the point to the CNC via an RS–232C interface.

(1) Coordinate system

1) Initial coordinate system
The upper left corner of the input area of the touch pad is set as the origin (X = 0, Y = 0) of the coordinate system. The maximum coordinates of the lower right corner is (639, 479).

2) Coordinate system after compensation
The coordinate system can be subjected to compensation as described later, so the initial origin and maximum coordinates can be maintained.

(2) Reception data sampling interval
The coordinates of a point on the touch pad are output via an RS–232C interface. The information is received at sampling intervals of 50 ms.

(3) Positional precision
A positional precision of +2.5 mm can be maintained by performing 9–point compensation as described later.

Software key control

When a rectangular soft key button on the screen is pressed with a finger, the soft key button display is indented. This type of a soft key behaves in the same manner as conventional soft keys. In addition, the touch pad soft keys can be customized using the C executor.

Cursor control

When the cursor is displayed and can be moved (for data setting) on the 10.4–inch LCD panel with a touch pad, pressing the desired point on the screen causes the cursor to move to that point, in the same way as with the ↑, ↓, →, and ← cursor keys.

C executor

Touch pad–based application programs can be created using the C executor, which provides touch pad functions.
Refer to the C Executor Programming Manual (B–62443JA–3) for details of the C executor.

Compensation

(1) Functions
Compensation functions as follows:
1) Specifies an effective input area.
2) Specifies maximum coordinates.
3) Corrects linearity.
4) Makes the panel input position match the LCD display position.

(2) Execution time
Compensation must be executed when:
1) A new touch pad is installed.
2) An existing touch pad is replaced.
3) Memory (SRAM) is cleared to all 0s.

(3) Cautions
1) If an alarm condition occurs during compensation, the alarm screen appears when the compensation screen is exited.
2) Compensation can be made regardless of what the current mode is.
3) Once the system gets started, compensation should be made promptly before actual operations begin.

(4) Procedure
1 Enable touch pad compensation by setting parameter DCL (bit 5 of parameter No. 3113) to 1.
2 Press the SYSTEM function key.
3 Press the rightmost soft key, + (next menu page key) three times.

4 Press the [TP CAL] soft key.
The following touch pad compensation screen appears.

```
CALIBRATION OF TOUCH PANEL

  PLEASE PUSH CALIBRATED POINTS (+ OF 9 POINTS).
  IF CALIBRATION IS ENDED, PLEASE PUSH <INPUT> KEY.
  IF CALIBRATION IS CANCELED, PLEASE PUSH <CAN> KEY.
  IF OPERATION IS ENDED, PLEASE PUSH FUNCTION KEY.
```

5 Press the nine compensation points using the dedicated pen. When a point is pressed exactly, the corresponding “+” display starts blinking. If a point off a “+” display is pressed, the message [CALIBRATED POINT DOES NOT MATCH. PLEASE PUSH AGAIN.] appears.

6 After pressing the nine compensation points, press the INPUT to complete compensation. To discontinue or rerun compensation, press the CAN key.

7 When compensation ends normally, the message [CALIBRATION WAS ENDED.] appears.

8 Pressing another function key causes you to exit the compensation screen, enabling compensation to be terminated or canceled.

9 Once compensation was completed, disable touch pad compensation by resetting parameter DCL (bit 5 of parameter No. 3113) to 0 in order to prevent malfunction.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3113			DCL					

[Data type] Bit

DCL Specifies whether to enable compensation for the touch pad.
0 : Disabled.
1 : Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3119						DDS		

[Data type] Bit

DDS Specifies whether to enable touch pad operation.

0 : Enabled.
1 : Disabled.

As described in “Hardware connection,” the touch pad is connected to RS–232–C serial port 2 (JD36B) on the CNC motherboard. When the touch pad is used, serial port 2 (JD36B) is set up for touch pad operation, regardless of an I/O channel (input/output unit selection) set up in parameter Nos. 20, 21, 22, and 23. Use JD36A for other input/output units.

The existing settings of parameter Nos. 100, and 121 to 123 are disabled for channel 2 (JD36B), and fixed as follows:

- Baud rate: 9600 bps
- Stop bit: 1 bit
- Parity check: None

If no touch pad is used, parameter DDS (bit 2 of parameter No. 3119) must be set to 1.

Alarm and message

Number	Message	Description
085	COMMUNICATION ERROR	A communication error occurred during touch pad initialization. It is likely that the touch pad is not connected correctly or is defective. After removing the cause of the error, switch the power off then back on again. This alarm message is used also for errors in existing input/output units. If a conventional input/output unit is connected to RS–232C serial port 1 (JD36A), this alarm message may be displayed because of an error on serial port 1, regardless of the touch pad.

Caution

CAUTION

1 The touch pad soft keys cannot be used after memory (SRAM) is cleared to all 0s. In this case, it is necessary to use the MDI (cursor and page keys) for setup.

2 After the power is switched on, do not use the touch pad until a power–on reset sequence is completed. Otherwise, a P/S alarm (No. 085) will occur to indicate a communication error.

3 If more than one point on the touch pad is pressed simultaneously, it is assumed that only one point that represents a gravity position determined according to the way each point is pressed, is pressed. Be sure to press only one point at a time.

12.1.21 Periodic Maintenance Screen

General

The periodic maintenance screen shows the current statuses of those consumables that require periodic replacement (backup battery, LCD backlight, touch pad, etc.). An item whose service life has expired is indicated by the machine run time or the like.

Screen displays and settings

1 Press the SYSTEM function key.

2 Press the ▷ next menu page key several times until the [MAINTE] soft key appears.

3 Press the [MAINTE] soft key. The periodic maintenance screen appears.

The periodic maintenance screen consists of two pages: status screen and setup screen, either of which can be selected using the [CHANGE] soft key.

● Status screen displays and settings

On the status screen, it is possible to register up to 10 consumables names. The remaining lifetime and count status for each registered item are displayed on the screen.

```
PERIODICAL MAINTENANCE           O0001 N12345
   (STATUS)
        ITEM NAME                        REMAIN
  *01  BATTERY FOR CONTROLLER                0H
  @02  BATTERY FOR PULSECODER             5000H
   03  FAN MOTOR                     10000H
  @04  LCD BACK LIGHT                      720H
   05
   06
   07
   08
   09
   10
>_
EDIT *** ***** *** ****    19:27:05
[      ][ MAINTE ][       ][       ][ (OPRT) ]
```

```
[ CHANGE ][ ENTRY ][ CLEAR ][ +INPUT ][ INPUT ]
```

```
[       ][       ][       ][  CAN  ][  EXEC  ]
```

(1)Consumables names

The names of consumables to be subjected to periodic maintenance are set up here. They can be registered using either the corresponding menu or MDI keys.

1) Menu–based setup

1 To display the setting menu, place the cursor on the desired item, and press the [ENTRY] soft key. There are two types of setup menus: machine consumables menu and CNC consumables menu.

2 Pressing the [MACHIN] or [NC] soft key displays a menu that contains typical machine or CNC model names.

3 Place the cursor on the registered item and press the [SELECT] soft key. Now press the [EXEC] soft key to return to the status screen where the selected item name is set up.

4 Pressing the [CAN] soft key restores the previous screen.

5 Pressing the [MAINTE] soft key displays the status screen again.

On the machine consumables screen, item names can be registered according to the procedure below.

(a) Program–based registration

An item name can be registered by executing a program in the following format.

Format

G10 L61 Px [n]

P : Registration number

n : Item name
[Alphanumeric character]

(b) MDI–based registration

An item name can be registered by first keying in data in the following format, then pressing the [INPUT] soft key (or the INPUT key). In addition, pressing the [+INPUT] soft key can modify an existing item name.

NOTE

1 The asterisk (*) is used as a control code, so it cannot be used in the item name. Likewise, square brackets "[" and "]" and parentheses "(" and ")" must be excluded from the item name.

2 If a null item name is selected on the machine consumables screen, the warning message "EDIT REJECTED" will appear. If a null item name is selected on the NC consumables screen, a blank will be set up.

To delete a registered item name, place the cursor on it, and press the [CLEAR] soft key, then the [EXEC] soft key.

[Machine consumables screen]

```
PERIODICAL MAINTENANCE           O0001 N12345
   (MACHINE)
        ITEM NAME
   01
   02
   03
   04
   05
   06
   07
   08
   09
   10
>_
EDIT *** ***** *** ****    19:27:05
[      ][ STATUS ][ MACHIN ][  NC  ][ (OPRT) ]
```

```
[ SELECT ][      ][ CLEAR ][ +INPUT ][ INPUT ]
```

```
[       ][        ][       ][  CAN  ][ EXEC  ]
```

```
[       ][  READ  ][ PUNCH ][       ][       ]
```

```
[       ][        ][       ][  CAN  ][ EXEC  ]
```

[CNC consumables screen]

```
PERIODICAL MAINTENANCE          O0001 N12345
   (NC)
        ITEM NAME
   01  BATTERY FOR CONTROLLER
   02  BATTERY FOR PULSECODER
   03  FAN MOTOR
   04  LCD BACK LIGHT
   05
   06
   07
   08
   09
   10
>_
EDIT *** ***** *** ****    19:27:05
[      ][ STATUS ][ MACHIN ][  NC  ][ (OPRT) ]
```

```
[ SELECT ][       ][       ][       ][       ]
```

```
[        ][       ][       ][  CAN  ][  EXEC  ]
```

NOTE
On the CNC consumables screen, it is impossible to register, delete, input, and output item names.

2) MDI–based setup
An item name can be registered by first keying in data in the following format, then pressing the [INPUT] soft key (or the INPUT key). In addition, pressing the [+INPUT] soft key can modify an existing item name.

NOTE
The asterisk (*) is used as a control code, so it cannot be used in the item name. Likewise, square brackets “[” and “]” and parentheses “(” and “)” must be excluded from the item name.

To delete a registered item name, place the cursor on it, and press the [CLEAR] soft key, then the [EXEC] soft key.
When an item name is deleted, the related service life, remaining lifetime, and count type are also deleted.

(2) Remaining lifetime
The remaining lifetime of an item is the time allowed before it must be replaced. It is displayed in a count–down format. When the remaining lifetime becomes less than the percentage specified in parameter No. 8911 of the corresponding service life, the remaining lifetime display turns red. Count–down continues even after the service life has expired.

NOTE
No setup can be made on the status screen. Setup is possible only on the setting screen.

(3) Count status
The count status of each item is displayed at the left of the corresponding item number as listed below:

Display	Count status
Blank	Counting is at a halt.
@	Counting is under way.
*	Service life has expired.

- **Setting screen displays and settings**

On the setting screen, it is possible to specify the service life, remaining lifetime, and count type for each registered item. It also displays the same count status as displayed on the status screen.

```
PERIODICAL MAINTENANCE          O0001 N12345
   (SETTING)
        LIFE       REMAIN     COUNT TYPE
  *01   10000H         0H      ALL TIME
  @02   20000H      5000H      LIV TIME
   03   32767H     10000H      --------
  @04    1500H       720H      RUN TIME
   05
   06
   07
   08
   09
   10
>_
EDIT *** ***** *** ****    19:27:05
[ CHANGE ][ TYPE ][ CLEAR ][ +INPUT ][ INPUT ]
```

↓

```
[ EFFECT ][  ALL  ][  LIV  ][  RUN  ][  CUT  ]
```

```
[        ][  READ  ][ PUNCH ][       ][       ]
```

(1) Service life
The service life of consumables can be specified here. To specify the service life, key in the corresponding data and press the [INPUT] soft key (or INPUT key). The same data is set up as both the service life and remaining lifetime. In addition, the count type is indicated as: "______"

Pressing the [+INPUT] soft key causes newly entered data to be added to the existing service life and the remaining lifetime.

The valid data range for this item is: 0 to 65535 (hours)

NOTE
1 If an attempt is made to enter data for an item number for which no item name has been registered, the warning message "EDIT REJECTED" appears.
2 If an attempt is made to enter data that does not fall in the valid data range, the warning message "DATA IS OUT OF RANGE" appears.
3 If an attempt is made to enter data that would set the service life or remaining lifetime below 0, they will be clamped to 0.
4 If the [CLEAR] or [TYPE] soft key is pressed, the warning message "EDIT REJECTED" appears.

(2) Remaining lifetime
The remaining lifetime of an item is the time allowed before it must be replaced. It is displayed in a count–down format. When the remaining lifetime becomes less than the percentage specified in parameter No. 8911 of the corresponding service life, the remaining lifetime display turns red. Count–down continues even after the service life has expired.

To specify the remaining lifetime, key in the corresponding data and press the [INPUT] soft key (or INPUT key).

Pressing the [+INPUT] soft key causes newly entered data to be added to the current remaining lifetime.

The valid data range for this item is: 0 to the corresponding service life

Pressing the [CLEAR] soft key, then the [EXEC] soft key causes the remaining lifetime to be set with the same value as the service life.

NOTE
1 If an attempt is made to enter data for an item number for which no item name or service life has been registered, the warning message “EDIT REJECTED” appears.
2 If an attempt is made to enter data that does not fall in the valid data range, the warning message “DATA IS OUT OF RANGE” appears.
3 If an attempt is made to enter data that would set the remaining lifetime below 0, it will be clamped to 0.
4 If the [TYPE] soft key is pressed, the warning message “EDIT REJECTED” appears.

(3) Count type
The way counting is carried out can be specified here.
Pressing the [TYPE] causes the following count types to be displayed as soft keys. Select the desired one and press the [EXEC] soft key to set it up.

Software	Meaning	Display
[EFFECT]	No counting takes place (the counter is halted).	– – –
[ALL]	Counting continues unconditionally (nonstop counting).	ALL
[LIV]	Counting continues as long as the power is on.	LIV
[RUN]	Counting continues throughout run time.	RUN
[CUT]	Counting continues throughout cutting time.	CUT

NOTE
1 If an attempt is made to enter data for an item number for which no item name or service life has been registered, the warning message “EDIT REJECTED” appears.
2 Neither the [INPUT] nor [+INPUT] soft key does not function.
3 Nonstop counting will have an error of 24 hours in a leap year.

Inputting and outputting the registered data

The registered data related to the service life of consumables can be output to an external unit, using the [PUNCH] soft key. Similarly, data can be input from an external unit for registration, using the [READ] soft key. These input/output operations can be performed on the status, setting, and menu (machine consumables only) screens.

```
EDIT *** ***** *** ****    19:27:05
[      ][  READ  ][ PUNCH ][      ][      ]

[      ][        ][       ][  CAN ][ EXEC ]
```

- **Data output**

In EDIT mode, pressing the [PUNCH] soft key causes registered data to be output in the following formats.

○ Status and setting screens

Format

```
G10 L60 P01 Aa Rr [n] Qq ;
G10 L60 P02 Aa Rr [n] Qq ;
G10 L60 P03 Aa Rr [n] Qq ;
        :
```

○ Menu screen (machine consumables only)

Format

```
G10 L61 P01 [n] ;
G10 L61 P02 [n] ;
G10 L61 P03 [n] ;
        :
```

a : Service life

r : Remaining lifetime

n : Item name
[Alphanumeric character]

q : Count type
0 = No counting.
1 = Nonstop counting.
2 = Counting continues as long as the power is on.
3 = Counting continues throughout run time.
4 = Counting continues throughout cutting time.

- **Data input**

In EDIT mode, pressing the [READ] soft key causes input data to be registered according to a specified format (G10). Registration is possible even if the format (G10) is already in program memory as long as the programmable data input option is available.

NOTE
Registration may not be performed correctly unless the input format (G10) matches the output format.

Parameter

8911	Ratio to the service life of each item on the periodic maintenance screen

[Data type] Byte

[Unit of data] 1%

[Valid data range] 0 to 100

On the periodic maintenance screen, the remaining lifetime display turns red for warning purposes, if the remaining lifetime becomes below a specified percentage of the corresponding service life.

12.2 EDIT

12.2.1 Part Program Storage Length

General

One of the following part program length can be selected.

FS16*i*/160 *i*(m)	—		40	80	160	320	640	1280	2560	5120
FS18*i*/180*i* (m)		20	40	80	160	320	640	1280	—	—
FS21*i*/210*i* (m)	10	20	40	80	160	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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.3.1	Displaying Memory Used and a List of Programs
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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.
63/125/200/400/1000

Alarm and message

Number	Message	Description
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option), or 1000(option). Delete unnecessary programs and execute program registeration again.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.3.1	Displaying Memory Used and a List of Programs
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.3.1	Displaying Memory Used and a List of Programs
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.3.1	Displaying Memory Used and a List of Programs

12.2.3 Memory Protection Key

General

A key called the data protection key is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously.

Signal

Memory protection signal KEY1 to KEY4 <G046#3 to #6>

[Classification] Input signal

[Function] Enables the changing of the memory contents from the MDI panel. Four signals are provided. The operations that can be performed on the contents of memory by each signal vary depending on the setting of bit 7 (KEY) of parameter No. 3290.

When KEY = 0

- KEY1: Enables the input of tool compensation values and the workpiece zero point offset values, and workpiece coordinate systems shift amount.
- KEY2: Enables the input of setting data and macro variables, and tool life management data.
- KEY3: Enables program loading and editing.
- KEY4: Enables PMC data (counter data tables)

When KEY = 1

- KEY1: Enables program loading and editing, as well as the input of PMC parameters.
- KEY2 to KEY4: Not used

[Operation] When a signal is set to 0, the associated operations are disabled.
When a signal is set to 1, the associated operations are enabled.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G046		KEY4	KEY3	KEY2	KEY1			

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3290	KEY							

KEY For memory protection keys:

0 : The KEY1, KEY2, KEY3, and KEY4 signals are used.
1 : Only the KEY1 signal is used.

NOTE
The functions of the signals depend on whether KEY = 0 or KEY = 1.

Alarm and message

Warning message	Contents
WRITE PROTECT	Key input is invalid because of memory protect signal.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11	SETTING AND DISPLAYING DATA
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11	SETTING AND DISPLAYING DATA
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11	SETTING AND DISPLAYING DATA
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11	SETTING AND DISPLAYING DATA

12.2.4 Password Function

General

The password function locks NE9 (bit 4 of parameter No. 3202), used to protect program Nos. 9000 to 9999, by using the PASSWD (No. 3210) and KEYWD (No. 3211) parameters. When NE9 is locked, NE9 cannot be set to 0. Therefore, the protection for programs numbered 9000 to 9999 cannot be released unless the correct keyword is entered.

NE9 is locked when different values are set in the PASSWD and KEYWD parameters. The values set in the two parameters are not displayed. NE9 is unlocked when the value preset in the PASSWD parameter is set in the KEYWD parameter. When 0 is indicated for the PASSWD parameter, a value has not yet been set for PASSWD.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				

[Data type] Bit

NE9 Editing of subprograms with program numbers 9000 to 9999

0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)
(2) Program output (Even when outputting all programs is specified, programs with program numbers 9000 to 9999 are not output.)
(3) Program number search
(4) Program editing after registration
(5) Program registration
(6) Program collation
(7) Displaying programs

3210	Password(PASSWD)

[Data type] Two–word

Set a secret number to this parameter. Its value is not displayed.

CAUTION
This parameter show 0, when no value is set to this parameter. Once a key is lock, parameter NE9 cannot become 0 and PASSWD cannot be changed unless you perform an unlock operation or perform the memory all clear operation. When an attempt is made to modify the password by MDI input operation in this state, the warning message “WRITE PROTECTED” is displayed to indicate that the password cannot be modified. When an attempt is made to modify the password with G10 (programmable parameter input), P/S alarm No. 231 is issued.

3211	Keyword(KEYWD)

[Data type] Two–word

When the value set as the password (set in parameter No. 3210) is set to this parameter, the locked state is released and the user can now modify the password and the value set in bit 4 (NE9) of parameter No. 3202 becomes 0.

NOTE
The value set in this parameter is not displayed. When the power is turned off, this parameter is set to 0.

Alarm and message

Number	Message	Description
231	FORMAT ERROR IN G10 L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1) Address N or R was not entered. 2) A number not specified for a parameter was entered. 3) The axis number was too large. 4) An axis number was not specified in the axis–type parameter. 5) An axis number was specified in the parameter which is not an axis type. 6) An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.9.9	PASSWORD FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.9.9	PASSWORD FUNCTION
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.9.9	PASSWORD FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.9.9	PASSWORD FUNCTION

12.2.5 Background Editing

General

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory.
During background editing, all programs cannot be deleted at once.

Alarm and message

Number	Message	Description
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. Use background editing correctly.
239	BP/S alarm	Background editing was performed while the external punch was being executed in external I/O device control.
240	BP/S alarm	Background editing was done while in MDI operation.

NOTE
Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.9.8	BACKGROUND EDITING
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.9.8	BACKGROUND EDITING
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.9.8	BACKGROUND EDITING
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.9.8	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
7100							THD	

[Data type] Bit type

THD Manual handle feed in TEACH IN JOG mode

0 : Valid
1 : Invalid

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.10.3	CREATING PROGRAMS IN TEACH IN MODE
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.10.4	CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.10.4	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. Pitch error compensation data
5. Custom macro common variables.

Parameter

This CNC has three 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 CHANNEL.
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 (parameter No. 0020)
↓

0020	I/O CHANNEL

Specify a channel for an input/output device.
I/ O CHANNEL
=0 : Channel1
=1 : Channel1
=2 : Channel2
=3 : Channel3

I/O CHANNEL	Parameter	Description
I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
	0102	Number specified for the input/output device
	0103	Baud rate
I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
	0112	Number specified for the input/output device
	0113	Baud rate
I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
	0122	Number specified for the input/output device
	0123	Baud rate
I/O CHANNEL=3 (channel 3)	0131	Stop bit and other data
	0132	Number specified for the input/output device
	0133	Baud rate
	0134	Selection of protocol
	0135	Selection of RS–422 or RS–232–C, and other data

	#7	#6	#5	#4	#3	#2	#1	#0
0000							ISO	TVC

Setting entry is acceptable.

[Data type] Bit

TVC TV check
0 : Not performed
1 : Performed

ISO Code used for data output
0 : EIA code
1 : ISO code

0020	I/O CHANNEL: Selection of an input/output device

Setting entry is acceptable.

[Data type] Byte

[Valid data range] 0 to 3

0 : Select the device of channel 1 (I/O device connected to JD36A of motherboard)

1 : Select the device of channel 1 (I/O device connected to JD36A of motherboard)

2 : Select the device of channel 2 (I/O device connected to JD36B of motherboard)

3 : Select the device of channel 3 (I/O device connected to Serial communication board)

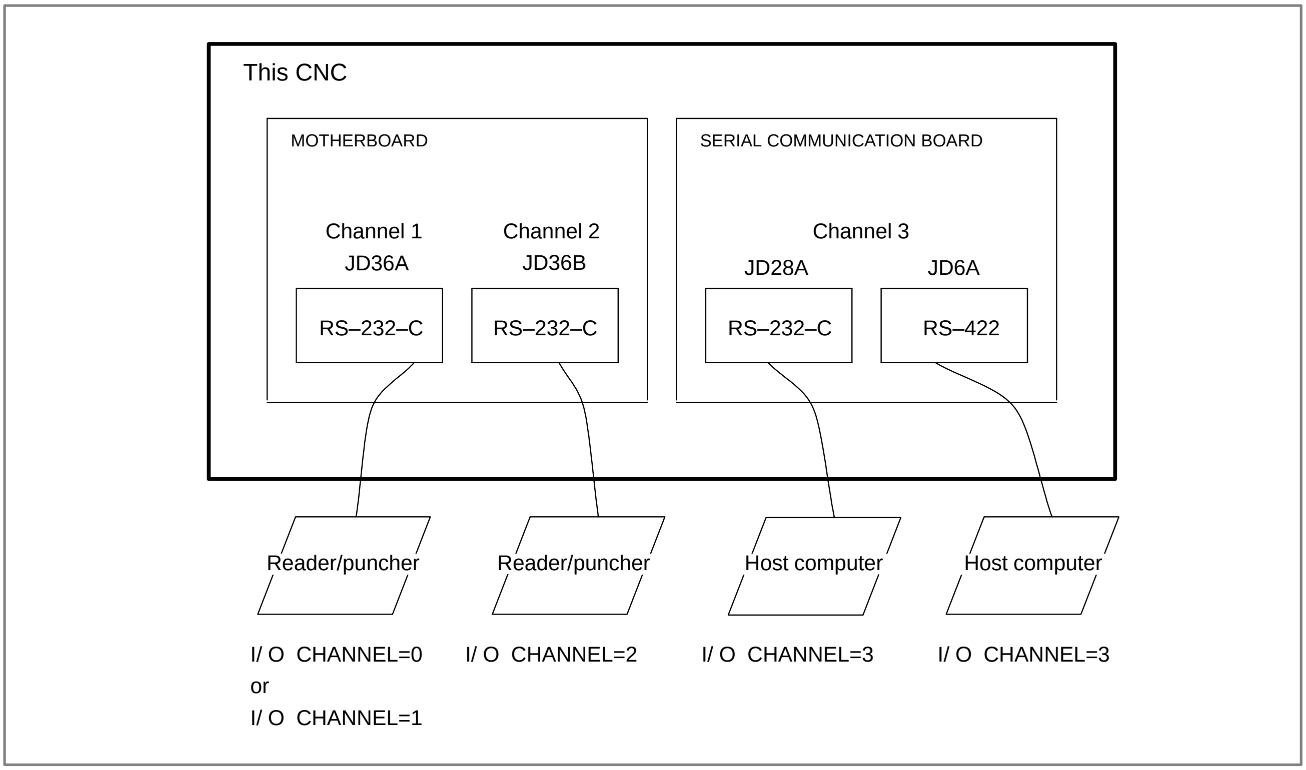
This CNC has three channels for interfacing I/O devices. Set the channels for interfacing I/O devices. Set the channel to which an I/O device to be used is connected.

NOTE

1 The input/output device used can be selected also on the setting screen. Using the setting screen is a more common method for selecting the device.

2 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.
I/O CHANNEL=0 and I/O CHANNEL=1 both refer to channel 1. For each, parameters to set the baud rate, the number of stop bits, and other data are provided separately.

3 JD36A, JD36B, JD28A and JD6A are the connectors on the PCB.



(1) Parameters common to all channels

	#7	#6	#5	#4	#3	#2	#1	#0
0100	ENS				NCR		CTV	

Setting entry is acceptable.

[Data type] Bit type

CTV Character counting for TV check in the comment section of a program.

0 : Not performed
1 : Performed

NCR Output of the end of block (EOB) in ISO code

0 : LF, CR, CR are output.
1 : Only LF is output.

ENS Action taken when a NULL code is found during read of EIA code

0 : An alarm is generated.
1: The NULL code is ignored.

(2) Parameters for channel 1 (I/O CHANNEL=0)

	#7	#6	#5	#4	#3	#2	#1	#0
0101	NFD				ASI			SB2

[Data type] Bit type

SB2 The number of stop bits

0 : 1 bit
1 : 2 bits

ASI Code used at data input

0 : EIA or ISO code (automatically distinguished)
1 : ASCII code

NFD Feed before and after the data at data output

0 : Output
1 : Not output

NOTE
When input/output devices other than the FANUC PPR are used, set NFD to 1.

0102	Number specified for the input/output device (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 0, with one of the set values listed in Table 13.1 (a).

Table 13.1 (a)

Set value	Input/output device
0	RS–232–C (Used control codes DC1 to DC4)
1	FANUC CASSETTE ADAPTOR 1 (FANUC CASSETTE B1/ B2)
2	FANUC CASSETTE ADAPTOR 3 (FANUC CASSETTE F1)
3	FANUC PROGRAM FILE Mate, FANUC FA Card Adaptor FANUC FLOPPY CASSETTE ADAPTOR, FANUC Handy File FANUC SYSTEM P-MODEL H
4	RS–232–C (Not used control codes DC1 to DC4)
5	Portable tape reader
6	FANUC PPR FANUC SYSTEM P-MODEL G, FANUC SYSTEM P-MODEL H

0103	Baud rate (when the I/O CHANNEL is set to 0)

[Data type] Byte

Set baud rate of the input/output device used when the I/O CHANNEL is set to 0, with a set value in Table 13.1 (b).

Table 13.1 (b)

Set value	Baud rate (bps)
1	50
2	100
3	110
4	150
5	200
6	300

Set value	Baud rate (bps)
7	600
8	1200
9	2400
10	4800
11	9600
12	19200

(3) Parameters for channel 1 (I/O CHANNEL=1)

	#7	#6	#5	#4	#3	#2	#1	#0
0111	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 1. The meanings of the bits are the same as for parameter 0101.

0112	Number specified for the input/output device (when I/O CHANNEL is set to 1)

[Data type] Byte

Set the number specified for the input/output device used when the I/O CHANNEL is set to 1, with one of the set values listed in Table 13.1 (a).

0113	Baud rate (when I/O CHNNEL 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 CHANNEL=2)

	#7	#6	#5	#4	#3	#2	#1	#0
0121	NFD				ASI			SB2

[Data type] Bit

These parameters are used when I/O CHANNEL is set to 2. The meanings of the bits are the same as for parameter 0101.

0122	Number specified for the input/output device (when I/O CHANNEL is set to 2)

[Data type] Byte

Set the number specified for the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (a).

0123	Baud rate (when the I/O CHANNEL is set to 2)

[Data type] Byte

Set the baud rate of the input/output device used when I/O CHANNEL is set to 2, with a value in Table 13.1 (b).

Alarm and message

Number	Message	Description
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	III.8	DATA INPUT/OUTPUT
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	III.8	DATA INPUT/OUTPUT
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	III.8	DATA INPUT/OUTPUT
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	III.8	DATA INPUT/OUTPUT

13.2 REMOTE BUFFER

Refer to Remote Buffer DESCRIPTIONS (B–61802E–1) 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

0020	I/O CHANNEL: Selection of an input/output device

Setting entry is acceptable.

[Data type] Byte

Set value. :10

0133	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.

0141	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.

0142	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 SCREEN

1. Communication screen

1.1 Operational Proceclure

1) Press function key <SYSTEM>.
2) Press right–end soft key repeatedly until the soft key [C–SERV],[C–OPER] are displayed shown below:

```
  MDI    ***  STOP  ***  ***  ***  12 : 34 : 53

[C–OPER][C–SERV][            ][            ][ ( OPRT  ) ]
```

1.2 Description of screen

1.2.1 Setting screen (one page)

Press soft key [C–OPER] and the following screen is displayed.

```
COMMUNICATION  OPERATION                         O0001   N00000

    DNC   FILE   SELECTION









>
MDI      ***   STOP   ***   ***   ***   12 :  34 : 53

[C–OPER][C–SERV][            ][            ][ (  OPRT   )  ]
```

1.2.2 Service Screen

Press soft key [C–SERV] and the following screen is displayed.
Three pages are available and one of the pages is selected by page key.

```
COMMUNICATION  PARAMETER                              O0001    N00000
 NC    APPLICATION  NAME



 HOST  APPLICATION  NAME



>
  MDI      ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][              ][                ][ (  OPRT  )  ]
```

```
COMMUNICATION  PARAMETER                              O0001    N00000
    CNC STATUS ( UNSOLICITED STATUS )
        RISING      UPPER    word      00000000   11111111
                    LOWER    word      11111111   11111111
        FALLING     UPPER    word      00000000   00000000
                    LOWER    word      01010100   00000010
    INFORMATION REPORT MASK
                    UPPER    word      00000000   00000000
                    LOWER    word      00000000   00000000
    CNC ALARM ( INFORMATION REPORT )
                    UPPER    word      11110001   00000000
                    LOWER    word      01000001   10000111

>
  MDI      ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][              ][                ][ (  OPRT  )  ]
```

```
COMMUNICATION  PARAMETER                              O0001    N00000
    PASCAL STACK ADDRESS
                  UPPER LIMIT                          01ABC000
                  LOWER LIMIT                          01ABC0FF
    SERVICE MODE      1
                                             01010100   00000010
    SERVICE MODE      2
                                             01000000   00000001
    FILE REQUEST TIME OUT
                                                        12345678
    REMOTE REQUEST TIME OUT
                                                        12345678

>
  MDI     ***   STOP   ***   ***   ***   12 : 34 : 53

[C–OPER][C–SERV][              ][              ][ (  OPRT  )  ]
```

1.3 Entry Method

1) Setting screen
   DNC file selection   String input
2) Service screen page 1
   NC application   String input
   Host application   String input
3) Service screen page 2
   Status post enable   Bit input
   Status post mask   Bit input
   Alarm post   Bit input
4) Service screen page 3
   Pascal stack address   Hexadecimal input
   Service mode   Bit input
   File request time-out   Integer input
   Remote request time-out   Integer input

   String input: Details are described in Section 1.4, ”String Input Mode.”

   Integer input: Entered as positive integers from 0 to 99999999

   Bit input: Entered as 0/1; 00000000 to 11111111

   Hexadecimal input: Entered as a hexadecimal number from 00000000 to FFFFFFFF

[Setting procedure]

1 Put the system in the MDI mode.
2 Cause the setting screen or service screen to appear, and press the [(OPRT)] soft key.
3 Move the cursor to the item you want to specify, using the page and cursor keys.
4 Enter the setting value from the keypad, and press the [INPUT] soft key.

1.4 String Entry Mode

1) On the string input screen, the cursor is placed on top.

```
COMMUNICATION  PARAMETER                              O0001   N00000
■  DNC   FILE  SELECTION
└ The cursor is positioned here at first.
```

2) Press the [(OPRT)] key, and the following soft keys will appear on the screen.

```
[  STRING  ]  [  INPUT  ]  [  CLEAR  ]  [  INS. CH  ]  [  DEL. CH  ]
```

3) Press the [STRING] key, then it enters the string input mode.

```
      DNC   FILE  SELECTION
          ■
         ↑
         └   When the cursor is here, it is the string input mode.
>
MDI    ***   STOP   ***   ***   ***   12 : 34 : 53
[  STRING  ]  [  INPUT  ]  [  CLEAR  ]  [  INS. CH  ]  [  DEL. CH  ]
```

4) Specify the DNC file name.
(Example) O1000. PRG

5) Press the [INPUT] soft key to input the values.

```
   DNC   FILE  SELECTION
          O1000.PRG■


>
  MDI      ***   STOP   ***   ***   ***   12 : 34 : 53
[  STRING  ]  [  INPUT  ]  [  CLEAR  ]  [  INS. CH  ]  [  DEL. CH  ]
```

6) Deleting the DNC file name

- If you want to delete the entire name, press the [CLEAR] soft key.
- If you want to delete a letter at the cursor, press the [DEL.CH] soft key.

(Example) ABCDEFG■

Cursor

To delete letter E, move the cursor to that letter:

ABCDEFG

Press [DEL.CH], and the result will be:

ABCDFG

7) Overwriting

(Example) When you want to overwrite letters starting at C with letters XYZ, move the cursor to letter C.

ABCDEFG

Enter ”XYZ,” then press the [INPUT] soft key.

ABXYZFG

8) Inserting letters

(Example) To insert string ABC after string ABC, move the cursor to letter D.

ABCDEFG

Enter letters ABC.

ABCDEFG

Press the [INS.CH] soft key.

ABCABC EFG

9) Canceling the string input mode

If you press the [STRING] soft key, the cursor goes back to the position shown below, and the string input mode is canceled.

■ DNC file selection

O1000. PRG

NOTE

1 The page keys do not work during the string input mode.

2 Switching the CNC mode during a string input mode cancels the string input mode.

2. PARAMETER DESCRIPTION

(1) Setting screen

- DNC file selection
  To start DNC operation, specify a file name in the host computer.
  Format: Oxxxx. PRG (where xxxx is a four–digit decimal number.)

(2) Service screen

- CNC and host application names
  Specify these parameters with CNC and host application names in the string input mode.
- Status post enable

Rising change	Upper word:	11111111	11111111
	Lower word:	01110000	10111110
Falling change	Upper word:	00000000	00000000
	Lower word:	11111111	11111111

Each bit in this parameter specifies whether to send the CNC status information to the DNC1 board according to a local request.
The CNC status information consists of a bit pattern listed in Table A. Each bit in the bit pattern corresponds to the bits in the rising and falling change words.
Setting value 0: Mask
1: Post
The rising change word specifies that a status information bit change from 0 to 1 be posted, and the falling change word, 1 to 0.

> **NOTE**
> Use the lower word only.

○ CNC status information bit pattern (Table A)
Lower word

15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00

Bit position: Signal name	— CNC status information —
00: RWD	Rewind signal
01: AL	Alarm output signal
02: RST	Resetting signal
03: SPL	Feed hold lamp signal
04: STL	Cycle start lamp signal
05: OP	Cycle operating 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	M03 decode output signal (*)

* For the T series (two–path control), bits 08 to 11 correspond to M00 to M30 at HEAD2 respectively, and bits 12 to 15, at HEAD1.

- Status post mask
  Not used
- Alarm post
  This parameter specifies whether the bit position of a CNC alarm is posted to the host when a status change occurs according to a local request.

  Setting value
  0: Not to post
  1: To post

  The relationships between the parameter bits and alarms are as follows:

Upper word bit parameter

31	30	29	28	27	26	25	24	23	22	21	20	19	18	17	16

Lower word bit parameter

15	14	13	12	11	10	09	08	07	06	05	04	03	02	01	00

Bit number: 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
16:	Battery alarm

- Service mode 2
  The DNC1 all–file directory information read function works as follows:
  00000000 00000001: Only file numbers are read.
  00000000 00000000: File numbers and sizes are read.
- File request time-out
- Remote request time-out
  This parameter specifies when a time-out is to occur for a request from the host.

  Parameter setting: 0 to 99999999
  Unit of time: ms

NOTE
1 If value 0 is specified in the parameter, a conventional fixed time of about 25.6 seconds is specified as a time-out value.
2 The time for the time-out is counted in 32 ms units internally. The actual time for a time-out to occur is calculated as:

Actual time-out time = (parameter setting ÷ 32 + 1)* x 32 [ms]*

The term enclosed in parentheses is rounded down at a decimal point.

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 <G058#1>

[Classification] Input signal

[Function] Programs are registered through the reader/puncher 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 KEY3 <G046#5> determines whether to register or collate.)
- Bit 1 (RAL) of parameter No. 3201 selects whether to register all programs in a file or one program at a time. Bit 0 (RDL) of parameter No. 3201 can be used to delete all programs previously stored in the part program memory. However, it is impossible to delete programs protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.
- When programs are being registed or collated, the read/punch busy signal (RPBSY) is kept to be logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external read start signal EXRD is ignored.
- When programs are being registered or collated, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the registeration or collation is discontinued.
- If the foreground processing is already using the reader/puncher interface (for example, during DNC operation or program reading in the edit mode), the external read start signal EXRD is ignored.

- There are some other conditions to determine whether a program can be registered or collated. For example, a program cannot be registered or collated, if a program with the same program number is being executed in the foreground processing.

External Punch Start Signal EXWT <G058#3>

[Classification] Input signal

[Function] Programs stored in the part program memory are output to an external unit via the reader/puncher interface.

[Operation] When this signal becomes logical 1, the CNC operates as follows:

- In all modes other than the MDI mode, the background edit function outputs all programs stored in the part program memory to an external output device.
- When programs are being output, the read/punch busy signal RPBSY becomes logical 1.
- When the background processing–activated signal BGEACT is logical 1 (for example, during background editing or MDI mode), the external punch start signal EXWT is ignored.
- When programs are being output, if the system is reset or the external read/punch stop signal EXSTP becomes logical 1, the output is discontinued.
- If the foreground processing is already using the reader/puncher interface (for example, during DNC operation or program reading in the edit mode), the external punch start signal EXWT is ignored.
- There are some other conditions to determine whether all programs can be output. For example, a program cannot be output, if it is running or protected by bits 0 (NE8) and 4 (NE9) of parameter No. 3202.

External Read/Punch Stop Signal EXSTP <G058#2>

[Classification] Input signal

[Function] When the external read/punch stop signal becomes logical 1, it stops program registeration, collation, or output via the reader/puncher interface 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 <F053#4>

[Classification] Output signal

[Function] This signal indicates that the background edit function is operating.

[Output condition] This signal becomes logical 1 when:

- The [BG EDIT] soft key is pressed to put the CNC in the background edit mode.
- The MDI mode is selected.
- The external read or punch start signal starts program registeration, collation, or output.
- Program upload or download is started by the DNC1, DNC2, or MMC.

This signal becomes logical 0 when:

- The [BG END] soft key is pressed to terminate the background edit mode.
- The CNC shifts from the MDI mode to another mode.
- Program registeration or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).
- Program upload or download started by the DNC1, DNC2, or MMC is ended.

Read/punch busy signal RPBSY <F053#2>

[Classification] Output signal

[Function] This signal indicates that program registeration, collation, or output triggered by the external read or punch start signal is under way.

[Output condition] This signal becomes logical 1, when:

- The external read or punch start signal triggers program registeration, collation, or output.

This signal becomes logical 0, when:

- Program registeration collation or output triggered by the external read or punch start signal ends either normally or abnormally (reset or requested by the EXSTP signal).

Read/punch alarm signal RPALM <F053#3>

[Classification] Output signal

[Function] This signal indicates that an alarm condition has occurred during program registeration, collation, or output triggered by the external read or punch start signal.

[Output condition] This signal becomes logical 1, when:

- An alarm condition occurs during program registeration, collation, or output triggered by the external read or punch start signal.

This signal becomes logical 0, when:

- The system is reset, or the external read/punch stop signal EXSTP is input.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058					EXWT	EXSTP	EXRD	

	#7	#6	#5	#4	#3	#2	#1	#0
F053				BGEACT	RPALM	RPBSY		

Parameter

Input/output channel number (parameter No. 0020)
↓

0020 I/O CHANNEL	I/O CHANNEL=0 (channel 1)	0101	Stop bit and other data
		0102	Number specified for the input/output device
		0103	Baud rate
	I/O CHANNEL=1 (channel 1)	0111	Stop bit and other data
		0112	Number specified for the input/output device
		0113	Baud rate
	I/O CHANNEL=2 (channel 2)	0121	Stop bit and other data
		0122	Number specified for the input/output device
		0123	Baud rate
	I/O CHANNEL=3 (channel 3)	0131	Stop bit and other data
		0132	Number specified for the input/output device
		0133	Baud rate
		0134	Selection of protocol
		0135	Selection of RS–422 or RS–232–C, and other data

Specify a channel for an input/output device.
I/ O CHANNEL
=0 : Channel1
=1 : Channel1
=2 : Channel2
=3 : Channel3

	#7	#6	#5	#4	#3	#2	#1	#0
3201		NPE	N99			REP	RAL	RDL

[Data type] Bit

RDL When a program is registered by input/output device external control
0 : The new program is registered following the programs already registered.
1 : All registered programs are deleted, then the new program is registered.
Note that programs which are protected from being edited are not deleted.

RAL When programs are registered through the reader/puncher interface
0 : All programs are registered.
1 : Only one program is registered.

REP Action in response to an attempt to register a program whose number is the same as that of an existing program
0 : An alarm is generated.
1 : The existing program is deleted, then the new program is registered.
Note that if the existing program is protected from being edited, it is not deleted, and an alarm is generated.

N99 With an M99 block, when bit 6 (NPE) of parameter No. 3201 = 0, program registration is assumed to be:

0 : Completed
1 : Not completed

NPE With an M02, M30, or M99 block, program registration is assumed to be:

0 : Completed
1 : Not completed

	#7	#6	#5	#4	#3	#2	#1	#0
3202				NE9				NE8

[Data type] Bit type

NE8 Editing of subprograms with program numbers 8000 to 8999

0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 8000 to 8999 are not deleted.)

(2) Program output (Even when outputting all programs is specified, programs with program numbers 8000 to 8999 are not output.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

NE9 Editing of subprograms with program numbers 9000 to 9999

0 : Not inhibited
1 : Inhibited

The following edit operations are disabled:

(1) Program deletion (Even when deletion of all programs is specified, programs with program numbers 9000 to 9999 are not deleted.)

(2) Program punching (Even when punching of all programs is specified, programs with program numbers 9000 to 9999 are not punched.)

(3) Program number search

(4) Program editing after registration

(5) Program registration

(6) Program collation

(7) Displaying programs

Alarm and message

Number	Message	Description
079	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.
233	BP/S ALARM	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.8.4	PROGRAM INPUT/OUTPUT
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.8.4	PROGRAM INPUT/OUTPUT
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.8.4	PROGRAM INPUT/OUTPUT
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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) Select the DNC operation mode.
(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) Select the DNC operation mode.
(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 <G058#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 <G058#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
G058		STWD	STRD					

Parameter

3218	Program number registered in synchronous input/output operation (for the four–digit program number function)

[Data type] Word

[Valid data range] 1 to 9999

This parameter sets a program number for such a program that is input by an input/output unit in the synchronous input/output operation mode and is executed while being stored in memory.

NOTE

1 If a value beyond the valid data range is specified, the program number of the input program is assumed.

2 When the eight–digit program number function is provided, the program number should not be set in this parameter but in parameter 3219.

3219	Program number registered in synchronous input/output operation (for the eight–digit program number function)

[Data type] 2–word

[Valid data range] 1 to 99999999

This parameter sets a program number for such a program that is input by an input/output unit in the synchronous input/output operation mode and is executed while being stored in memory.

NOTE

1 If a value beyond the valid data range is specified, the number of the input program is registered.

2 When the eight–digit program number function is not provided, the program number should not be set in this parameter but in parameter 3218.

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/M99	M198 and M99 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.
222	DNC OP. NOT ALLOWED IN BG–EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.

Note

NOTE

1 M198 (file access) cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in P/S 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 P/S 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.

To use external program input start signal MINP <G058#0> to start data input, the following conditions must be satisfied:

- The reader/punch interface function is enabled.
- Bit 7 (MIP) of parameter No. 3201 is set to 1.
- Programs are not protected using the memory protection signal.
- Automatic operation mode is set.
- The current state is other than the start of automatic operation, that is, cycle start lamp signal STL is set to 0. (Automatic operation signal OP may be set to 1.)

Signal

External program input start signal MINP <G058#0>

[Classification] Input signal

[Function] This signal starts loading of a program from an input unit into CNC memory.

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

- When memory operation mode is set, but no automatic operation is being performed and program loading is not inhibited by the setting of the memory protection key, the CNC deletes all currently loaded programs, then loads a program from the external input unit into CNC memory.
- When the FANUC Handy File or FANUC Floppy Cassette is being used as the input unit, a desired file can be searched for using the workpiece number search signals (PN1 to PN16), after which the program can be loaded into CNC memory.
  File numbers are indicated using the workpiece number search signals, as follows:

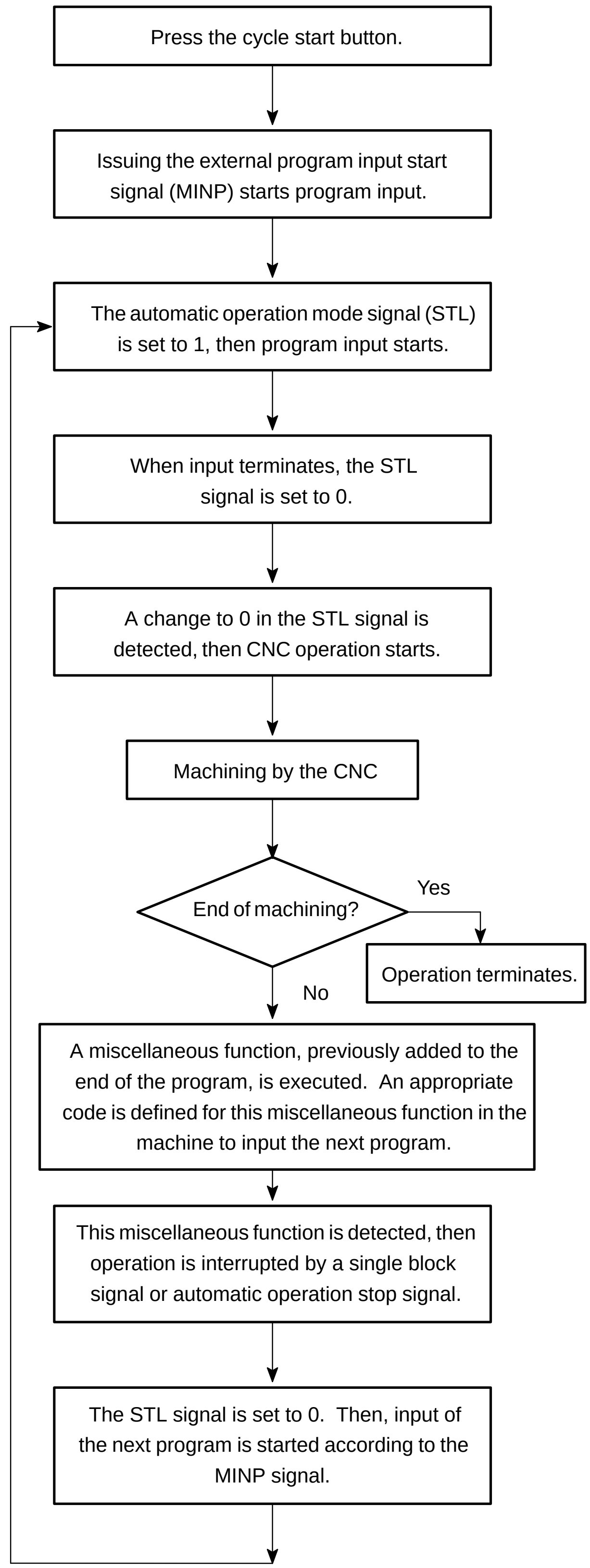
Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02

Workpiece no. search signal					File no.
PN16	PN8	PN4	PN2	PN1	
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

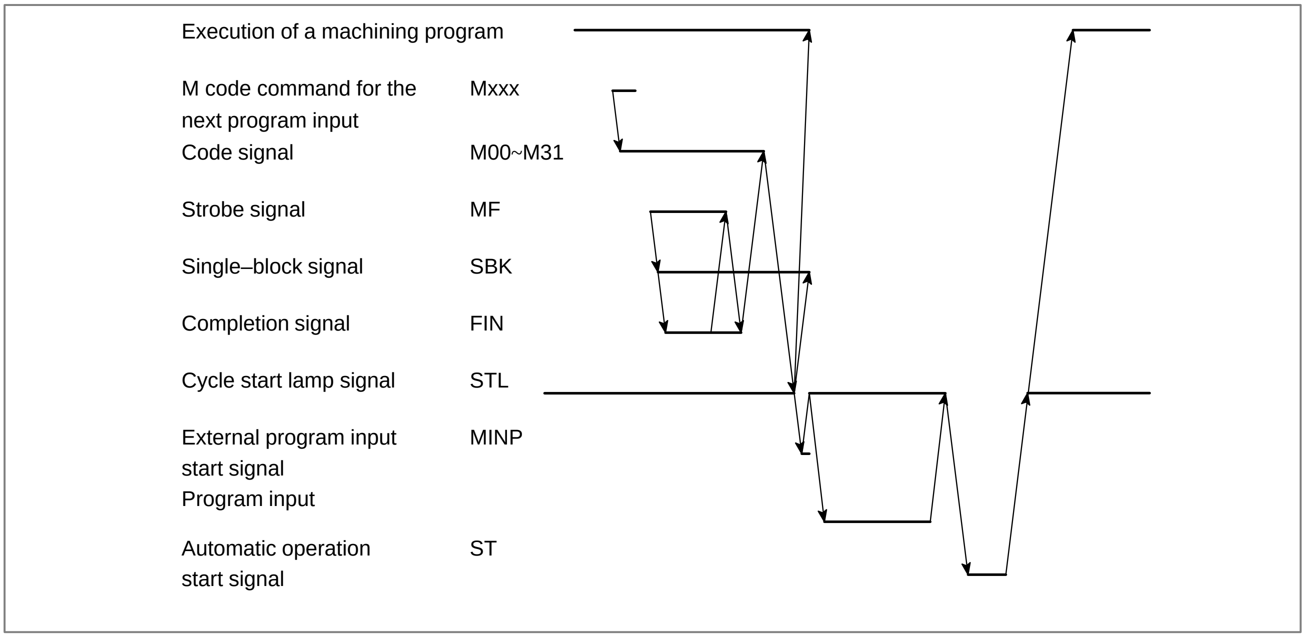
File No. 00 is used for special specification; specifying file No. 00 means that no search operation is to be performed. Therefore, numbers 01 to 31 can be assigned to files.

[Application] This function is applicable to the following case:
When a program to be used for machining is too large to be loaded into CNC memory, the program is divided into several segments. These segments are loaded into memory and executed, one by one.

In this case, the general operation flow is as shown below.



The timing chart for data reading is shown below.



> **CAUTION**
> The M code used for input of the next program must not be buffered.

> **NOTE**
> While a program is being input, the automatic operation mode signal STL is set to 1. Upon termination of program input, STL is set to 0.

Signal Address

	#7	#6	#5	#4	#3	#2	#1	#0
G058								MINP

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3201	MIP							

[Data type] Bit type

MIP Specifies whether to load a program into memory according to the external program input start signal (MINP).

0 : Does not load a program into memory.
1 : Loads a program into memory.

Note

NOTE

1 A program can be input according to the external program input start signal only when the program has only one program number.
To read programs having multiple program numbers, reset the CNC each time the CNC reads one program. After reset, search for a desired program by using the workpiece number search signals, then input the program according to the external program input start signal.

2 When program input is activated by the external program input activation signal, all programs are deleted from the CNC, at the beginning.

3 In background editing, the external program input activation signal is ineffective.

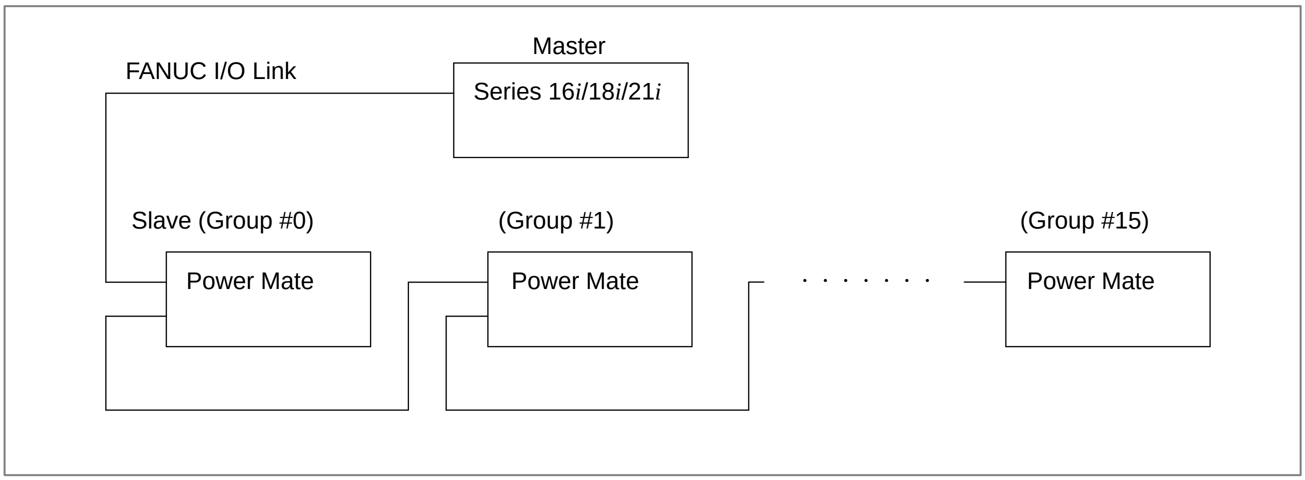
13.8 DATA INPUT/OUTPUT FUNCTIONS BASED ON THE I/O LINK

General

Power Mate programs, parameters, macro variables, and diagnostic (PMC) data are input/output through the FANUC I/O Link.

With the FANUC I/O Link, slaves from group 0 to group 15 can be connected, thus enabling data input/output to and from a maximum of 16 Power Mates.

When data input/output function B based on the I/O Link is used, the external I/O device control function is associated with an I/O Link to enable the specification of an input/output group number and program number from the PMC. The external I/O device control function operates in the background. This means that when no other background operation is being performed, data can be input/output, regardless of the CNC mode and the screen selected.



The programs, parameters, macro variables, and diagnostic (PMC) data of a slave Power Mate are stored in tape format within the part program storage length; these data items are stored as master program data in a master program memory area.

Data input/output can be performed between the master and the slave of a selected group. When the ordinary data input/output function based on the I/O Link is used, a group is selected by means of parameter setting. When data input/output function B based on the I/O Link is used, a group is selected by issuing the DI signal. Data input/output cannot be performed between the master and more than one groups at a time.

For details of the Power Mate signals, refer to the “FANUC Power Mate Connection/Maintenance Manual.”

For details of the FANUC I/O Link, refer to the ”FANUC PMC Programming Manual.”
For details of the external I/O device control function, see Section 13.5.

- **Basic data input/output procedure**

(1) Program input/output

(a) Program input

· When the data input/output function based on the I/O Link is used

1) Specify a number between 20 to 35 as the I/O channel on the setting screen to specify a group number.
2) Specify EDIT mode.
3) Display the program screen with function key PROG .
4) To change the program numbers, enter a new program number as follows:
   (i) Select address O.
   (ii) Key in the new program number.
5) Using soft keys [(OPRT)], continuous–menu key ▷ , [READ], and [EXEC], read the program.

· When data input/output function B based on the I/O Link is used

1) To read programs without changing the program numbers, specify –9999. To read programs by changing the program numbers, specify a desired number with the PMC function instruction (WINDW). A program number thus specified serves as the first program number when multiple programs are received. The second and any additional programs are numbered sequentially starting with the first number.
2) Specify a desired group number with group number specification signals SRLNI0 to SRLNI3.
3) Select an I/O Link by setting I/O Link specification signal IOLS to 1.
4) When external punch start signal EXRD is set to 1, programs are read from the Power Mate.

> **NOTE**
> The user cannot read a program having an arbitrary program number registered in a Power Mate. All programs are always read from a Power Mate.

(b) Program output

· When the data input/output function based on the I/O Link is used

1) Set a number between 20 and 35 as the I/O channel on the setting screen used to specify a group number.
2) Set EDIT mode.
3) Display the program screen by pressing the function key PROG .
4) Select address O.

5) Key in a program number.

6) Using soft keys [(OPRT)], continuous–menu key ▷, [PUNCH], and [EXEC], output the program corresponding to the keyed–in program number.

· When data input/output function B based on the I/O Link is used

1) Using the PMC function instruction (WINDW), set the program number of the program to be output. Specify –9999 to output all programs.

2) Specify a desired group number with group number specification signals SRLNI0 to SRLNI3.

3) Select an I/O Link by setting the I/O Link specification signal IOLS to 1.

4) When external punch start signal EXWT is set to 1, the program is output to the Power Mate.

(2) Parameter input/output
A Power Mate parameter in tape format is stored as a master–side NC program in a program memory area. The following program number is assigned to a Power Mate parameter of group n:
<u>(Value set in parameter No. 8760) + n × 10</u>

(a) Parameter input

· When the data input/output function based on the I/O Link is used
In step 4) of program input (described in (1).(a)), key in a program number for a Power Mate parameter. The other steps are the same as those for program input.

· When data input/output function B based on the I/O Link is used
In step 1) of program input (described in (1).(a)), specify a program number for the Power Mate parameters. The remaining steps are the same as those for program input.

(b) Parameter output

· When the data input/output function based on the I/O Link is used
In step 5) of program output (described in (1).(b)), key in a program number for a Power Mate parameter. The other steps are the same as those for program output.

· When data input/output function B based on the I/O Link is used
In step 1) of program output (described in (1).(b)), specify a program number for a Power Mate parameter. The remaining steps are the same as those for program output.

(3) Macro variable input/output
In the same way as for parameters, a Power Mate macro variable in tape format is stored as a master side NC program in a program memory area. The following program number is assigned to a Power Mate macro variable of group n:
<u>(Value set for parameter No. 8760) + n × 10 + 1</u>

(a) Macro variable input

- When the data input/output function based on the I/O Link is used
  With Power Mate DI signals EDG00 to EDG15, specify a start number for the macro variables to be read. With EDN00 to EDN15, specify the number of macro variables to be read.
  In step 4) of program input (described in (1).(a)), specify a program number for the Power Mate macro variables. The other steps are the same as those for program input.
- When data input/output function B based on the I/O Link is used
  With Power Mate DI signals EDG00 to EDG15, specify a start number for the macro variables to be read. With EDN00 to EDN15, specify the number of macro variables to be read.
  In step 1) of program input (described in (1).(a)), specify a program number for Power Mate macro variables. The remaining steps are the same as those for program input.

(b) Macro variable output

- When the data input/output function based on the I/O Link is used
  In step 5) of program output (described in (1).(b)), specify a program number for a Power Mate macro variable. The other steps are the same as those for program output.
- When data input/output function B based on the I/O Link is used
  In step 1) of program output (described in (1).(b)), specify a program number for a Power Mate macro variable. The remaining steps are the same as those for program output.

NOTE
The setting parameter or bit 1 (ISO) of parameter No. 0000 must be set to 1 to enable the use of the ISO code for macro variable output. When a macro variable is output in EIA code, the Power Mate issues alarm 001, while the Series 16*i*/18*i*/21*i* issues alarm 86.

(4) Diagnostic (PMC) data input/output
In the same way as the parameters and macro variables, a Power Mate diagnostic data item in tape format is stored as a master–side NC program in a program memory area. The following program number is assigned to a Power Mate diagnostic data item of group n:
<u>(Value set in parameter No. 8760) + n × 10 + 2</u>

(a) Diagnostic (PMC) data input

· When the data input/output function based on the I/O Link is used
With Power Mate DI signals EDG00 to EDG15, specify a start number for the diagnostic data items to be read. With EDN00 to EDN15, specify the number of diagnostic data items to be read.
In step 4) of program input (described in (1).(a)), specify a program number for the Power Mate diagnostic data items. The other steps are the same as those for program input.

· When data input/output function B based on the I/O Link is used
With Power Mate DI signals EDG00 to EDG15, specify a start number for the diagnostic data items to be read. With EDN00 to EDN15, set the number of diagnostic data items to be read.
In step 1) of program input (described in (1).(a)), specify a program number for Power Mate diagnostic data items. The remaining steps are the same as those for program input.

(b) Diagnostic (PMC) data output

· When the data input/output function based on the I/O Link is used
In step 5) of program output (described in (1).(b)), specify a program number for a Power Mate diagnostic data item. The other steps are the same as those for program output.

· When data input/output function B based on the I/O Link is used
In step 1) of program output (described in (1).(b)), specify a program number for a Power Mate diagnostic data item. The remaining steps are the same as those for program output.

NOTE
The addresses of Power Mate DI signals EDG00 to EDG15 and EDN00 to EDN15 differ between Model–D and the other models, as indicated in the table below. For macro variable and diagnostic data input, ensure that these signals are processed correctly. If an invalid value or number is set, the Power Mate's external I/O device control function is not started.

Signal name	Power Mate A/B/C/E	Power Mate D, F, H
EDG00 to EDG07 EDG08 to EDG15	G100.0 to G100.7 G101.0 to G101.7	G252.0 to G252.7 G253.0 to G253.7
EDN00 to EDN07 EDN08 to EDN15	G102.0 to G102.7 G103.0 to G103.7	G254.0 to G254.7 G255.0 to G255.7

- **Stopping input/output**

There are two methods of forcibly terminating input/output.

(1) Termination by a reset
Input/output can be terminated by a reset. In this case, however, slave read/write stop signal ESTPIO is not output. Therefore, the operation of the Power Mate is not terminated even if the Power Mate is performing input/output. To terminate Power Mate operation, create a ladder program so that ESTPIO is set to 1 upon the occurrence of a reset.

(2) Termination using external read/punch stop signal EXSTP (applicable only when data input/output function B based on the I/O Link is used)
Input/output can be terminated by setting external read/punch stop signal EXSTP to 1. In this case, slave read/write stop signal ESTPIO is output, and all processing is terminated once the termination of Power Mate input/output has been confirmed. See the timing chart for the case where the Series 16*i*/18*i*/21*i* issues an alarm.

- **Specifying the PMC functions**

The PMC function instruction (WINDW) is used to set the program numbers used with data input/output function B based on the I/O Link.

[Data] This function is a window function for specifying the program numbers used to perform program input/output with the I/O device control function via the I/O Link.

[Input data structure]

Top address +0	+2	+4	+6	+8	+10
Function code	**Completion code**	**Data length**	**Data number**	**Data attribute**	**Data**
194	–	2	0	0	Program number

(‘–’ : Need not be set.)

Specify 0 in the data number and data attribute fields.
Specify 2 in the data length field.
Specify a desired two–byte program number (1 to 9999, –9999) in the data field.

[Output data structure]

Top address +0	+2	+4	+6	+8	+10
Function code	**Completion code**	**Data length**	**Data number**	**Data attribute**	**Data**
194	?	Input data	Input data	Input data	Input data

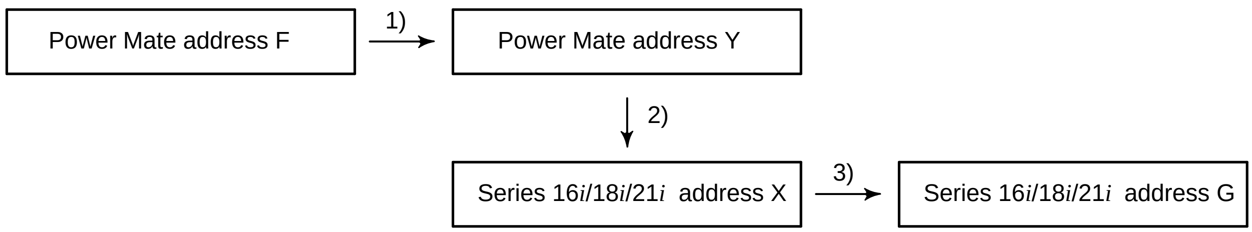
[Completion codes]

0 : Normal termination
5 : Data other than 1 to 9999 or –9999 was specified.

The data length, data number, and data attribute fields are not checked.
For details of the PMC function instructions, refer to the “FANUC PMC Programming Manual.”

- **Power Mate state signals (input)**

When the data input/output function based on the I/O Link is used, the state signals for a specified Power Mate must be reported to the Series 16*i*/18*i*/21*i*. These signals must be posted to the Series 16*i*/18*i*/21*i* via the following path:

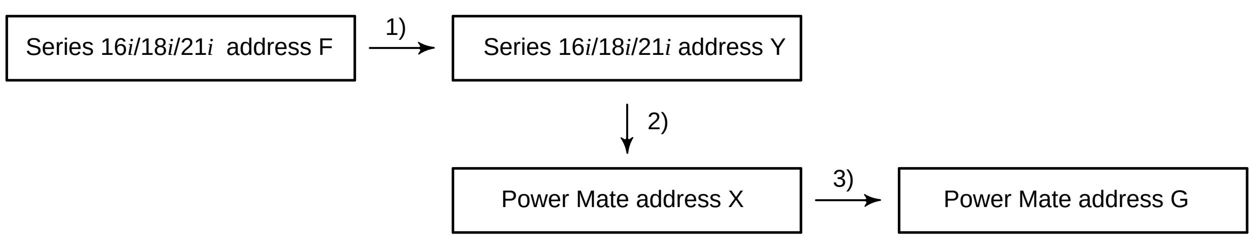


1) Depends on the ladder program of the Power Mate.
2) Depends on the I/O Link.
3) Depends on the ladder program of the Series 16*i*/18*i*/21*i*.

For an explanation of the functions of the Power Mate state signals, see item Signal.

- **Power Mate control signals (output)**

When the data input/output function based on the I/O Link is used, Power Mate control signals must be output from the Series 16*i*/18*i*/21*i* to control the external I/O device control function of a specified Power Mate. These signals must be posted to the Power Mate via the following path:



1) Depends on the ladder program of the Series 16*i*/18*i*/21*i*.
2) Depends on the I/O Link.
3) Depends on the ladder program of the Power Mate.

For an explanation of the functions of the Power Mate control signals, see Section 13.8.(2).

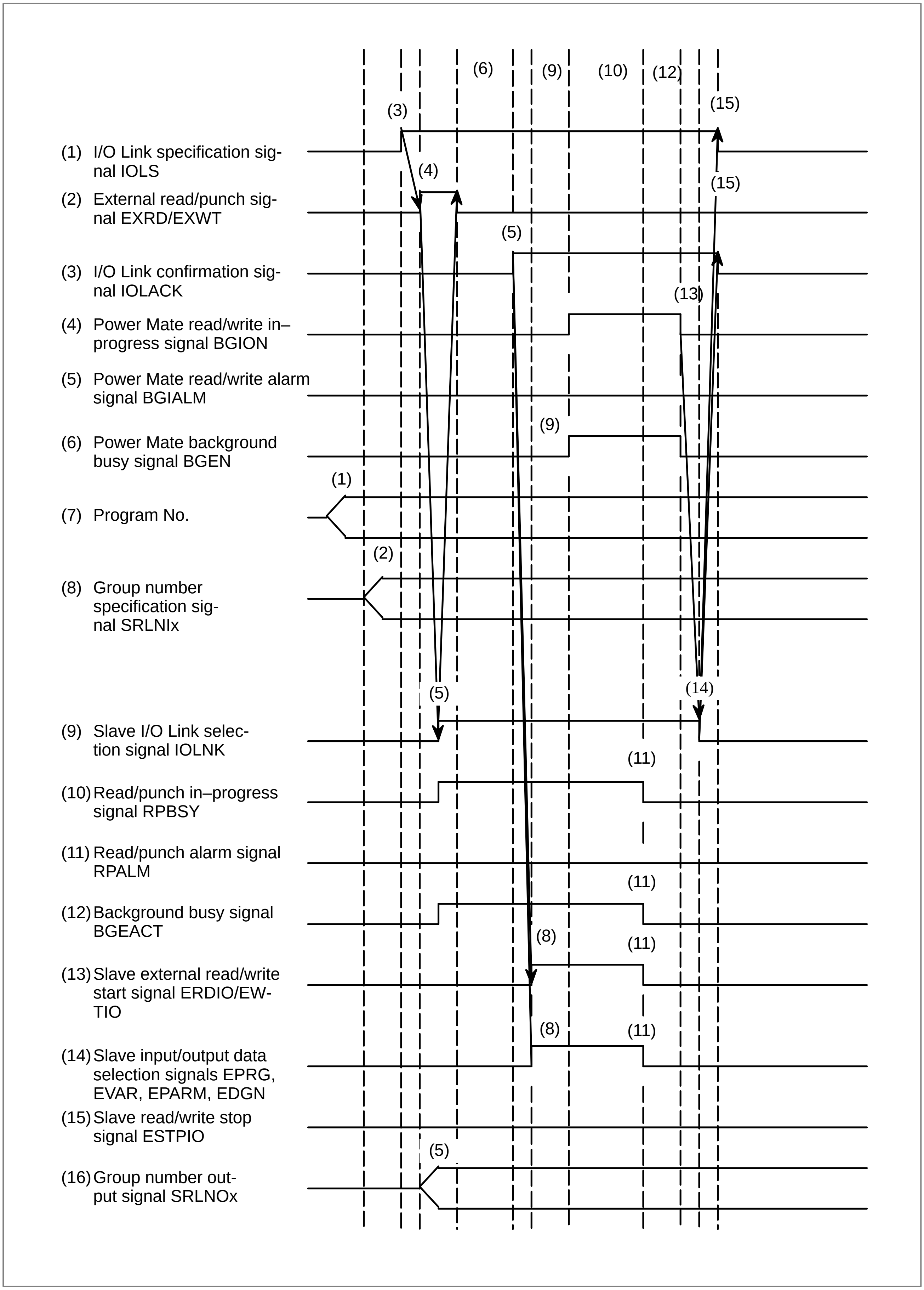
- **DI/DO signal timing charts**

The DI/DO signal timing charts applicable when data input/output function B based on the I/O Link is used are shown below. When the ordinary data input/output function based on the I/O Link is used, 1) through 4) in the figures are subject to MDI–based input/output.

(1) When ordinary input/output is performed

1) Specify a program number with the PMC function instruction (WINDW).

2) Specify the group number of the Power Mate to be used for input/output with group number specification signals SRLNI0 to SRLNI3.
3) Set I/O Link specification signal IOLS to 1.
4) For input, set external read start signal EXRD to 1.
For output, set external punch start signal EXWT to 1.
5) Processing by the Series 16*i*/18*i*/21*i* outputs slave I/O Link selection signal IOLNK and group number output signals SRLNO0 to SRLNO3.
6) Using a ladder program, make the necessary preparations for data input/output using the I/O Link.
7) Upon the completion of step 6), set the I/O Link confirmation signal IOLACK to 1. The Series 16*i*/18*i*/21*i* waits until IOLACK is set to 1.
8) The Series 16*i*/18*i*/21*i* outputs slave external read/write signal ERDIO/EWTIO, and slave input/output data selection signals EPRG, EVAR, EPARM, and EDGN.
9) The Series 16*i*/18*i*/21*i* waits for the external I/O device control function of the Power Mate to start. (The Series 16*i*/18*i*/21*i* waits for the external I/O device control function of the Power Mate to start.)
10) The Series 16*i*/18*i*/21*i* performs data input/output through the I/O Link.
11) Upon the completion of data input/output, the Series 16*i*/18*i*/21*i* sets the slave external read/write start signal and slave input/output data selection signals to 0.
12) The Series 16*i*/18*i*/21*i* waits for the completion of the external I/O device control function of the Power Mate. The Series 16*i*/18*i*/21*i* waits for termination or completion of the external I/O device control function of the Power Mate.
13) Upon the completion of the operation of the external I/O device control function of the Power Mate, the Power Mate read/write in–progress signal BGION is set to 0.
14) The Series 16*i*/18*i*/21*i* sets the slave I/O Link selection signal IOLNK to 0.
15) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(1) I/O Link specification signal IOLS
(2) External read/punch signal EXRD/EXWT
(3) I/O Link confirmation signal IOLACK
(4) Power Mate read/write in–progress signal BGION
(5) Power Mate read/write alarm signal BGIALM
(6) Power Mate background busy signal BGEN
(7) Program No.
(8) Group number specification signal SRLNIx
(9) Slave I/O Link selection signal IOLNK
(10) Read/punch in–progress signal RPBSY
(11) Read/punch alarm signal RPALM
(12) Background busy signal BGEACT
(13) Slave external read/write start signal ERDIO/EWTIO
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(15) Slave read/write stop signal ESTPIO
(16) Group number output signal SRLNOx
(1)
(2)
(3)
(4)
(5)
(6)
(8)
(9)
(10)
(11)
(12)
(13)
(14)
(15)

(2) When an alarm is issued by the Series 16*i*/18*i*/21*i* (including the case where processing is stopped by external read/punch signal EXSTP)

Steps 1) to 10) are the same as those for ordinary input/output.

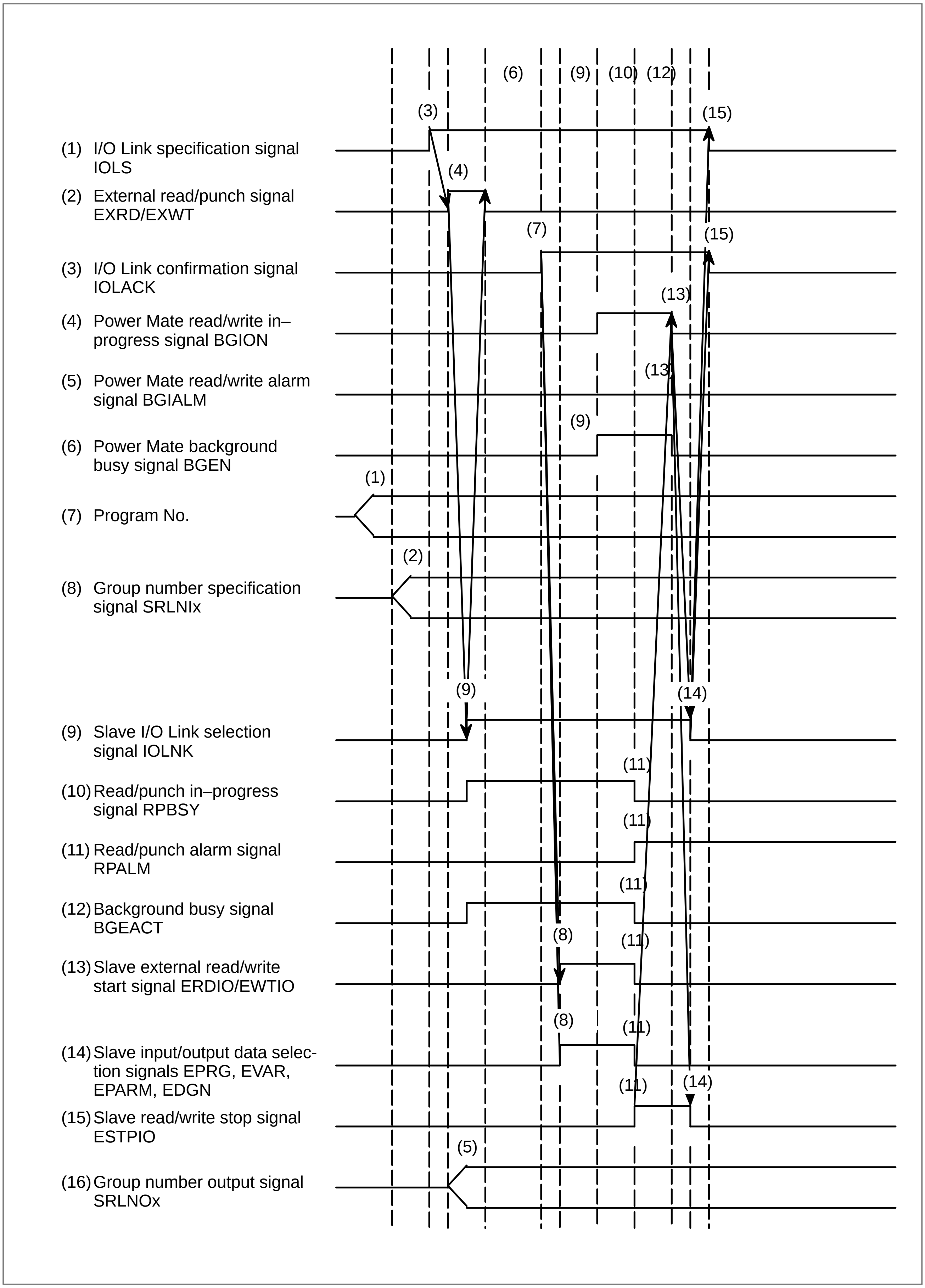
11) When the Series 16*i*/18*i*/21*i* issues an alarm, or when external read/punch stop signal EXSTP is set to 1, slave read/write stop signal ESTPIO is set to 1.

12) The Series 16*i*/18*i*/21*i* waits for the external I/O device control function of the Power Mate to terminate. The Series 16*i*/18*i*/21*i* merely waits until the external I/O device control function of the Power Mate terminates.

13) When the external I/O device control function of the Power Mate terminates, Power Mate read/write in–progress signal BGION is set to 0.

14) The Series 16*i*/18*i*/21*i* sets slave I/O Link selection signal IOLNK and slave read/write stop signal ESTPIO to 0.

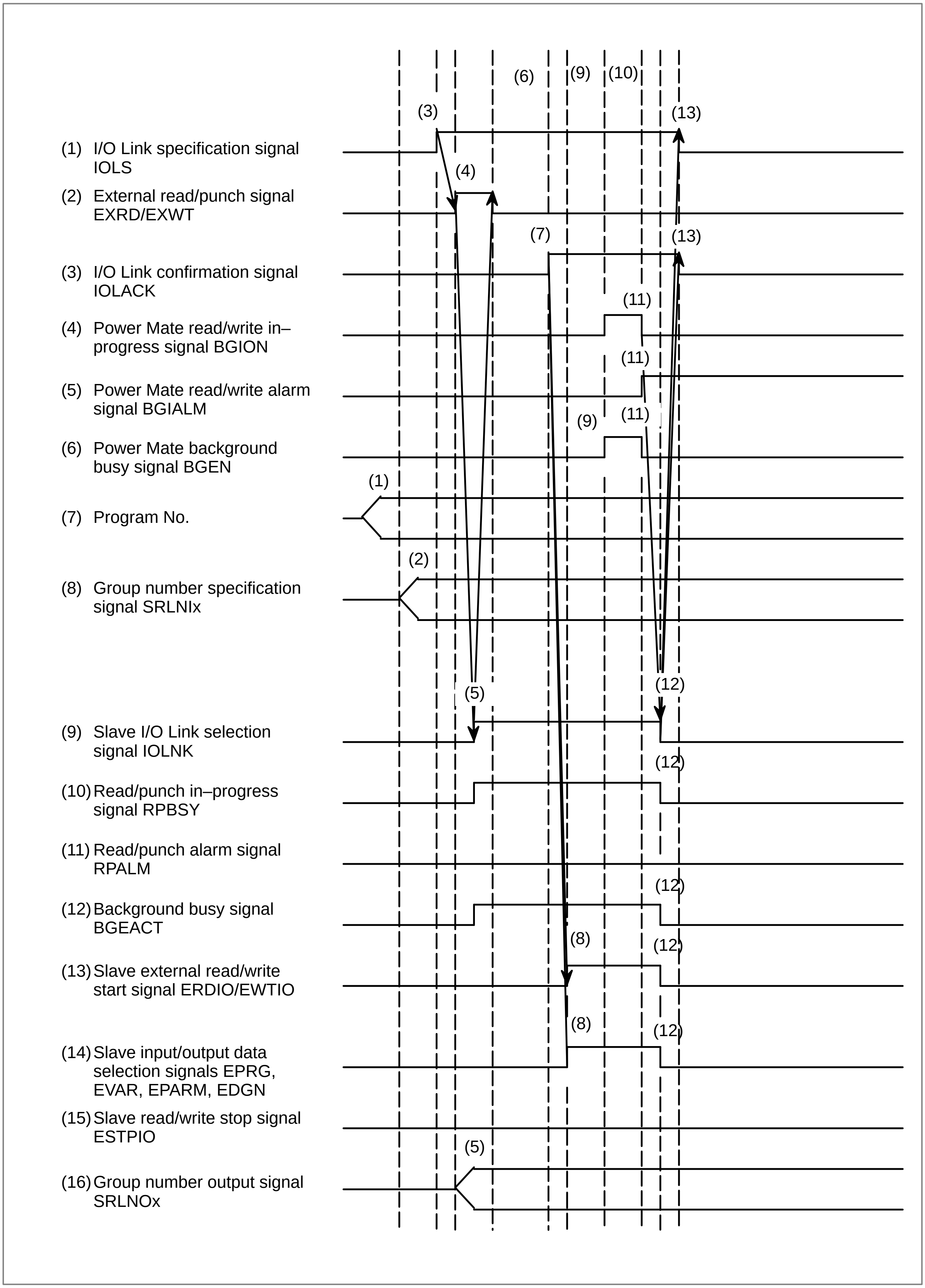
15) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(6)
(9)
(10)
(12)
(3)
(15)
(1) I/O Link specification signal IOLS
(4)
(2) External read/punch signal EXRD/EXWT
(7)
(15)
(3) I/O Link confirmation signal IOLACK
(13)
(4) Power Mate read/write in–progress signal BGION
(13)
(5) Power Mate read/write alarm signal BGIALM
(9)
(6) Power Mate background busy signal BGEN
(1)
(7) Program No.
(2)
(8) Group number specification signal SRLNIx
(9)
(14)
(9) Slave I/O Link selection signal IOLNK
(11)
(10) Read/punch in–progress signal RPBSY
(11)
(11) Read/punch alarm signal RPALM
(11)
(12) Background busy signal BGEACT
(8)
(11)
(13) Slave external read/write start signal ERDIO/EWTIO
(8)
(11)
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(11)
(14)
(15) Slave read/write stop signal ESTPIO
(5)
(16) Group number output signal SRLNOx

(3) When an alarm is issued by the Power Mate

Steps 1) to 10) are the same as those for ordinary input/output.

11) When the Power Mate issues an alarm, Power Mate read/write alarm signal BGIALM is set to 1, and Power Mate read/write in–progress signal BGION is set to 0.
12) The Series 16*i*/18*i*/21*i* sets slave I/O Link selection signal IOLNK and slave read/write stop signal ESTPIO to 0.
13) I/O Link specification signal IOLS and I/O Link confirmation signal IOLACK are set to 0.


(6)
(9)
(10)
(3)
(13)
(1) I/O Link specification signal IOLS
(4)
(2) External read/punch signal EXRD/EXWT
(7)
(13)
(3) I/O Link confirmation signal IOLACK
(11)
(4) Power Mate read/write in–progress signal BGION
(11)
(5) Power Mate read/write alarm signal BGIALM
(9)
(11)
(6) Power Mate background busy signal BGEN
(1)
(7) Program No.
(2)
(8) Group number specification signal SRLNIx
(5)
(12)
(9) Slave I/O Link selection signal IOLNK
(12)
(10) Read/punch in–progress signal RPBSY
(11) Read/punch alarm signal RPALM
(12)
(12) Background busy signal BGEACT
(8)
(12)
(13) Slave external read/write start signal ERDIO/EWTIO
(8)
(12)
(14) Slave input/output data selection signals EPRG, EVAR, EPARM, EDGN
(15) Slave read/write stop signal ESTPIO
(5)
(16) Group number output signal SRLNOx

- **Troubleshooting**

The data input/output function based on the FANUC I/O Link is implemented by various elements such as ladder programs, I/O Link assignment, Series 16*i*/18*i*/21*i* parameters, and Power Mate parameters. So, problems may occur when the function is started.

The table below lists the symptoms of the problems that may occur, together with their causes and corresponding corrective actions.

Symptom	Cause and corrective action
Data input/output is started, but the Series 16*i*/18*i*/21*i* and Power Mate return no response.	The background is already active (the background busy signal BGEACT is set to 1). In this case, the function cannot be used. Examples: 1) In MDI mode, the program screen is selected. 2) Background editing has already started.
	Communication via the RS–232–C interface is in progress. Multiple communication operations cannot be performed at the same time, such that any subsequent attempt to start the function is ignored.
	The selected slave is not a Power Mate. Data input/output function B based on the I/O Link is not started if the slave specified with a group number set with group number specification signals SRLNI0 to SRLNI3 is other than a Power Mate.
	A ladder program error may prevent the function from being started. Check the ladder program for errors such as mis–specified addresses.
When an attempt is made to output data to a Power Mate: Series 16*i*/18 *i*/21*i*: OUTPUT blinks continuously. Power Mate : LSK blinks continuously . (Caution)	Power Mate DO signal RPBSY is not correctly passed to Power Mate read/write in–progress signal RPBSY of the Series 16*i*/18*i*/21*i*.

Symptom	Cause and corrective action
When an attempt is made to output data to a Power Mate: Series 16*i*/18*i*/21*i*: OUTPUT blinks continuously. Power Mate : No response is returned. (Caution)	I/O Link confirmation signal IOLACK is not set to 1. A ladder program error, I/O Link assignment error may prevent.
When an attempt is made to read data from a Power Mate: Series 16*i*/18*i*/21*i* : LSK blinks continuously. Power Mate : No response is returned. (Caution)	
When an attempt is made to output parameters to a Power Mate: Series 16*i*/18*i*/21*i*: OUTPUT blinks continuously. Power Mate : No response is returned. (Caution)	PWE of the Power Mate is set to 0. To modify the Power Mate parameters, PWE must be set to 1. (Caution)
When an attempt is made to output diagnostic data to a Power Mate: Series 16*i*/18*i*/21*i*: OUTPUT blinks continuously. Power Mate : No response is returned. (Caution)	DWE of the Power Mate is set to 0. To enable the modification of Power Mate diagnostic data, DWE must be set to 1. (Caution)

Symptom	Cause and corrective action
When an attempt is made to output macro variables to a Power Mate: Series 16*i*/18*i*/21*i* : Alarm 86 is issued. Power Mate : Alarm 1 is issued.	Bit 1 (ISO) of parameter No. 0000 is set to 1 (EIA code). The EIA code does not include #, such that an ISO code must be set.
When an attempt is made to read macro variables or diagnostic data from a Power Mate: Series 16*i*/18*i*/21*i* : LSK blinks continuously. Power Mate : No response is returned. (Caution)	A start number and the number of data items to be transferred are not set correctly in Power Mate DI signals G100 to G103. Examples: 1 A ladder program error or I/O allocation error prevented a start number and the number of data items from being transferred to Power Mate DI signals G100 to G103. 2 An invalid start number is specified, or the specified number of data items to be transferred is invalid.

CAUTION

If these symptoms are detected, the Series 16*i*/18*i*/21*i* waits for a condition to be satisfied in its internal processing. While such a state exists, the screen is not updated. So, the states of signals cannot be checked on a real–time basis on a screen such as the PMC diagnostic screen.

Signal

Power Mate read/write in–progress signal BGION <G092#2>

[Classification] Input signal

[Function] This signal indicates that the Power Mate is performing data input/output.

[Operation] This signal is set to 1 when the Power Mate is performing data input/output.
This signal is a Power Mate state signal. The corresponding Power Mate side signal is RPBSY <F223#2, F053#2>.
(the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Power Mate read/write alarm signal BGIALM <G092#3>

[Classification] Input signal

[Function] This signal indicates that an alarm has been issued while the Power Mate was performing data input/output.

[Operation] This signal is set to 1 upon the issue of an alarm while the Power Mate is performing data input/output.
This signal is a Power Mate state signal. The corresponding Power Mate side signal is RPALM <F223#3, F053#3>.
(the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Power Mate background busy signal BGEN <G092#4>

[Classification] Input signal

[Function] This signal indicates that the Power Mate is performing background editing.

[Operation] This signal is set to 1 when the Power Mate is performing background editing.
This signal is a Power Mate state signal. The corresponding Power Mate side signal is BGEACT <F223#4, F053#4>.
(the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

I/O Link confirmation signal IOLACK <G092#0>

[Classification] Input signal

[Function] This signal indicates that the Power Mate state signals are valid.

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

– All Power Mate state signals become valid.

For data input/output between the Series 16*i*/18*i*/21*i* and Power Mate, the Power Mate state signals are used. For this purpose, ladder program processing is required. Upon the completion of ladder program processing, I/O Link confirmation signal IOLACK is set to 1 to make the Power Mate state signals active.

I/O Link specification signal IOLS <G092#1>

[Classification] Input signal

[Function] This signal specifies whether those signals that are shared by the external I/O device control function are to be used with data input/output function B based on the I/O Link.

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

– The signals (EXRD, EXSTP, EXWT, RPBSY, RPALM, and BGEACT) for the external I/O device control function are used with data input/output function B based on the I/O Link.

NOTE
I/O Link specification signal IOLS is not used with the ordinary data input/output function based on the I/O Link.

Group number specification signals SRLNI0 to SRLNI3 <G091#0 to #3>

[Classification] Input signal

[Function] These signals specify the group number of the Power Mate that acts as a slave when data input/output function B based on the I/O Link is used.

[Operation] The group number of the Power Mate that acts as a slave is specified with the values of four digits binary code signals.

NOTE
Group number specification signals SRLNI0 to SRLNI3 are not used with the ordinary data input/output function based on the I/O Link.

Slave I/O Link selection signal IOLNK <F177#0>

[Classification] Output signal

[Function] This signal instructs the Power Mate to perform data input/output based on the I/O Link.

[Output condition] This signal is set to 1 in the following case:

– When data input/output is performed

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is IOLNK <G099#7/G251#0> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave external read start signal ERDIO <F177#1>

[Classification] Output signal

[Function] This signal indicates that the Series 16*i*/18*i*/21*i* has started data output.

[Output condition] This signal is set to 1 in the following case:

– When data output is started

This signal is set to 0 in the following case:

– When data output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EXRD <G098#1/G058#1> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave read/write stop signal ESTPIO <F177#2>

[Classification] Output signal

[Function] This signal forcibly terminates Power Mate data input/output.

[Output condition] This signal is set to 1 in the following cases:

– When the Series 16*i*/18*i*/21*i* issues an alarm

– When data input/output function B based on the I/O Link is used, and external read/punch stop signal EXSTP is set to 1

This signal is set to 0 in the following case:

–When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EXSTP <G098#2/G058#2> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave external write start signal EWTIO <F177#3>

[Classification] Output signal

[Function] This signal indicates that the Series 16*i*/18*i*/21*i* has started data input.

[Output condition] This signal is set to 1 in the following case:

– When data input is started

This signal is set to 0 in the following case:

– When data input is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EXWT <G098#3/G058#3> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave program selection signal EPRG <F177#4>

[Classification] Output signal

[Function] This signal notifies the Power Mate that the input/output data consists of programs.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of programs, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EPRG <G098#4/G251#4> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave macro variable selection signal EVAR <F177#5>

[Classification] Output signal

[Function] This signal notifies the Power Mate that the input/output data consists of macro variables.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of macro variables, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EVAR <G098#5/G251#5> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave parameter selection signal EPARM <F177#6>

[Classification] Output signal

[Function] This signal notifies the Power Mate that the input/output data consists of parameters.

[Output condition] This signal is set to 1 in the following case:

– When the input/output data consists of parameters, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EPARM <G098#6/G251#6> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Slave diagnostic data selection signal EDGN <F177#7>

[Classification] Output signal

[Function] This signal notifies the Power Mate that input/output data consists of diagnostic (PMC) data.

[Output condition] This signal is set to 1 in the following case:

– When input/output data consists of diagnostic (PMC) data, and data input/output has been started

This signal is set to 0 in the following case:

– When data input/output is terminated

This signal is a Power Mate control signal. The corresponding Power Mate side signal is EDGN <G098#7/G251#7> (the second address being for Power Mate–MODEL D, F, H, and the first address for all other models).

Group number output signals SRLNO0 to SRLNO3 <F178#0 to #3>

[Classification] Output signal

[Function] These signals indicate the group number of the Power Mate that is acting as a slave.

[Operation] The group number of the Power Mate that is acting as a slave is specified with the values of four binary code signals.
These signals become active when the slave I/O Link selection signal IOLNK is set to 1.

External read start signal EXRD <G058#1>

External punch start signal EXWT <G058#3>

External read/punch stop signal EXSTP <G058#2>

Background busy signal BGEACT <F053#4>

Read/punch in–progress signal RPBSY <F053#2>

Read/punch alarm signal RPALM <F053#3>

The signals listed below are used with data input/output function B based on the I/O Link. For details, see Section 13.5.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G058					EXWT	EXSTP	EXRD	
G091					SRLNI3	SRLNI2	SRLNI1	SRLNI0
G092				BGEN	BGIALM	BGION	IOLS	IOLACK
F053				BGEACT	RPALM	RPBSY		
F177	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLINK
F178					SRLNO3	SRLNO2	SRLNO1	SRLNO0

Parameter

0020	I/O CHANNEL: Selection of an input/output device

Setting entry is acceptable.

[Data type] Byte

[Valid data range] 0 to 35

This CNC provides the following interfaces for data transfer to and from the host computer and external input/output devices:

– Input/output device interface (RS–232–C serial port 1, 2)

– Remote buffer interface (RS–232–C/RS–422)

– DNC1/DNC2 interface

In addition, data can be transferred to and from the Power Mate via the FANUC I/O Link. (This parameter need not be set when data input/output function B based on the I/O Link is used.)

This parameter is used to select the interface used to trnsfer data to and from an input/output device.

Setting	Description
0, 1	RS–232–C serial port (connector JD36A on motherboard)
2	RS–232–C serial port (connector JD36B on motherboard)
3	Remote buffer interface (connector JD28A (RS–232–C interface) or connector JD6A (RS–422 interface) on serial communication board)
4	Memory card interface
5	Data server interface
10 11	DNC1/DNC2 interface, OSI–Ethernet DNC1 interafce #2
20 21 22 | 34 35	Group 0 Group 1 Group 2 | Group 14 Group 15 Data is transferred between the CNC and Power Mate in group n (n: 0 to 15) via the FANUC I/O Link.

NOTE
An input/output device can also be selected using the setting screen. Usually the setting screen is used.

8760	Program number for data registration (data input/output function using the I/O link)

[Data type] Word

[Valid data range] 0 to 9999

When the data input/output function using the I/O link is used, this parameter sets the program numbers of the programs to be used for registering data (parameters, macro variables, and diagnostic data) from Power Mates.

For a Power Mate in group n, the following program numbers are used:
For parameters: Setting + n × 10 + 0
For macro variables: Setting + n × 10 + 1
For diagnostic data: Setting + n × 10 + 2

Example: When 8000 is set

8000: Parameters of group 0 (I/O channel = 20)
8001: Macro variables of group 0 (I/O channel = 20)
8002: Diagnostic data of group 0 (I/O channel = 20)
8010: Parameters of group 1 (I/O channel = 21)
8011: Macro variables of group 1 (I/O channel = 21)
8012: Diagnostic data of group 1 (I/O channel = 21)
8020: Parameters of group 2 (I/O channel = 22)
8021: Macro variables of group 2 (I/O channel = 22)
8022: Diagnostic data of group 2 (I/O channel = 22)

8150: Parameters of group 15 (I/O channel = 35)
8151: Macro variables of group 15 (I/O channel = 35)
8152: Diagnostic data of group 15 (I/O channel = 35)

NOTE
When 0 is set, the input/output of parameters, macro variables, and diagnostic data cannot be performed, but program input/output processing is performed.

Alarm and message

Number	Message	Description
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, or FANUC I/O Link an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective. Or, the slave is not ready to perform data input/output (or the slave is performing background editing) if this alarm is issued when data input/output using the FANUC I/O Link is started. Alternatively, slave–side input/output is stopped (with alarm output) if this alarm is issued during input/output.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective. Alternatively, for data read using the FANUC I/O Link, the master directs the termination of a read operation, but the slave does not stop output.

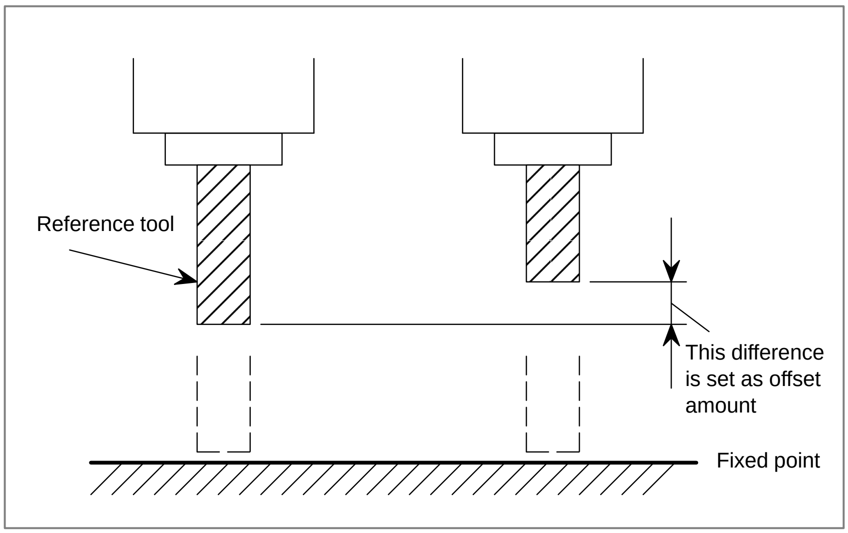
14 MEASUREMENT

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. Then select the reference tool and set it at the fixed point on the machine by manual operation. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by manual operation. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts.



Reference Item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.2	Tool Length Measurement
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.4.2	Tool Length Measurement

14.2 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES)

General

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset.

Signal

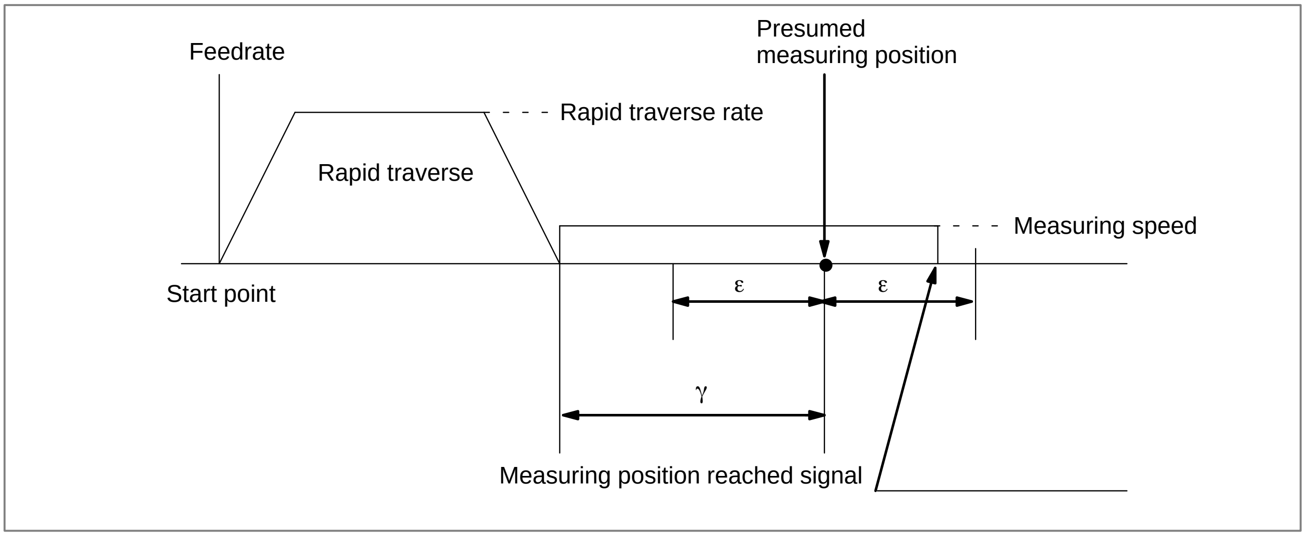
Measuring position reached signals XAE<X004#0>, YAE<X004#1>, ZAE<X004#2>(M series) XAE<X004#0>, ZAE<X004#1>(T series)

[Classification] Input signal

[Function] If the measuring position specified by a program command differs from the measuring position which a tool has reached in practice, that is, the position at the moment the measuring position reached signal has just been turned “1”, the difference in the coordinate value is added to the current tool compensation value to update the compensation value. The tool is first fed to the specified measuring position by rapid traverse in a block where one of the following commands has been specified:

G37 (M series)
G36, G37 (T series)

The tool decelerates and temporarily stops at the distance γ before the measuring position.
The tool then moves to the measuring position at the speed preset by a parameter No. 6241. If the measuring position reached signal corresponding to the G code is turned “1” after the tool has approached within distance ε of the measuring position and before the tool overshoots the measuring position by distance ε, the control unit updates the compensation value and terminates the move command for the block. If the measuring position reached signal is not turned “1” even after the tool has overshot the measuring position by distance ε, the control unit enters an alarm state and terminates the move command for the block without updating the compensation value.



[Operation] When the signal is turned “1”, the control unit works as follows:

- Reads the position of the tool along the axis currently specified and updates the current compensation value based on the difference between the specified measuring position and the read measuring position in the following case: When the measuring position reached signal corresponding to the G code is turned on in a block where G36 (T series) or G37 is specified after the tool is within distance ε of the measuring position specified by a program and before the tool overshoots the measuring position by distance ε. The control unit then stops the tool, and terminates the move command for the block.
- Enters an alarm state and terminates the move command for the block without updating the compensation value in the following case: When the measuring position reached signal corresponding to the command is turned “1” in a block where G36 (T series), G37 is specified after the tool is within distance γ of the measuring position but before the tool is within distance ε of the measuring position.
- The control unit does not monitor the measuring position reached signal for its rising edge but monitors the state of the signal. If the signal remains “1” when the next corresponding automatic tool length measurement (automatic tool offset) is specified, the control unit enters an alarm state when the tool is within distance γ of the measuring position.

NOTE
1 The measuring position reached signal requires at least 10 msec.
2 The CNC directly inputs the measuring position reached signals from the machine tool; the PMC does not process them.
3 If automatic tool offset 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
X004							ZAE	XAE
						ZAE	YAE	XAE

Parameter

6241	Feedrate during measurement of automatic tool offset
	Feedrate during measurement of tool length automatic measurement

[Data type] Word type

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Metric input	1 mm/min	6 to 15000	6 to 12000
Inch input	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

This parameter sets the feedrate during measurement of automatic tool offset (T series) and tool length automatic compensation (M series).

6251	γ value on X axis during automatic tool offset
	γ value during tool length automatic measurement
6252	γ value on Z axis during automatic tool offset

[Data type] Two–word type

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the γ value during automatic tool offset (T series) or tool length automatic measurement (M series).

CAUTION
Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

6254	ε value on X axis during automatic tool offset
	ε value during tool length automatic measurement
6255	ε value on Z axis during tool automatic offset

[Data type] Two–word type

[Unit of data]

Increment system	IS–A	IS–B	IS–C	Unit
Metric input	0.01	0.001	0.0001	mm
Inch input	0.001	0.0001	0.00001	inch

[Valid data range] 1 to 99999999

These parameters set the ε value during automatic tool offset (T series) or tool length automatic measurement (M series).

CAUTION
Set a radius value irrespective of whether the diameter programming or the radius programming is specified.

Alarm and message

Number	Message	Description
080	G37 ARRIVAL SIGNAL NOT ASSERTED (M series)	In the automatic tool length measurement function (G37), the measurement position reached signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT ASSERTED (T series)	In the automatic tool offset function (G36, G37), the measurement position reached signal (XAE or ZAE) is not turned on within an area specified in parameter 6254, and 6255 (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37 (M series)	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
	OFFSET NUMBER NOT FOUND IN G37 (T series)	Automatic tool offset (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.

Number	Message	Description
082	H–CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool measurement (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
	T–CODE NOT ALLOWED IN G37 (T series)	T code and automatic tool offset (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37 (M series)	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
	ILLEGAL AXIS COMMAND IN G37 (T series)	In automatic tool offset (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.

Note

NOTE

1 Measurement speed, γ, and ε are set as parameters. ε must be positive numbers and keep condition of $\gamma > \varepsilon$.

2 The compensation value is updated by the following formula:
New compensation value =(Current compensation value)+[(Current position of the tool along the specified axis when the measuring position reached signal is "1") – (specified measuring position)]
The following compensation values are updated:
(1) In a M series, the compensation value corresponding to the tool compensation number selected by an H code.
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

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.14.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.14.6	AUTOMATIC TOOL OFFSET (G36, G37)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	II.14.2	AUTOMATIC TOOL LENGTH MEASUREMENT (G37)
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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 #5068, as follows:

#5061 Fisrt axis coordinate value
#5062 Second axis coordinate value
#5063 3rd axis coordinate value
:
:
#5068 8th axis coordinate value

Signal

Skip Signal SKIP<X004#7> SKIPP<G006#6> (T series)

[Classification] Input signal

[Function] This signal terminates skip cutting. That is, the position where a skip signal turns to “1” in a block containing G31 is stored in a custom macro variable, and the move command of the block is terminated at the same time.

[Operation] When a skip signal turns to “1”, the control unit works as described below.

· When a block contains a skip cutting command G31, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

· The skip signal is monitored not for a rising edge, but for its state. So, if a skip signal continues to be “1”, a skip condition is assumed to be satisfied immediately when the next skip cutting is specified.

NOTE
1 The skip signal width requires at least 10 msec.
2 The CNC directly reads the skip signal SKIP<X004#7> from the machine tool; the PMC no longer requires to process the signal.
3 If the skip function G31 is not used, the PMC can use the signal terminal SKIP<X004#7> corresponding to the skip signal as a general purpose input signal.

Signal address

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

	#7	#6	#5	#4	#3	#2	#1	#0
G006		SKIPP						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	GSK
	SKF						SK0	

[Data type] Bit type

GSK In skip cutting (G31), the signal SKIPP (bit 6 of G006) is:

0 : Not used as a skip signal.
1 : Used as a skip signal.

SK0 This parameter specifies whether the skip signal is made valid under the state of the skip signal SKIP (bit 7 of X004) and the multistage skip signals (bits 0 to 7 of X004) (for the T series only).

0 : When these signals are 1, they are assumed to be input (skip).
1 : When these signals are 0, they are assumed to be input (skip).

SKF Dry run, override, and automatic acceleration/deceleration for G31 skip command

0 : Disabled
1 : Enabled

Alarm and message

Number	Message	Description
035	CAN NOT COMMANDED G31 (T series)	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
036	CAN NOT COMMANDED G31 (M series)	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.

Warning

WARNING

Disable feedrate override, dry run, and automatic acceleration/deceleration (enabled with parameter No. 6200#7 SKF=1) when the feedrate per minute is specified, allowing for reducing an error in the position of the tool when a skip signal is input. These functions are enabled when the feedrate per rotation is specified.

Note

NOTE

1 The G31 block is always set to G01 mode. The feedrate is specified by an F code.

2 When the measuring motion is made by utilizing the skip signal, program a constant feedrate; otherwise, if the feedrate changes, the measuring error will be noticeable. With a constant feedrate, the maximum measuring error can be calculated as follows:

$$\mathrm{ERRmax} = \mathrm{Fm} \times \frac{1}{60} \times \frac{4}{1000}$$

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:

$$\mathrm{Qmax} = \mathrm{Fm} \times \frac{1}{60} \times \frac{1}{1000} \left(20^{\#1} + \mathrm{Tc} + \mathrm{Ts}\right)$$

Qmax : Overtravel amount (mm or inch)
Fm : Feedrate (mm/min or inch/min)
Tc : Cutting time constant (ms)
Ts : Servo time constant (ms) (1 loop gain)

#1 : The value becomes 28 when the skip signal SKIPP <G006#6> is used. (Also it changes according to the processing time of ladder program).

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Machining Center) (B–63014EN)	II.4.16	SKIP FUNCTION(G31)
	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.4.13	SKIP FUNCTION(G31)
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Machining Center) (B–63094EN)	II.4.8	SKIP FUNCTION(G31)
	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.4.10	SKIP FUNCTION(G31)

14.3.2 High–speed Skip Signal

General

The skip function operates based on a high-speed skip signal (HDI0 ~ HDI7 : connected directly to the CNC; not via the PMC) instead of an ordinary skip signal (X004#7). In this case, up to eight signals can be input. The Series 21*i*/210*i*, however, uses just a single high–speed skip signal (HDI0).

Delay and error of skip signal input is 0 ~ 2 msec at the CNC side (not considering those at the PMC side).

This high-speed skip signal input function keeps this value to 0.004 msec or less, thus allowing high precision measurement.

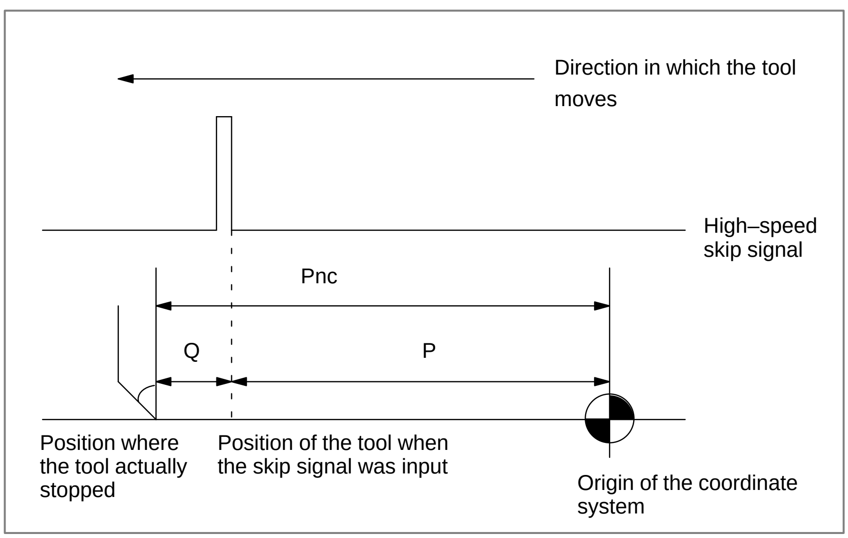
Acceleration / Deceleration and Servo Delay Compensation (Type A/B)

The skip function causes the NC to memorize the “current position” of the tool when a skip signal is input. However, the “current position” includes a delay in the servo system. In other words, the “current position” deviates by the distance corresponding to the servo delay from the position where the tool actually was when the skip signal was input. This deviation can be calculated from the positional error in the servo system and the number of remaining pulses due to feedrate acceleration/ deceleration in the NC. Taking this deviation into account eliminates the necessity to include the servo delay in a measurement error.

The deviation of the “current position” can be compensated for by either of the following two types, using parameter SEA (bit 0 of parameter No. 6201) or parameter SEB (bit 1 of parameter No. 6201).

(1) Type A: The deviation is calculated from the cutting time constant and the servo time constant (loop gain).

(2) Type B: The deviation is assumed to be a sum of the number of remaining pulses due to acceleration/deceleration caused when the skip signal is turned on, and the positional error.



Pnc : Position where the tool actually stopped after the skip signal was input [mm/inch]

P : Distance to be measured [mm/inch]

Q : Servo delay [mm/inch]

Under the conditions shown above, the NC calculates the following equation using parameter SEA (bit 0 of parameter No. 6201) or SEB (bit 1 of parameter No. 6201):

$$P = Pnc - Q$$

For type A (SEA bit 0 of parameter No. 6201 is "1"), the deviation is calculated by:

$$Q = Fm \times 1/60 \times (\alpha \times Tc/1000 + Ts/1000)$$

where

Fm : Feedrate [mm/min or inch/min]

Tc : Cutting time constant [msec]

Parameter No. 1622: Exponential acceleration/deceleration

Parameter No. 1628: Linear acceleration/deceleration after interpolation

If parameter SKF (bit 7 of parameter No. 6200) = 0, Tc = 0.

Ts : Servo time constant [msec]

Assuming that the loop gain (parameter No. 1825) is G (unit: 1/s):

Ts = 1000/G

α : = 1 Exponential acceleration/deceleration

= 1/2 Linear acceleration/deceleration after interpolation

NOTE

For type A (parameter SEA (No. 6201 #0)=1), the skip signal must be turned on when the tool moves at constant feedrate.

Signal

High Speed Skip Staus Signal HDO0 to HDO7<F122>

[Classification] Output signal

[Function] This signal informs the PMC of the input status of the high-speed skip signal. The signal-to-bit correspondence is as follows:

High-speed skip signal		Bit name
HDI0	——	HDO0
HDI1	——	HDO1
HDI2	——	HDO2
HDI3	——	HDO3
HDI4	——	HDO4
HDI5	——	HDO5
HDI6	——	HDO6
HDI7	——	HDO7

[Output condition] Each bit is set to 1 when:

- The corresponding high-speed skip signal is logical 1.

Each bit is set to 0 when:

- The corresponding high-speed skip signal is logical 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Data type] Bit type

HSS 0 : The skip function does not use high-speed skip signals.

1 : The skip function uses high-speed skip signals.

SRE When a high-speed skip signal is used:

0 : The signal is considered to be input at the rising edge (0 → 1).
1 : The signal is considered to be input at the falling edge (1 → 0).

	#7	#6	#5	#4	#3	#2	#1	#0
6201				IGX			SEB	SEA

[Data type] Bit type

SEA When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:

0 : Ignored.
1 : Considered and compensated (type A).

SEB When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:

0 : Ignored.
1 : Considered and compensated (type B).

IGX When the high-speed skip function is used, SKIP (bit 7 of X004), SKIPP (bit 6 of G006), and +MIT1 to –MIT2 (bits 2 to 5 of X004) are:

0 : Enabled as skip signals.
1 : Disabled as skip signals.

NOTE
SKIPP (bit 6 of G006) and +MIT1 to –MIT2 (bits 2 to 5 of X004) are enabled only when bit 0 (GSK) of parameter No. 6200 is set to 1 and bit 3 (MIT) of parameter No. 6200 is set to 1. Note also that these signals are enabled only for the T series.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1

1S1~1S8

Specify which high-speed skip signal is enabled when the G31 skip command is issued. The bits correspond to the following signals:

0 : The skip signal corresponding to the bit is disabled.
1 : The skip signal corresponding to the bit is enabled.

1S1 —— HDI0
1S2 —— HDI1
1S3 —— HDI2
1S4 —— HDI3
1S5 —— HDI4
1S6 —— HDI5
1S7 —— HDI6
1S8 —— HDI7

NOTE
HDI0 to HDI7 are high-speed skip signals.

Reference item

CONNECTION MANUAL (Hardware) (B–63003EN)	7.7	HIGH SPEED DI SIGNAL INTERFACE

14.3.3 Multi–step Skip

General

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable and cancels the remaining distance that the block was supposed to be moved when a skip signal (8 points) or high-speed skip signal (8 points however in the case of Series 21*i*/210*i* 1 point) is turned on.
Also in a block specifying Q1 to Q4 after G04, this function skips a dwell when the skip signal or high speed skip signal has turned on.
A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Signal

Skip signal SKIP, SKIP2 to SKIP8 <X004>

[Classification] Input signal

[Function] These signals terminate skip cutting. That is, the position where a skip signal turns to “1” in a command program block containing G31P1 (or G31), G31P2, or G31P3, G31P4 is stored in a custom macro variable, and the move command of the block is terminated at the same time. Furthermore, in a block containing G04, G04Q1, G04Q2, G04Q3 or G04Q4, the dwell command of the block is terminated.

In either case, until all other commands (such as miscellaneous functions) of the block are completed, machining never proceeds to the next block.

Which of the eight skip signals is applicable to blocks containing the G codes can be determined by parameter (no. 6202 to 6206). The eight skip signals can correspond to the G codes on a one-to-one basis. One skip signal can also be made applicable to multiple G codes. Conversely, multiple skip signals can be made applicable to one G code.

[Operation] When a skip signal turns to “1”, the control unit functions as described below.

· When a block contains a G code from (G31, G31P1 to P4) for skip cutting, and the skip signal is made applicable by parameter setting to the command, the control unit reads and stores the current position of the specified axis at that time. The control unit stops the axis, then cancels the remaining distance that the block was supposed to be moved.

· When a block contains a G04, or G04Q1 to Q4 code for dwell, and the skip signal is made applicable by parameter setting to the command, the control unit stops dwell operation, and cancels any remaining dwell time.

· The skip signal is 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	
X004	SKIP	ESKIP / SKIP6	–MIT2 / SKIP5	+MIT2 / SKIP4	–MIT1 / SKIP3	+MIT1 / SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
	SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	ZAE / SKIP2	YAE / SKIP8	XAE / SKIP7	(M series)

WARNING

1 SKIP2 to SKIP6 are at the same addresses as skip signal ESKIP (axis control by PMC) and axial manual feed interlock signals +MIT1, –MIT1, +MIT2, and –MIT2 (direct input B for tool compensation measurements). Be careful when using both. (T series)

2 SKIP2 and SKIP6 to SKIP8 are at the same addresses as skip signal ESKIP (axis control by PMC) and measurement position arrival signal XAE, YAE, and ZAE (tool length automatic measurement). Be careful when using both. (M series)

CAUTION

If the automatic tool compensation option is used, SKIP5 to SKIP8 cannot be used. (T series)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200			SLS	HSS				

[Data type] Bit type

HSS 0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SLS 0 : The multi–step skip function does not use high-speed skip signals while skip signals are input.
1 : The multi–step skip function uses high-speed skip signals while skip signals are input.

NOTE

Skip signals (SKIP and SKIP2 to SKIP8) do not depend on the setting of this parameter. They are always enabled. And, it is possible to set disable with parameter IGX bit 4 of No. 6201.

	#7	#6	#5	#4	#3	#2	#1	#0
6202	1S8	1S7	1S6	1S5	1S4	1S3	1S2	1S1
6203	2S8	2S7	2S6	2S5	2S4	2S3	2S2	2S1
6204	3S8	3S7	3S6	3S5	3S4	3S3	3S2	3S1
6205	4S8	4S7	4S6	4S5	4S4	4S3	4S2	4S1
6206	DS8	DS7	DS6	DS5	DS4	DS3	DS2	DS1

[Data type] Bit type

1S1~1S8, 2S1~2S8, 3S1~3S8, 4S1~4S8, DS1~DS8

Specify which skip signal is enabled when the skip command (G31, or G31P1 to G31P4) and the dwell command (G04, G04Q1 to G04Q4) are issued with the multi–step skip function.

The following table shows the correspondence between the bits, input signals, and commands.

The settings of the bits have the following meanings:

0 : The skip signal corresponding to the bit is disabled.
1 : The skip signal corresponding to the bit is enabled.

Multi–step skip function

Command / Input signal	G31 G31P1 G04Q1	G31P2 G04Q2	G31P2 G04Q2	G31P4 G04Q4	G04
SKIP/HDI0	1S1	2S1	3S1	4S1	DS1
SKIP2/HDI1	1S2	2S2	3S2	4S2	DS2
SKIP3/HDI2	1S3	2S3	3S3	4S3	DS3
SKIP4/HDI3	1S4	2S4	3S4	4S4	DS4
SKIP5/HDI4	1S5	2S5	3S5	4S5	DS5
SKIP6/HDI5	1S6	2S6	3S6	4S6	DS6
SKIP7/HDI6	1S7	2S7	3S7	4S7	DS7
SKIP8/HDI7	1S8	2S8	3S8	4S8	DS8

NOTE
HDI0 to HDI7 are high-speed skip signals.

Note

NOTE

The skip cutting commands G31 P1, G31 P2, G31 P3, and G31 P4 are all identical, except that they correspond to different skip signals. The tool moves along the specified axis 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.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.17	Multi–step Skip (G31)
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	II.4.14	Multi–step Skip (G31)
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	II.4.11	Multi–step Skip (G31)

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 or the SKIP signal <X0004#7> is input during the execution of G31 P99, the execution of the next block starts without executing the remaining portion of the move command.

When the motor torque limit is reached during the execution of G31 P98, the execution of the next block starts without executing the remaining portion of the move command. (The SKIP signal does not affect the execution of G31 P98.)

When no torque limit is specified before executing G31 P99 or P98, the move command is executed without performing the skip operation.

For G31 P99 and P98, the coordinate, indicating the position to which the tool is to be positioned after skip, is stored in the system variable of the custom macro.

Alarm No. 244 occurs if errors have accumulated to an amount (32767) that cannot be corrected in one distribution before the torque–limit–reached signal is input during the execution of G31 P99 or P98.

Signal

Torque limit reached signals TRQL1 to TRQL8 <F114>

[Classification] Output signal

[Function] Indicates that the torque limit has been reached.

[Output condition] Set to “1” when:

· The torque limit has been reached for the corresponding axis.

Set to “0” when:

· The torque limit has not been reached for the corresponding axis

Numbers 1 to 8 indicate the corresponding axis numbers.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F114	TRQL8	TRQL7	TRQL6	TRQL5	TRQL4	TRQL3	TRQL2	TRQL1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6201					TSA	TSE		

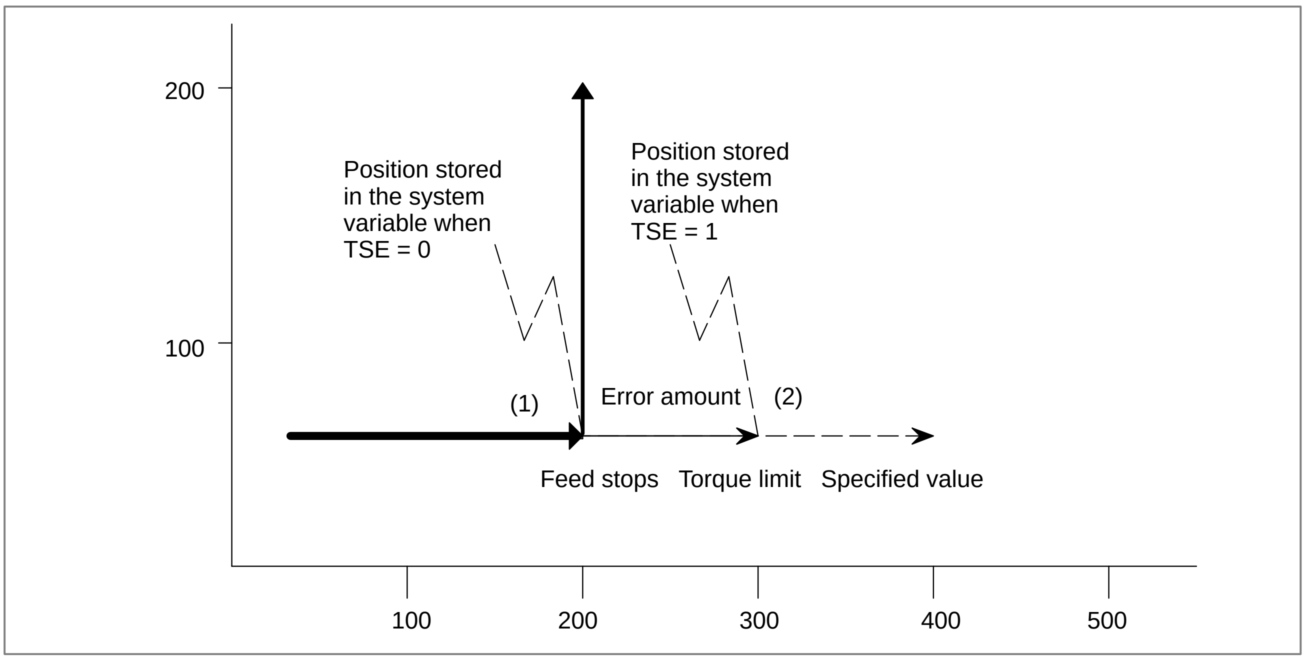
[Data type] Bit type

TSE When a skip operation is performed by the G31 P99 or P98 command used to specify torque limit skip:

0 : Corrects servo errors. (1)
1 : Does not correct servo errors. (2)

TSA Selects the axes to be monitored for whether the torque limit has been reached during the execution of the G31 P99 or 98 command used to specify torque limit skip:

0 : Monitors all axes.
1 : Monitors the axes specified by G31 P99 or P98.



Alarm and message

Number	Message	Description
015	TOO MANY AXES COMMANDED	In the block including the command for the skip function (G31 P99/P98), to be executed under the control of the torque limit reach signal, no axis move command is specified, or two or more axes are specified. In a single block, specify one axis only.
244	P/S ALARM	When the skip function to be executed under the control of the torque limit reach signal is enabled, an error value (32767) that exceeds the maximum return value that can be handled with a single distribution is detected before input of the torque limit reach signal. Retry the processing after changing the axis feedrate, torque limit, or other conditions.

Caution

CAUTION

1 Specify a torque limit before G31 P99/P98. If G31 P99/P98 is executed with no torque limit specified, the move command is executed without a skip operation.

2 When G31 P99 is specified, the SKIP signal causes a skip operation. However, avoid using the high–speed skip operation.

3 Before specifying G31 P99/P98, cancel tool–tip radius compensation with G40.

4 Set the SKF bit (bit 7 of parameter No. 6200) to 0 to disable the dry run, override, and automatic acceleration/deceleration functions for the G31 skip command.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR’S MANUAL (For Lathe) (B–63004EN)	II.4.13	TORQUE LIMIT SKIP
Series 21*i*/210*i*	OPERATOR’S MANUAL (For Lathe) (B–63084EN)	II.4.12	TORQUE LIMIT SKIP

14.3.5 Continuous High–speed Skip Function (M series)

General

The continuous high–speed skip function enables reading of absolute coordinates by using the high–speed skip signals (HDI0 to HDI7). Once a high–speed skip signal has been input in a G31P90 block, absolute coordinates are read into custom macro variables #5061 to #5068. The input of a skip signal does not stops axial movement, thus enabling reading of the coordinates of two or more points.

The rising and falling edges of the high–speed skip signal can be used as a trigger, depending on the parameter CSE (No. 6201#5) setting.

See “14.3.2 High–speed Skip Signal” for details of high–speed skip signal.

• Custom macro variables

Once a high–speed skip signal has been input in a G31P90 block, absolute coordinates are read into custom macro variables #5061 to #5068. These variables are immediately updated once the tool reaches the next skip position. The feedrate must, therefore, be specified such that the tool does not reach the next skip position before the application completes reading of the variables. The PMC program must be created to satisfy this condition.

#5061 Coordinate along the first axis
#5062 Coordinate along the second axis
#5063 Coordinate along the third axis
⋮
#5068 Coordinate along the eighth axis

• High–speed skip signal

This function is enabled only when a high–speed skip signal is used.

The high–speed skip signal to be used is selected with bits 0 to 7 of parameter No. 6208 (9S1 to 9S8).

• End of block

The G31P90 block is terminated when the tool reaches the end point.

• Application (Example)

(1) Reads custom macro variables, using the PMC window function.

(2) Saves the values in the PMC data table (address D).
(The PMC window function cannot be used to save data.)

(3) An execution or auxiliary macro is used for computation. They can directly read PMC address D, using the address function.

Limitations

• Controlled axes

Only one axis can be specified in the block for the continuous high–speed skip function (G31P90). If two or more axes are specified, P/S alarm No. 5068 is issued.

Signal

See Subsec. 14.3.2 High–speed Skip Signal

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
6200		SRE		HSS				

[Data type] Bit type

HSS 0 : The skip function does not use high-speed skip signals.
1 : The skip function uses high-speed skip signals.

SRE When a high-speed skip signal is used:
0 : The signal is considered to be input at the rising edge (0 → 1).
1 : The signal is considered to be input at the falling edge (1 → 0).

	#7	#6	#5	#4	#3	#2	#1	#0
6201			CSE				SEB	SEA

[Data type] Bit type

SEA When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:
0 : Ignored.
1 : Considered and compensated (type A).

SEB When a high speed skip signal goes on while the skip function is used, acceleration/deceleration and servo delay are:
0 : Ignored.
1 : Considered and compensated (type B).

NOTE
Compensation can be performed in either of two ways: type A or type B. The skip signal of the skip function causes the NC to store the current position. The current position, stored in the NC, is subject to the servo delay and differs from the machine position by an amount corresponding to the servo delay. This difference can be determined from the position error of the servo system and the accumulated pulses caused by feed acceleration/deceleration executed by the NC. Provided this difference is considered, the servo delay need not be considered as a measurement error. The difference can be reduced by applying either of the following methods, as determined by the parameter settings.
1) Type A:
A value calculated from the cutting time constant and servo time constant (loop gain) is taken as the difference.
2) Type B:
The value calculated from the position error and the accumulated pulses caused by acceleration/deceleration, when the skip signal is enabled, is taken as the difference.

CSE For continuos high–speed skip command G31 P90, high–speed skip signals are :
0 : Effective at either a rising or falling edge (depending on the setting of bit 6 (SRE) of parameter No. 6200)
1 : Effective for both the rising and falling edges

		#7	#6	#5	#4	#3	#2	#1	#0
6208		9S8	9S7	9S6	9S5	9S4	9S3	9S2	9S1

[Data type] Bit type

9S1 to 9S8 Specify valid high–speed skip signals for high–speed skip command G31P90. The bits correspond to signals as follows:

9S1 —— HDI0
9S2 —— HDI1
9S3 —— HDI2
9S4 —— HDI3
9S5 —— HDI4
9S6 —— HDI5
9S7 —— HDI6
9S8 —— HDI7

Set each bit as follows:

0 : The corresponding skip signal is invalid.
1 : The corresponding skip signal is valid.

6220	Period during which input is ignored for continuous high–speed skip signal

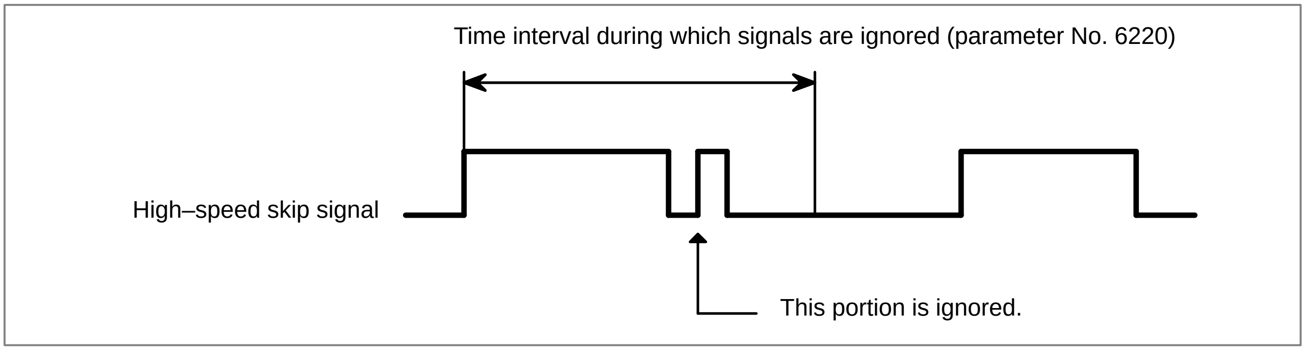
[Data type] Byte type

[Unit of data] 8 msec

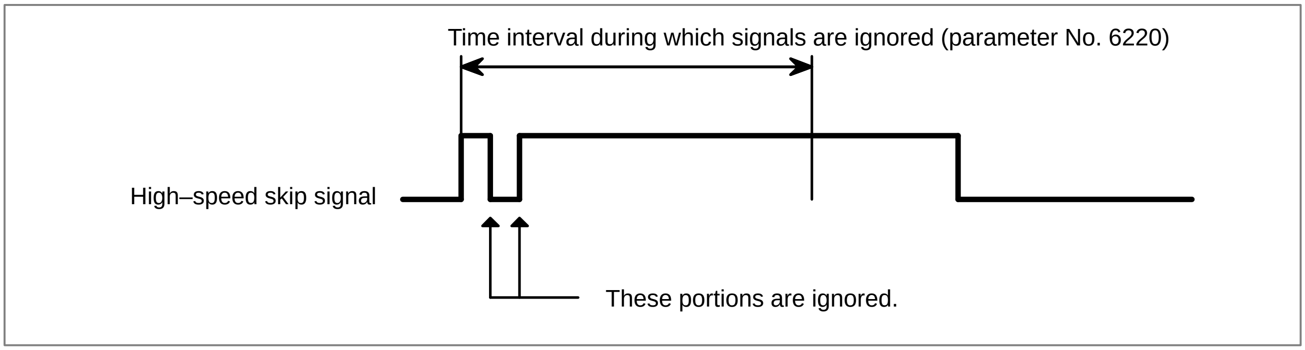
[Valid data range] 3 to 127 ($\times$ 8 msec)

If a value that falls outside this range is specified, 3 ($\times$ 8 msec) is assumed.
This parameter specifies the period that must elapse between a high–speed skip signal being input and input of the next high–speed skip signal being enabled, for the continuous high–speed skip function. This parameter is used to ignore chattering in skip signals.

(Example 1: Time interval specified when CSE (bit 5 of parameter No. 6201) = 0)



(Example 2: Time interval specified when CSE (bit 5 of parameter No. 6201) = 1)



Alarm and message

Number	Message	Contents
5068	G31 P90 FORMAT ERROR	No axis is specified for movement. Two or more axes were specified for movement.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	II.4.19	Continuous high–speed skip signal

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 reference face is made as Z = 0.) When the measured value is entered on the offset value display screen, NC inputs the difference between the input measured value and the coordinate value recorded in NC, as the offset value of the offset number.

If you release the tool without moving the tool in the axis along which an offset value is entered but moves the tool along the other axis, an offset value can be set without using the position record signal.

The workpiece coordinate system can be shifted using the technique of directly inputting the measured value for offset. This technique is used when the coordinate system planned in the program does not match with the coordinate system actually set.

The procedures are the same as those for direct input for offset, except a difference of using the standard tool on the work shift screen.

Signal

Position record signal (PRC) <G040#6>

[Classification] Input signal

[Function] This signal is prepared for the function of input of offset value measured A. It is used to store in the control unit the data on the positions of the tool for tentative cutting. After measuring a dimension of the workpiece, input the measured value by the specified manual operation. The difference is then stored as the specified tool compensation value.

[Operation] The control unit stores the current position along X and Z axes when the signal turns to “1”.

> **NOTE**
> To use this signal, set parameter PRC (No.5005#2) to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G040		PRC						

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

[Data type] Bit type

PRC Direct input of tool offset value and workpiece coordinate-system shift value

0 : Not use a PRC signal
1 : Uses a PRC signal

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.2	Direct Input of Tool Offset Measured
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.4.2	Direct Input of Tool Offset Measured

14.4.2
Input of Tool Offset Value Measured B (T series)

General

When the touch sensor is provided, the tool offset value 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**

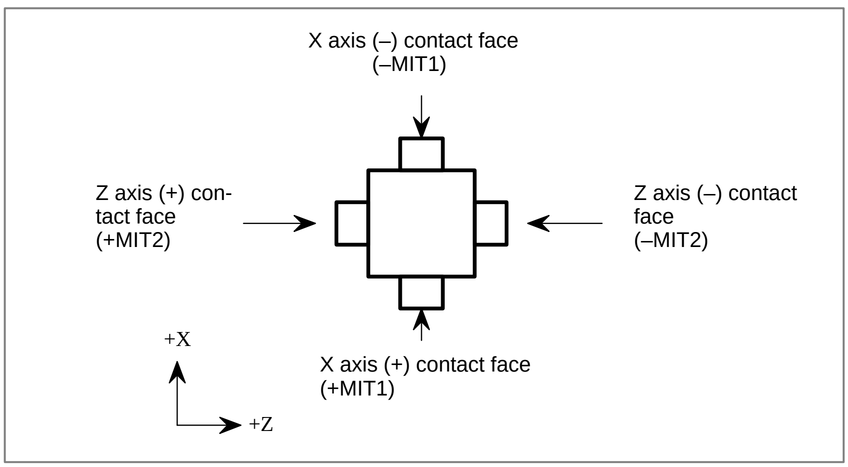
This touch sensor makes contact in two directions in each axis, and outputs four signals when it detects a tool contact. The contact faces are selected according to the tool nose figures to be measured.

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 No. 5015 to 5018. Select the tool whose offset value is to be measured, and bring it to touch the sensor, receiving a contact detection signal (tool compensation value write signal). The mechanical coordinate value is the distance from the tool nose position of the measuring tool at the mechanical reference (home) position to the measuring position; set the difference between this value and the reference value (parameter setting) into the tool offset value memory as the tool geometry offset value. The corresponding tool wear offset value becomes 0.

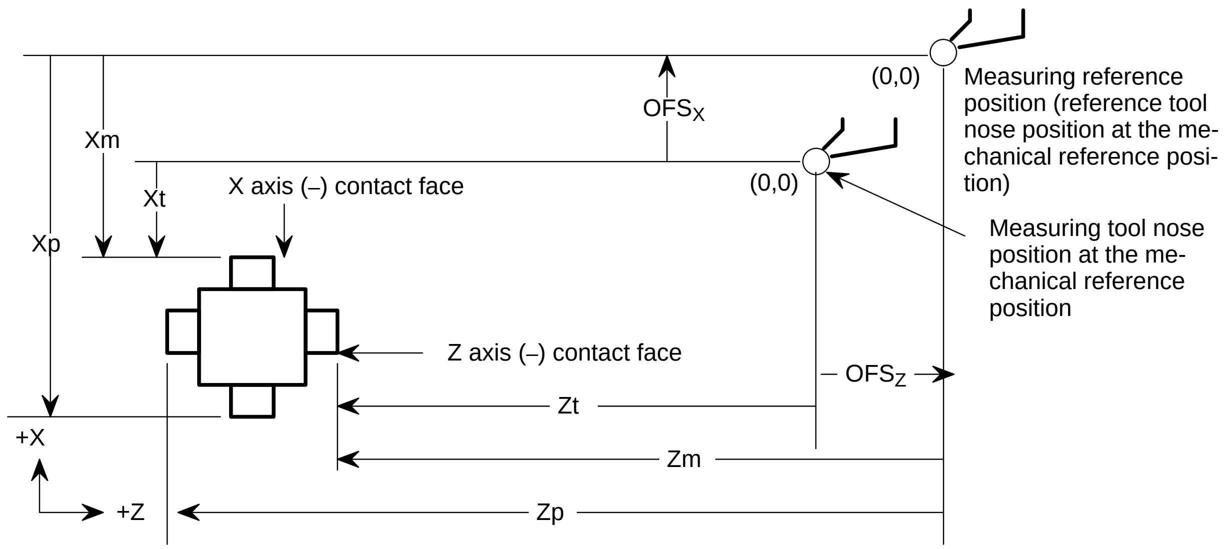
(Tool offset value to be set)

= (Mechanical coordinate value when tool compensation value write signal has become “1”) – (Reference value (parameter value) corresponding to the tool compensation value write signal)

The tool offset value to be set differs according to the method of determining the measuring reference position.

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. 5015)

Xm : Distance from the measuring reference position to X-axis (–) contact face (parameter no. 5016)

Zp : Distance from the measuring reference position to Z-axis (+) contact face (parameter no. 5017)

Zm : Distance from the measuring reference position to Z-axis (–) contact face (parameter no. 5018)

Xt : X-axis direction moving distance of the measuring tool up to the contact face of sensor (X-axis machine coordinate value)

Zt : Z-axis direction moving distance of the measuring tool up to the contact face of sensor (Z-axis machine coordinate value)

(when Xt and Zt touch the X-axis (–) contact face and Z-axis (–) contact face in the above figure)

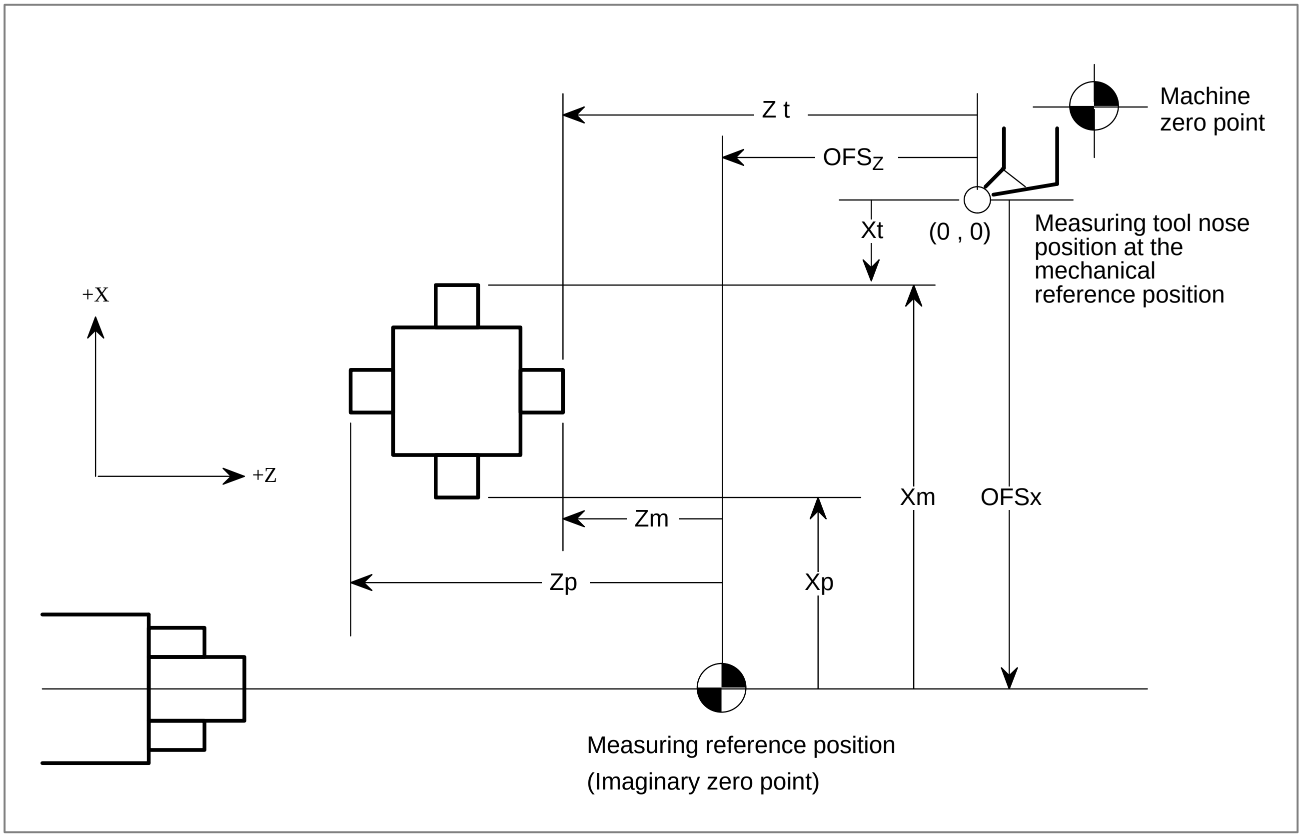
OFSx : Tool offset value to be set (X-axis): OFSx = Xt - Xm

OFSz : Tool offset value to be set (Z-axis): OFSz = Zt - Zm

When the reference tool nose tip position is set as the measuring reference position

Example 2

The measuring reference point may be an imaginary point (imaginary zero point), as shown in the figure below. The difference between the imaginary zero point and the measuring tool nose tip position at the mechanical reference point 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

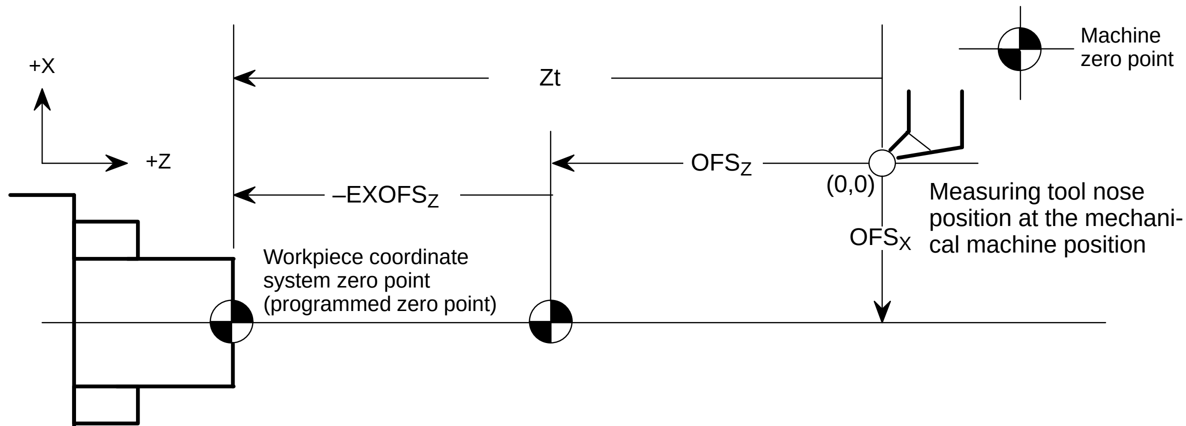
- **Setting the workpiece coordinate system shift amount**

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.



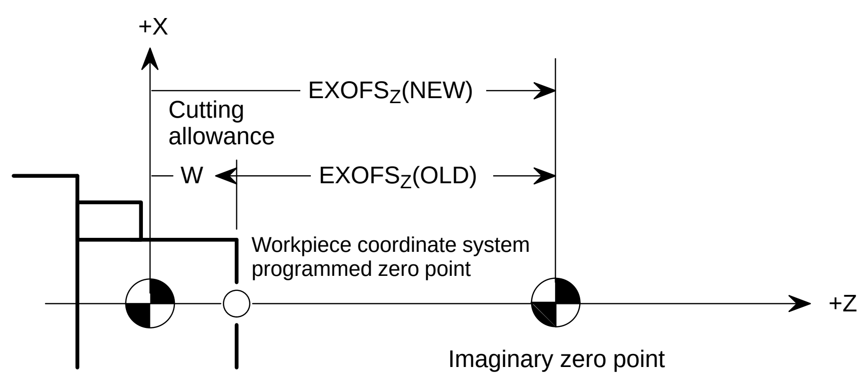
EXOFSz : Workpiece coordinate system shift amount to be set

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.5005#5)=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.

(5) Near the tool to the sensor by manual operation.

(6) Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the tool compensation value writing signals (+MIT1, –MIT1, +MIT2 or –MIT2) to input to CNC.
The tool compensation value writing signal is set to “1”, and the :

i) The axis is interlocked in this direction and its feeding is stopped.

ii) The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.

(7) For both X-axis and Z-axis, their offset 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.5020) in advance.
Besides the tool offset number can be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).

(6) Manually approach the tool to an end face of the workpiece.

(7) Place the tool edge to the end face (sensor) of the workpiece 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 <G039#7>)

[Classification] Input signal

[Function] Selects the mode for writing tool compensation.

[Operation] When this signal is turned “1” in a manual operation mode, the mode for writing tool compensation is selected. The control unit then automatically switches the screen displayed on the CRT to the tool geometry compensation screen and blinks the “OFST” status display 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 <X004#2, #4> –MIT1, –MIT2 <X004#3, #5>

[Classification] Input signal

[Function] Each of these signals inhibits the tool from being fed along the corresponding axis during manual operation. When signal GOQSM for selecting the mode for writing tool compensation is set to "1", the manual feed is inhibited and also the tool geometry compensation along the axis is automatically calculated and the result is set in tool compensation memory.

[Operation] When these signals are turned "1", 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 OFN5, OFN6 <G039#0 to #5, G040#0>

[Classification] Input signal

[Function] Selects the tool offset number.

[Operation] When the mode for writing tool compensation is selected, the cursor is automatically positioned on the tool geometry compensation number selected by these signals.
A tool offset number is specifed by 7–bit binary number. Number 0 to 98 corresponds to the compensation number 1 to 99.

NOTE
This signal is available only when parameter QNI (No. 5005#5) =1.

Workpiece coordinate system shift value write mode select signal WOQSM <G039#6>

[Classification] Input signal

[Function] Selects the mode for writing the shift amount for the workpiece coordinate system.

[Operation] When this signal is turned "1" in a manual operation mode, the mode for writing the shift amount for the workpiece coordinate system is selected. The control unit then automatically switches the screen displayed to the WORK SHIFT screen and blinks the "OFST" status display 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 <G040#7>

[Classification] Input signal

[Function] Automatically calculates and sets the shift amount for the workpiece coordinate system.

[Operation] When this signal turns to "1" in the mode for writing the shift amount for the workpiece coordinate system, it triggers automatic calculation and setting of the shift amount for the workpiece coordinate system.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X004			–MIT2	+MIT2	–MIT1	+MIT1		
			SKIP5	SKIP4	SKIP3	SKIP2		

WARNING
Since the same addresses are used for both +MIT1, –MIT1, +MIT2, –MIT2 and skip signals SKIP2 to SKIP5 (multi–step skip), be careful for using two kinds of signals.

	#7	#6	#5	#4	#3	#2	#1	#0
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0

Parameter

G040	WOSET							OFN6

	#7	#6	#5	#4	#3	#2	#1	#0
3003					DIT			

[Data type] Bit type

DIT Interlock for each axis direction

0 : Enabled
1 : Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
5005			QNI					

[Data type] Bit type

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

5015	Distance (XP) betweeen reference position and X axis + contact surface
5016	Distance (XM) betweeen reference position and X axis – contact surface
5017	Distance (ZP) betweeen reference position and Z axis + contact surface
5018	Distance (ZM) betweeen reference position and Z axis – contact surface

[Data type] Two–word type

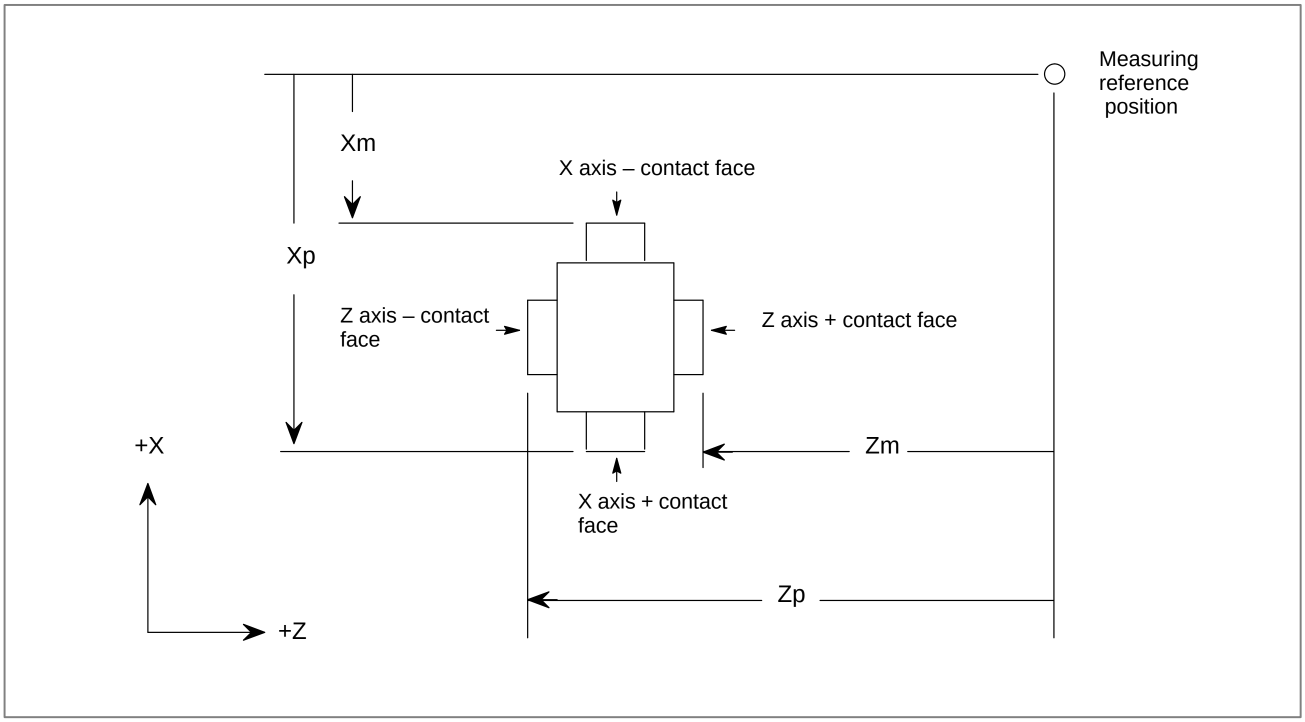
[Unit of data]

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 – 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.



5020	Tool offset number used for the input of tool offset value measured B

[Data type] Byte type

[Valid data range] 0 to the number of tools to be compensated.

Set tool offset number used for the input of tool offset value measured B function (i.e. when workpiece coordinate system shift value is set). (The tool offset number corresponding to the measured tool shall be set in advance.) This parameter is valid when the tool offset number is not selected automatically (QNI, #5 of parameter 5005, is zero).

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.3	Direct Input of Tool Offset Measured B
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	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.

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.7	Input of Measured Workpiece Origin Offsets
	OPERATOR'S MANUAL (For Lathe) (B–63004EN)	III.11.4.11	Input of Measured Workpiece Origin Offsets
Series 21*i*/210*i*	OPERATOR'S MANUAL (For Machining Center) (B–63094EN)	III.11.4.7	Input of Measured Workpiece Origin Offsets
	OPERATOR'S MANUAL (For Lathe) (B–63084EN)	III.11.4.11	Input of Measured Workpiece Origin Offsets

14.5 TOOL LENGTH/ WORKPIECE ORIGIN MEASUREMENT B (M SERIES)

General

Two functions have been provided to measure the tool length:
The automatic tool length measurement function (Section 14.2) automatically measures the tool length at a programmed command (G37); The tool length measurement function (Section 14.1) measures the tool length after the target tool is manually moved until it touches the workpiece top surface or other reference position. In addition to the two conventional functions, tool length/workpiece origin measurement B has been provided to simplify the procedure of tool length measurement. This function facilitates the machining setup, resulting in a reduced machining setup time. The function also makes it easy to measure the workpiece origin offset.

With this function, the operator can program a T/M code or reference position return by specifying a manual numeric command on the tool length offset measurement screen.

Signal

Tool offset measurement mode selection signal GOQSM <G039, #7>

[Classification] Input signal

[Function] Selects tool offset value measurement mode.

[Operation] When the signal goes "1" in manual operation mode, tool offset measurement mode is selected. The screen is automatically replaced with the tool geometry compensation screen. In the status display field at the bottom of the screen, "OFST" blinks to indicate that tool offset measurement mode has been selected.

Tool offset number selection signals OFN0 to OFN5, OFN6 to OFN9 <G039, #0 to #5; G040, #0 to #3>

[Classification] Input signal

[Function] Selects a tool offset number.

[Operation] In tool offset measurement mode, the cursor automatically moves to the tool offset number selected by the tool offset number input signal.

The ten code signals (binary code) select a tool offset number. Code signals 0 to 998 correspond to tool offset numbers 1 to 999.

NOTE
This signal is valid only when the QNI bit (bit of 5 parameter No. 5005) is set to 1.

Workpiece origin offset measurement mode selection signal WOQSM <G039, #6>

[Classification] Input signal

[Function] Selects workpiece coordinate system shift measurement mode.

[Operation] When the signal goes “1” in manual operation mode, workpiece origin offset measurement mode is selected. The screen is automatically replaced with the workpiece coordinate system setting screen. In the status display field at the bottom of the screen, “WOFS” blinks to indicate that workpiece origin offset measurement mode has been selected.

Skip signal SKIP <X004, #7>

[Classification] Input signal

[Function] Stores the momentary position where the signal goes “1” in the workpiece origin offset measurement, as a measurement point. At the same time, the axial movement is stopped.

[Operation] When the skip signal is brought to “1” while the measurement probe is touching the reference plane or the external edge of the reference hole in the workpiece origin offset value measurement, the momentary machine coordinates are stored as the position of the measurement point. At the same time, the axial movement by manual handle feed or jog feed stops.

Until the skip signal is brought to “0”, the interlock prevents the movement in the direction in which the measurement probe was brought to the current measurement point.

Signal addresses

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

	#7	#6	#5	#4	#3	#2	#1	#0
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040					OFN9	OFN8	OFN7	OFN6

Parameter

5005	#7	#6	#5	#4	#3	#2	#1	#0
			QNI					

[Data type] Bit type

QNI When the tool length measurement B function is executed, a tool offset number is selected:

0 : According to the selection the operator makes on an MDI unit (by moving the cursor).
1 : According to the signal input from the PMC.

5007	#7	#6	#5	#4	#3	#2	#1	#0
			WMC	WMH	WMA	TMA	TC3	TC2

[Data type] Bit type

TC3, TC2

TC3	TC2	Description
0	0	The tool–change position is determined by the first reference position return.
0	1	The tool–change position is determined by the second reference position return.
1	0	The tool–change position is determined by the third reference position return.
1	1	The tool–change position is determined by the fourth reference position return.

TMA 0 : The tool length can be measured only on the Z–axis.
1 : The tool length can be measured on any axis.

WMA 0 : The workpiece origin offset from the reference plane can be measured only on the Z–axis.
1 : The workpiece origin offset from the reference plane can be measured on any axis.

WMH 0 : The workpiece origin offset from the reference hole cannot be measured.
1 : The workpiece origin offset from the reference hole can be measured.

WMC 0 : The axis of workpiece origin offset measurement is specified by entering the axis name.
1 : The axis of workpiece origin offset measurement is selected by moving the cursor.

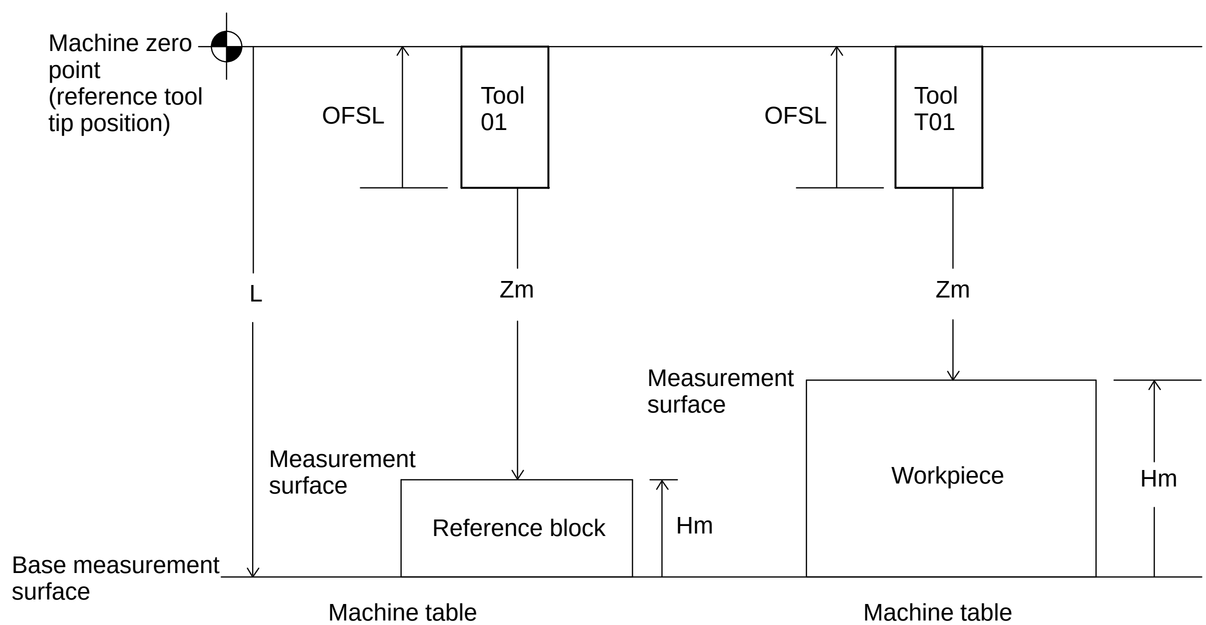
This parameter is valid when the WMA bit (bit 3 of parameter 5007) is set to 1.

5022	Distance from the reference tool tip position to the base measurement surface

[Data type] 2–word axis type

	Increment system		
	IS–A	**IS–B**	**IS–C**
Millimeter machine	0.01 mm	0.001 mm	0.0001 mm
Inch machine	0.001 inch	0.0001 inch	0.00001 inch

The distance L from the reference tool tip position to the base measurement surface with the machine placed at the machine zero point is set for each axis.



L : Distance from the reference tool tip position to the base measurement surface (machine coordinates of the base measurement surface)
Hm : Distance from the base measurement surface to the actual measurement surface
Zm : Distance from the tip of the tool to be measured to the measurement surface at the machine zero point
(Zt : Distance from the tip of the tool to be measured to the base measurement surface at the machine zero point)
OFSL : Tool length offset value (OFSL = Zm – Hm – L)

Reference item

Series 16*i*/160*i*/18*i*/180*i*	OPERATOR'S MANUAL (For Machining Center) (B–63014EN)	III.11.4.14	Tool length/workpiece origin measurement B

15 PMC CONTROL FUNCTION

15.1 PMC AXIS CONTROL

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.

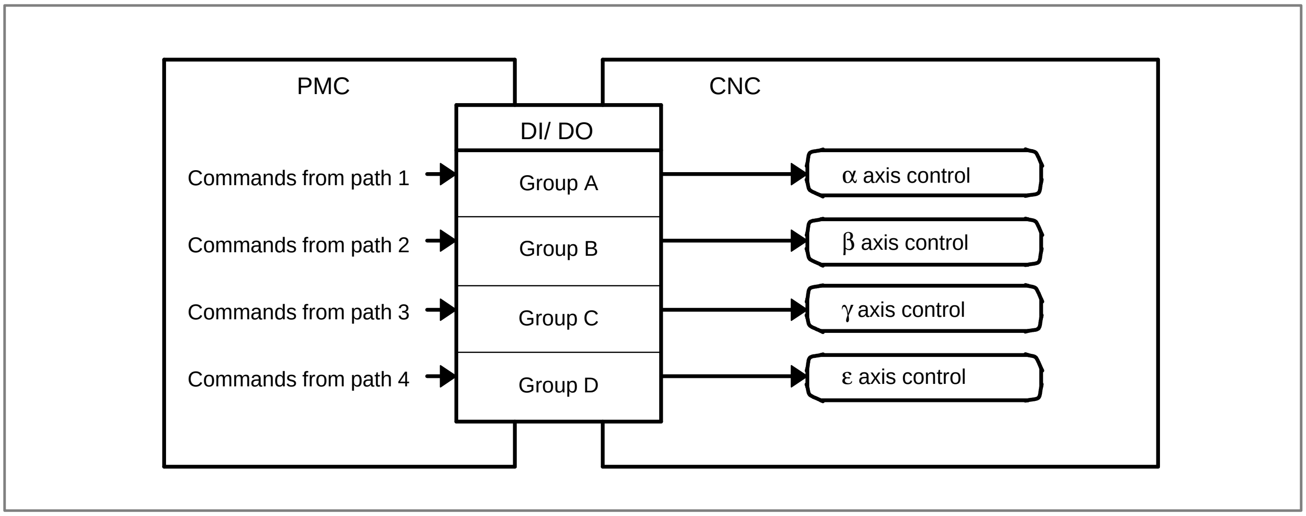
Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

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) Continuous feed
(7) Reference position return
(8) 1st reference position return
(9) 2nd reference position return
(10) 3rd reference position return
(11) 4th reference position return
(12) External pulse synchronization – Main spindle
(13) External pulse synchronization – first manual handle
(14) External pulse synchronization – second manual handle
(15) External pulse synchronization – third manual handle (for M series only)
(16) Feedrate control
(17) Auxiliary function, Miscellaneous function 2, Miscellaneous function 3
(18) Selection of the machine coordinate system
(19) Torque control

The PMC is provided with four paths to control these operations using input and output signals.

By issuing commands through these four paths, the PMC can simultaneously control multiple axes separately. Use parameter No. 8010 to determine which path controls which axis. Commands may be issued through one path to two or more axes, thus allowing the PMC to control multiple axes using one path.



In the following description, input/output signals from the four paths are called group A (path 1), group B (path 2), group C (path 3), and group D (path 4), respectively.

The name of an input/output signal used for PMC axis control always contains a lowercase “g”, as in EBUFg. However, there is no such signal as EBUFg. The actual signal names represented by EBUFg are EBUFA, EBUFB, EBUFC, and EBUFD, which respectively correspond to signals of group A (path 1), group B (path 2), group C (path 3), and group D (path 4).

Basic procedure

(1) In parameter No. 8010, specify which DI/DO signal group (A, B, C, or D) is to be used for PMC axis control on a per–axis basis.

When using the same group for simultaneously controlling two or more axes, check that the settings of the parameters related to feedrate (rapid traverse rate, acceleration/deceleration time constant, diameter/radius, linear axis/rotation axis, etc.) are identical for each axis to be controlled.

(2) To enable direct PMC axis control, set each control axis selection signal (EAX1 to EAX8), that corresponds to an axis to be controlled, to 1.

(3) Determine the operation.

The axis control command signals (EC0g to EC6g) specify the type of operation. The axis control feedrate signals (EIF0g to EIF15g) specify the feedrate. The axis control data signals (EID0g to EID31g) specify the moving distance and other data.

These signals, together with block stop prohibition signal EMSBKg (described later), determine one complete operation, which is tantamount to one block executed during CNC–controlled automatic operation. These signals may be collectively called the axis control block data signals.

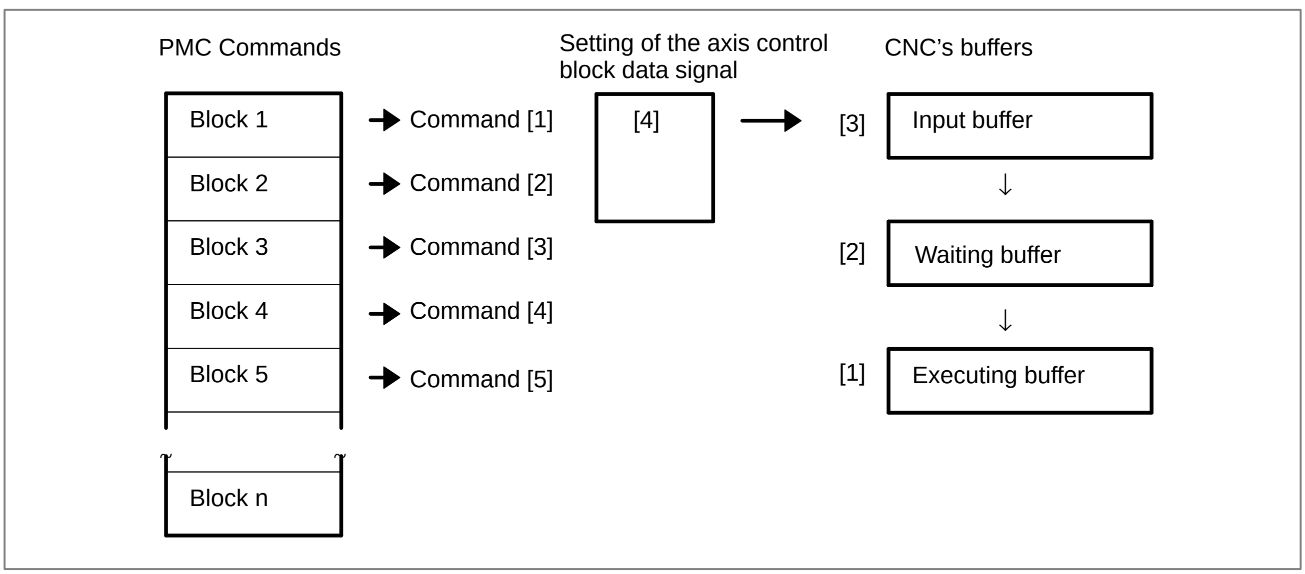
⊙ List of Signals Determining Data, Tantamount to One Block for PMC Axis Control

Generic name	Signal name	Symbol	Data type
Axis control block data signals	Block stop prohibition signal	EMSBKg	Bit type
	Axis control command signal	EC0g to EC6g	Byte type
	Axis control feedrate signal	EIF0g to EIF15g	Word type
	Axis control data signal	EID0g to EID31g	Two words type

(4) 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:

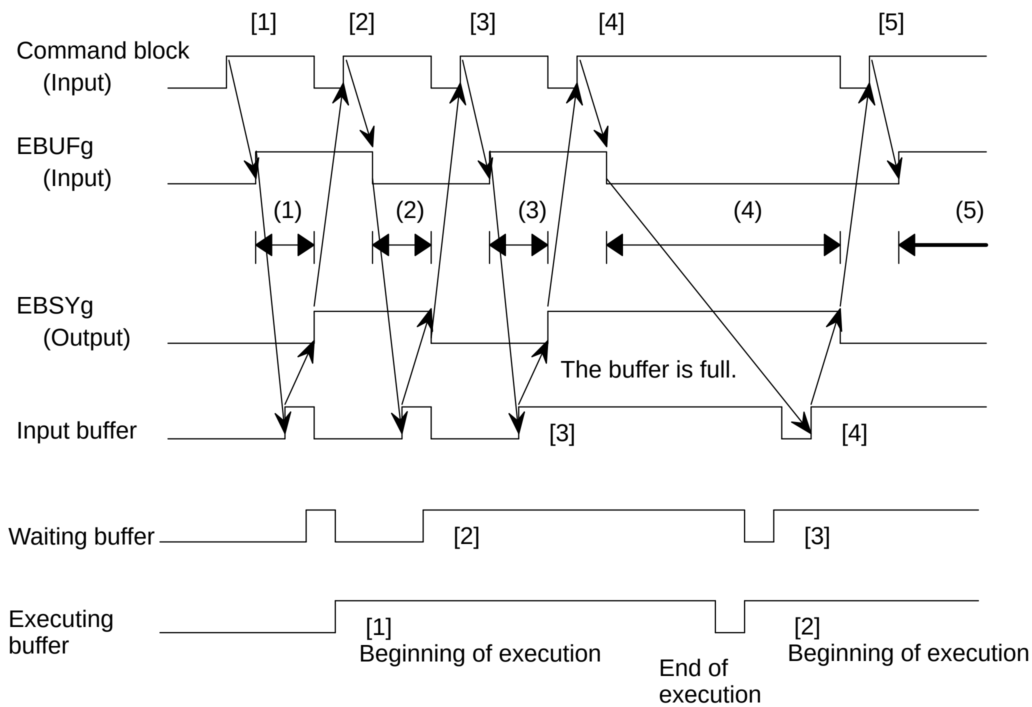
· command [2] is transferred from the waiting buffer to the executing buffer;

· 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.

(5) Repeat steps (3) and (4) until all the blocks have been issued.

When the final block has been issued, set control axis selection signals EAX1 to 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.

(6) When control axis selection signals EAX1 to EAX8 are set “1” 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 EM48g	Auxiliary function code signals
12	EMFg	Auxiliary function strobe signal
13	EMF2g	Miscellaneous function 2 strobe signal
14	EMF3g	Miscellaneous function 3 strobe signal
15	EFINg	Auxiliary function completion signal
16	ESOFg	Servo–off signal

No.	Symbol	Signal name
17	EMBUFg	Buffering disable signal
18	*EAXSL	Control axis selection status signal
19	EINPg	In–position signal
20	ECKZg	Following zero checking signal
21	EIALg	Alarm signal
22	EGENg	Axis moving signal
23	EDENg	Auxiliary function executing signal
24	EOTNg	Negative–direction overtravel signal
25	EOTPg	Positive–direction overtravel signal
26	*FV0E to *FV7E	Feedrate override signals
27	OVCE	Override cancellation signal
28	ROV1E, ROV2E	Rapid traverse override signals
29	DRNE	Dry run signal
30	RTE	Manual rapid traverse selection signal
31	EOV0	Override 0% signal
32	ESKIP	Skip signal
33	EADEN1 to EADEN8	Distribution completion signals
34	EABUFg	Buffer full signal
35	EACNT1 to EACNT8	Controlling signals
36	*+ED1 to *+ED8 *–ED1 to *–ED8	External deceleration signal
37	ELCKZg	Accumulated zero check signal
38	TRQMx	Torque control mode signal

Signal Detail

1 Control axis selection signals EAX1 to EAX8

[Classification] Input signal

[Function] When the signal is set to “1”, the corresponding axis becomes subject to PMC control.

When the signal is set to “0”, PMC control becomes invalid. Changing the setting of the control axis selection signal is possible only when control axis selection status signal *EAXSL is set to “0”. Changing the setting when *EAXSL is set to “1” results in the issue of a P/S alarm (No. 139). Alarm signal EIALg is set to “1”.

When NCC, bit 5 of parameter No. 8001, is set to “0”, a command issued from the CNC is executed while the control axis selection signal is set to “1” and signal *EAXSL is set to “0”. When the parameter is set to “1”, the same attempt results in the issue of a P/S alarm (No. 139). Note that the command is invalidated when the tool is moving along the axis in manual continuous feed mode.

If the control axis selection signal is set to “1” while the CNC is currently executing a command, a P/S alarm is generated. In manual continuous feed mode, setting this signal to “1” suspends the execution of the command. While *EAXSL is set to “0”, the status of alarm signal EIALg does not change to 1 when the control axis selection signal is set to 1 and a P/S alarm (No. 139) is generated. In this case, the axis can be controlled from the PMC, even when the CNC is in the alarm status.

NOTE
After setting control axis selection signals EAX1 to 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.

Axis control command (hexadecimal code)	Operation
02h	Cutting feed – feed per revolution (exponential acceleration/ deceleration or linear acceleration/deceleration after interpolation)
	Performs the same operation as G95 G01, used by the CNC.
03h	Skip – feed per minute
	Performs the same operation as G31 G01, used by the CNC.
04h	Dwell
	Performs the same operation as G04, used by the CNC.
05h	Reference position return
	Moves the tool in the direction of reference position return specified by ZMIx, bit 5 of parameter No. 1006, in rapid traverse mode, then performs the same operation as manual reference position return, done by the CNC.
06h	Continuous feed (exponential acceleration/deceleration)
	Moves the tool in the specified direction in jog feed mode. Performs the same operation as that of JOG feed, done by the CNC.
07h	1st reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G28 of the CNC.
08h	2nd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P2 of the CNC.
09h	3rd reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P3 of the CNC.
0Ah	4th reference position return
	Performs the same operation as done when positioning the tool to the reference position from the intermediate point specified by G30 P4 of the CNC.
0Bh	External pulse synchronization – main spindle
	Synchronizes with the main spindle.
0Dh	External pulse synchronization – 1st manual handle
	Synchronizes with the 1st manual handle.

Axis control command (hexadecimal code)	Operation
0Eh	External pulse synchronization – 2nd manual handle
	Synchronizes with the second manual handle.
0Fh	External pulse synchronization – 3rd manual handle
	Synchronizes with the 3rd manual handle.
10h	Speed command (linear acceleration/deceleration)
	Performs continuous feed at the specified speed.
11h	Torque control
	Continuous feed under torque control
12h	Auxiliary function
	Performs the same function as the miscellaneous function (M function), used by the CNC.
14h	Miscellaneous function 2
	Similar to the miscellaneous function of the CNC
15h	Miscellaneous function 3
	Similar to the miscellaneous function of the CNC
20h	Machine coordinate system selection
	Performs the same operation as G53, used by the CNC.

Rapid traverse rate

When using the rapid traverse command (EC0g to EC6g: 00h), the feedrate can be specified in either the same parameter as that used by the CNC (No. 1420) or the PMC's axis interface feedrate signals EIF0g to EIF15g. This can be set with RPD, bit 0 of parameter No. 8002.

Feed per revolution

When using the cutting feed – feed per revolution command (EC0g to EC6g: 02h), the optional function for threading in synchronous feed mode is necessary in the case of the M series.

This operation cannot be performed when IT0 to IT2, bit 4 to 6 of parameter No. 7501, specify high–speed cycle machining.

Reference without dogs position return

The reference position return command (EC0g to EC6g: 05h) enables the following operation: When DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs for each axis, is valid and the tool has not been returned to the reference position since the power was turned on, move each axis in the direction specified by the continuous feed command (EC0g to EC6g: 06h) (position the tool to a point near the reference position) and issue the reference position return command (EC0g to EC6g: 05h). This returns the tool to the reference position (positions the tool to the grid nearest the current position) without using the deceleration signal for reference position return.

Note that, when positioning the tool to a point near the reference position, the tool must be moved in the direction of reference position return at such a speed that the servo position error exceeds the value of parameter No. 1836.

The direction of the grid relative to the proximate position depends on ZMIx, bit 5 of parameter No. 1006.

After the reference position has been established, reference position return can be performed at high speed by issuing the reference position return command (EC0g to EC6g: 05h), irrespective of the reference position return direction specified by ZMIx, bit 5 of parameter No. 1006.

First reference position return for an incomplete reference position return without dogs

When using the 1st reference position return command (EC0g to EC6g: 07h), if DLZ, bit 1 of parameter No. 1002, specifying reference position return without dogs for all axes, or DLZx, bit 1 of parameter No. 1005, specifying reference position return without dogs on a per–axis basis is valid and the tool has not been returned to the reference position since the power was turned on, issuing the 1st reference position return command (EC0g to EC6g: 07h) results in the issue of a P/S alarm (No. 090).

1st to 4th reference position return

When using the 1st to 4th reference position return commands (EC0g to EC6g: 07h to 0Ah), the feedrate can be specified using RPD, bit 0 of parameter No. 8002, in the same manner as when using the rapid traverse command (EC0g to EC6g: 00h).

Note that, in the case of the 1st reference position return, if the tool has not been manually returned to the reference position after the power was turned on, the feedrate specified by parameter No. 1424 applies.

External pulse synchronization

When using the external pulse synchronization commands (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the tool moves backwards if the external pulse has a negative value. When a manual handle interrupt is executed for the axis to which the external pulse is being applied, the moving distance is the sum of the external pulse and the interrupt pulse.

When a serial spindle is used, synchronization with the main spindle (EC0g to EC6g: 0Bh) is not possible.

Display of remaining distance

When using the continuous feed command (EC0g to EC6g: 06h) and the external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh), the displayed remaining distance is always “0”.

Speed command

When using the speed command (EC0g to EC6g: 10h), specify the axis to be controlled as a rotation axis in ROTX, bit 0 of parameter No. 1006.

While position control is being executed for the continuous feed command (EC0g to EC6g: 06h), the speed command (EC0g to EC6g: 10h) exerts speed control over the servo motor, thus allowing the speed to be dynamically changed during continuous feed. This makes this command suitable for driving a rotation tool with a servo motor.

A linear acceleration/deceleration time constant can be set for each axis, using parameter No. 8028.

Note that, while jog feed is being executed by the speed command, no coordinate system values are changed. This will result in the loss of the tool position. Therefore, after continuous feed has been completed, always return the tool to the reference position before executing the move command.

Machine coordinate system selection

The machine coordinate system selection command (EC0g to EC6g: 20h) performs absolute positioning to move the tool in rapid traverse to a specified position on the machine coordinate system. This command is used to move the tool to a position specifically defined for the machine, such as the tool exchange position.

This command can be used, irrespective of whether the optional function for workpiece coordinate system setting is provided.

For a rotation axis, short cut rotation can also be specified. When using this command for the T series, cancel the tool offset and the tool nose radius compensation. For the M series, cancel cutter compensation, tool length compensation, and tool offset.

The machine coordinate system must be set before attempting to use this command. After turning on the power, return the tool to the reference position either manually or by using G28. When an absolute position detector is provided, returning the tool to the reference position is not necessary because the tool position will be stored in memory.

Torque control command

When torque control is selected (EC0g to EC6g: 11h), the PMC controlled axis can be subjected to torque control instead of position control. In the torque control mode, the servo motor outputs the torque specified by the NC.

(1) Switching from position control to torque control

a. Setting a torque control axis

Set the axis to be subjected to torque control in the TRQMx bit (bit 7 of parameter No. 2007). Set the torque control axis also in parameter No. 2105 of torque constant. The motor–specific standard value is automatically set in this parameter when the power is turned on with the DGPR bit (bit 1 of parameter No. 2000) set to 0.

b. Position management in torque control mode

In torque control mode, whether to perform follow–up can be selected by setting the TQF bit (bit 4 of parameter No. 1803).

When follow–up is not performed, an integrated travel value (error count) exceeding the value of parameter No. 1885 causes servo alarm 423 to be issued. When torque control is switched to position control, follow–up is always performed, even if follow–up suppression is selected.

The CNC manages the position even in torque control mode. After torque control is switched to position control, no reference position return is necessary.

c. Traveling direction and speed in torque control mode

In torque control mode, the torque specified by the axis control data signal is output. While there is no target of torque generation or while the output torque falls below the specified torque, the traveling direction is determined by the plus or minus sign added to the torque data, as in position control. When the traveling speed exceeds the specified speed, servo alarm 422 is issued.

d. Timing of switching to torque control

A mode is switched to torque control mode after the position error enters the effective range (in–position state).

(2) Switching from torque control to position control (canceling torque control mode)

Torque control mode is canceled when any of the following conditions is satisfied:

1) The reset signal ECLRg is brought to “1”.

2) A servo alarm is issued.

3) An OT alarm is issued on the torque control axis.

4) An emergency stop occurs.

5) The servo motor is turned off by the servo off signal ESOFg.

a. Timing for canceling torque control

When torque control by the PMC controlled axis is canceled, follow–up may or may not be performed, depending on the setting in the TRF bit (bit 4 of parameter No. 1803). If follow–up is suppressed, the torque control mode signal TRQMx immediately goes “0” when a cause to cancel torque control is detected. Then, position control mode is selected. The execution terminates when the position error enters the effective range. If follow–up is performed, the torque control mode signal TRQMx goes “0” when a cause to cancel torque control is detected. Then, follow–up starts. When the position error falls below or equals to the cancel limit value specified in parameter 1886, position control mode is selected. The execution terminates when the position error enters the effective range.

b. Command after torque control mode is canceled

When torque control mode is canceled, normal position control mode is selected. In torque control mode, the position is managed, and the machine coordinates are not displaced. The workpiece coordinates, however, are shifted from the machine coordinates. The shift must be canceled by specifying the command for setting the workpiece coordinate system or the like.

CAUTION

1 If the torque control axis may be moved in torque control mode, the follow–up parameter TRF (bit 4 of parameter No. 1803) must be set to “1”.
2 If torque control mode is canceled while the torque control axis is moving, the return to position control mode causes a mechanical impact. A faster movement causes a greater impact. To cancel torque control mode, decelerate or stop the movement in advance.
3 When specifying the torque control command after a manual reference position return is completed, set the feed direction selection signal to “0” in advance. Alternatively, select a mode other than manual reference position return mode in advance.
4 In torque control mode, never detach a controlled axis by the controlled axis detach signal DTCHx or by a setting parameter.

The following table shows the correspondence between the axis control commands and their data:

<table>
<tr><th colspan="3">Command block</th></tr>
<tr><th>Operation</th><th>Axis control code signal EC0g to EC6g</th><th>Command data</th></tr>
<tr><td>Rapid traverse</td><td>00h</td><td>Total moving distance
EID0g to EID31g
Rapid traverse rate
EIF0g to EIF15g

The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.</td></tr>
<tr><td>Cutting feed – feed per minute</td><td>01h</td><td rowspan="2">Total moving distance
EID0g to EID31g
Feedrate
EIF0g to EIF15g</td></tr>
<tr><td>Skip – feed per minute</td><td>03h</td></tr>
<tr><td>Cutting feed – feed per revolution</td><td>02h</td><td>Total moving distance
EID0g to EID31g
Feed per rotation
EIF0g to EIF15g</td></tr>
<tr><td>Dwell</td><td>04h</td><td>Dwell time
EID0 to EID31g</td></tr>
<tr><td>Reference position return</td><td>05h</td><td>None</td></tr>
<tr><td>Jog feed</td><td>06h</td><td>Feed direction EID31g
Jog feedrate
EIF0g to EIF15g</td></tr>
</table>

Operation	Axis control code signal EC0g to EC6g	Command data
1st reference position return	07h	Rapid traverse rate EIF0g to EIF15g The rapid traverse rate is valid when PRD, bit 0 of parameter No. 8002, is set to “1”.
2nd reference position return	08h	
3rd reference position return	09h	
4th reference position return	0Ah	
External pulse synchronization – main spindle	0Bh	Pulse weight EIF0g to EIF15g
External pulse synchronization – manual handle	0Dh	
	0Eh	
	0Fh <For M series only>	
Speed command	10h	Continuous feed EIF0g to EIF15g
Torque control	11h	Maximum feedrate, EIF0g to EIF15g Torque data, EID0g to EID31g
Auxiliary function	12h	Auxiliary function code EID0g to EID15g
Miscellaneous function 2	14h	Miscellaneous function code, EID0g to EID15g
Miscellaneous function 3	15h	Miscellaneous function code, EID0g to EID15g
Machine coordinate system setting	20h	Machine coordinate system setting (absolute value) EID0g to EID31g Rapid traverse rate EIF0g to EIF15g The rapid traverse rate setting is effective when bit 0 (RPD) of parameter No. 8002 is set to “1”.

3 Axis control feedrate signals EIF0g to EIF15g

[Classification] Input signal

[Function] (1) Rapid traverse (EC0g to EC6g: 00h)

(2) 1st reference position return (EC0g to EC6g: 07h)

(3) 2nd reference position return (EC0g to EC6g: 08h)

(4) 3rd reference position return (EC0g to EC6g: 09h)

(5) 4th reference position return (EC0g to EC6g: 0Ah)

(6) Machine coordinate system selection (EC0g to EC6g: 20h)

For these commands, signals EIF0g to EIF15g are used to specify the rapid traverse rate, in binary format, when bit 0 (RPD) of parameter No. 8002 is set to "1". For 1st reference position return, however, the rapid traverse rate specified with parameter No. 1424 is used if manual reference position return has not been performed after the power was first turned on.

[Unit of data]

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1		mm/min
	Inch machine	0.1		inch/min
Rotation axis		1		deg/min

[Valid data range] Specify data within the range given in the following table.

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	30 to 15000	30 to 12000	mm/min
	Inch machine	30 to 6000	30 to 4800	inch/min
Rotation axis		30 to 15000	30 to 12000	deg/min

(7) Cutting feed – feed per minute (EC0g to EC6g: 01h)

(8) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, the signals are used to specify, in binary format, the feedrate along an axis. The specified feedrate can be magnified by ten by the setting of bit 3 (F10) of parameter No. 8002.

[Unit of data] When bit 3 (F10) of parameter No. 8002 is set to 0

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1	0.1	mm/min
	Inch machine	0.01	0.01	inch/min
Rotation axis		1	0.1	deg/min

When bit 3 (F10) of parameter No. 8002 is set to 1

		Data unit		Unit
		IS–B	IS–C	
Linear axis	Metric machine	10	1	mm/min
	Inch machine	0.1	0.01	inch/min
Rotation axis		10	1	deg/min

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric machine	1 to 100000	0.1 to 12000.0	mm/min
	Inch machine	0.01 to 4000.00	0.001 to 480.000	inch/min
Rotation axis		1 to 100000	0.1 to 12000.0	deg/min

CAUTION
When "0" is specified, the CNC continues to perform buffering without moving the tool. In such a case, release the buffering by issuing reset signal ECLRg.
Cutting feedrate clamp is disabled.

(9) Cutting feed – feed per rotation (EC0g to EC6g: 02h)

For this command, the signals are used to specify the amount by which the tool is moved for every rotation of the spindle.

<For T series>

[Unit of data] The data increment depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.0001	0.000001	0.0001
0	0			
0	1	0.001	0.00001	0.001
1	0	0.01	0.0001	0.01

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.0001 to 500.0000		mm/rev
	Inch input	0.000001 to 9.999999		inch/rev
Rotation axis		0.0001 to 500.0000		deg/rev

<For M series>

[Unit of data] The data unit depends on the settings of bits 6 (FR1) and 7 (FR2) of parameter No. 8002, as listed in the following table.

Parameter		Metric input (mm/rev)	Inch input (inch/rev)	Rotation axis (deg/rev)
FR2	FR1			
1	1	0.01	0.0001	0.01
0	0			
0	1	0.1	0.001	0.1
1	0	1	0.01	1

[Valid data range] 1 to 65535
(Actual values must fall within the ranges given in the following table.)

		Data range		Unit
		IS–B	IS–C	
Linear axis	Metric input	0.01 to 500.00		mm/rev
	Inch input	0.0001 to 9.9999		inch/rev
Rotation axis		0.01 to 500.00		deg/rev

WARNING
1 The value of parameter No. 8022 is used as the upper limit for clamping the feedrate.
2 Override for the feedrate is effective. Dry run is invalid.

CAUTION
The specified feedrate can be magnified by 1, 10, or 100 by setting bits 6 (FR1) and 7 (FR2) of parameter No. 8002 accordingly.

(10) External pulse synchronization – main spindle
(EC0g to EC6g: 0Bh)

(11) External pulse synchronization – first manual handle
(EC0g to EC6g: 0Dh)

(12) External pulse synchronization – second manual handle
(EC0g to EC6g: 0Eh)

(13) External pulse synchronization – third manual handle
(EC0g to EC6g: 0Fh)

For these commands, the signals are used to specify the weight of the external pulses. A weight range of ±1/256 to ±127 can be set by using signals EIF0g to EIF7g for the figures after the decimal point. When a negative weight is specified, the tool is moved in the reverse direction. When a new pulse weight is specified while the tool is moving in synchronization with external pulses, inverting signal EBUFg causes the tool to move with the new pulse weight.

As commands for (10) to (13) are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

CAUTION
The pulse weight is clamped according to the value set for parameter No. 1424 (parameter for the manual rapid traverse rate for each axis).

NOTE
When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

(14) Continuous feed (EC0g to EC6g: 06h)

Set the feedrate as the same as for cutting feed – feed per minute (EC0g to EC6g: 01h). The feedrate can be changed during continuous feed.

Specify the feedrate with signals EIF0g to EIF15g, and invert the axis control command read signal EBUFg during continuous feed, then the tool moves at the new feedrate.

As commands for jog feed are executed without buffering, axis control command read completion signal EBSYg usually need not be checked.

The specified feedrate can be magnified by 10 by setting bit 3 (F10) of parameter No. 8002, and by 200 by setting bit 2 (JFM) of parameter No. 8004.

- Maximum feedrate (with override of 254%)

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magnified by 1	166458mm/min	1664.58inch/min	16645mm/min	166.45inch/min
Magnified by 10	1664589mm/min	16645.89 inch/min	166458mm/min	1664.58inch/min
Magnified by 200 (Note 1)	1966050mm/min	1966605.00 inch/min	196605mm/min	19660.50 inch/min

CAUTION
The maximum feedrate depends on whether override is applied or canceled. The following table lists the maximum feedrate when override is canceled.

NOTE
1 Magnification of 200 is valid only for the continuous feed command (EC0g to EC6g: 06h).
2 The actual speed may not be displayed correctly, depending on the feedrate.

	IS–B		IS–C	
	Metric input	**Inch input**	**Metric input**	**Inch input**
Magni-fied by 1	65535mm/min	655.35inch/min	6553mm/min	65.53inch/min
Magni-fied by 10	655350mm/min	6553.50inch/min	65535mm/min	655.35inch/min
Magni-fied by 200	13107000 mm/min	131070.00 inch/min	1310700mm/min	13107.00 inch/min

(15) Speed command (EC0g to EC6g: 10h)

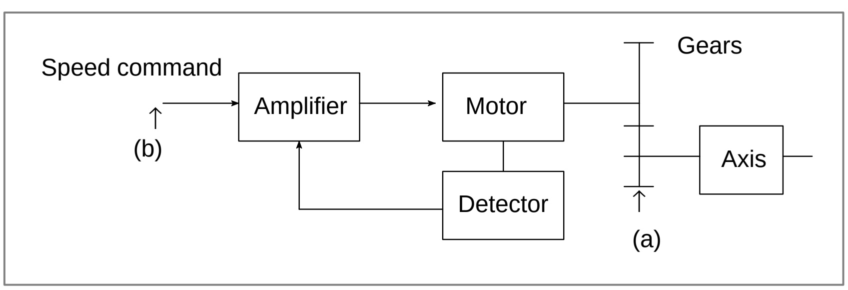
For this command, the signals are used to specify, in binary format, the servo motor speed.

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
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:

$$\mathrm{Fmin} = \frac{P \times 2}{15} \times \frac{1}{1000}$$

Fmin : Minimum increment for the motor speed
P : Number of pulses per rotation of the detector for velocity feedback

Specify the speed command using the value calculated by the following formula:

$$F = \frac{N \times P \times 2}{15} \times \frac{1}{1000}$$

F : Speed command value (integer)
N : Servo motor speed (rpm)
P : Number of detector pulses issued per rotation for velocity feedback

> **NOTE**
> In speed command mode, the speed after acceleration/deceleration is specified to the servo control unit. The loop gain for position control is invalid.

(16) When a torque control command (EC0g to EC6g: 11h) is specified

Specify the maximum speed in torque control mode, in rpm.

A servo alarm (No. 422) is issued if there is no object for which a torque is to be generated or if the feedrate exceeds the specified value while torque control is being applied.

The maximum feedrate in torque control mode can be changed in torque control mode by setting the new feedrate data in the signal then inverting the logic of axis control command read signal EBUFg.

Valid data range	Units
1 to 32767	rpm

> **NOTE**
> The data units will be cm/min when a linear motor is being used.

4 Axis control data signals EID0g to EID31g

[Classification] Input signal

[Function]

[Unit of data]

	IS–B	IS–C	Unit
Metric input Degree input	0.001	0.0001	mm deg
Inch input	0.0001	0.00001	inch

[Valid data range]

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) Cutting feed – feed per minute (EC0g to EC6g: 01h)

(3) Cutting feed – feed per rotation (EC0g to EC6g: 02h)

(4) Skip – feed per minute (EC0g to EC6g: 03h)

For these commands, signals EID0g to EID31g are used to specify, in binary format, the incremental moving distance, according to the input increment used for the axis.

	IS–B	IS–C	Unit
Metric input Degree input	±99999.999	±9999.9999	mm deg
Inch input	±9999.9999	±999.99999	inch

When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

(5) Dwell (EC0g to EC6g: 04h)

For this command, the signals are used to specify, in binary format, the dwell time.

Data range	Unit
1 to 9999999	ms

When diameter programming is specified with bit 3 (DIAx) of parameter No. 1006, bit 1 (CDI) of parameter No. 8005 can be used to specify whether a radius or diameter is to be used in a command.

> **NOTE**
> When the increment system IS–C is used, the least input increment for the dwell time can be set to 0.1 ms, according to the setting of bit 1 (DWE) of parameter No. 8002.

(6) Continuous feed (EC0g to EC6g: 06h)

For this command, signal EID31g is used to specify the direction of continuous feed, as follows:

0: Positive direction
1: Negative direction

Signals EID0g to EID30g are undefined.

(7) Auxiliary functions (EC0g to EC6g: 12h)

For this command, the signals are used to specify, in binary format, an auxiliary function code to be sent to the PMC. The auxiliary function code can be specified using either one or two bytes, depending on the setting of bit 6 (AUX) of parameter No. 8001, in signals EID0g to EID15g.

(8) Machine coordinate system selection (EC0g to EC6g: 20h)

For this command, the signals are used to specify, in binary format, an absolute machine coordinate, according to the increment system used by the axis.

Example: For absolute value “10000”

Input increment	inch	1.0000	mm	10.000
Output increment	mm	25.400	inch	0.3937

The direction of rotation can be specified about a rotation axis with a parameter. To enable the roll–over function, set bit 0 (ROAx) of parameter No. 1008 to 1. Then, select whether the tool is to be rotated in the direction corresponding to the sign of the specified value, or in whichever direction minimizes the distance to the end point, using bit 1 (RABx) of parameter No. 1008. The moving distance per rotation must be set in parameter No. 1260.

(9) When a torque control command (EC0g to EC6g: 11h) is specified

Specify the torque data.

Specify a positive value when the torque is to be applied in the positive direction. Specify a negative value when the torque is to be applied in the negative direction.

The specified torque data can be changed in torque control mode by setting the new data in the signal then inverting the logic of axis control command read signal EBUFg.

Valid data range	Units
–7282 to +7282	0.0001 Nm

NOTE

The data units will be 0.001 N when a linear motor is being used.

5 Axis control command read signal EBUFg

[Classification] Input signal

[Function] Directs the CNC to read a block of command data for PMC axis control. See “Basic procedure” for details of the operation performed when this signal is set from “0” to “1” or from “1” to “0”.

6 Axis control command read completion signal EBSYg

[Classification] Output signal

[Function] Notifies the system that the CNC has read a block of command data for PMC axis control and has stored the block in the input buffer. See “Basic procedure” for details of the output conditions and the procedure.

7 Reset signal ECLRg

[Classification] Input signal

[Function] Resets the corresponding PMC–controlled axis.
When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation.

Simultaneously, all buffered commands are canceled. Any control command is ignored while this signal is set to “1”.

The continuous feed command (EC0g to EC6g: 06h) and external pulse synchronization command (EC0g to EC6g: 0Bh, 0Dh to 0Fh) can be terminated by setting reset signal ECLRg to “1”. When these commands are terminated, the servo motor decelerates and stops, the axis moving signal EGENg is set to “0”, and the control axis selection status signal *EAXSL is set to “0”. Confirm that the control axis selection status signal *EAXSL has been set to “0” before issuing the next command. Do not set reset signal ECLRg to “0” until the control axis selection status signal *EAXSL has been set to “0”.

The speed command (EC0g to EC6g: 10h) can also be terminated by setting the reset signal ECLRg to “1”. When this command is terminated, the servo motor decelerates and stops, and the axis moving signal EGENg is set to “0”. Confirm that the axis moving signal EGENg has been set to “0” before issuing the next command. Do not attempt to set the reset signal ECLRg to “0” until the axis moving signal EGENg has been set to “0”.

8 Axis control temporary stop signal ESTPg

[Classification] Input signal

[Function] When this signal is set to “1”, the following is performed:

(1) When the tool is moving along the axis: Decelerates and stops the tool.

(2) When the tool is dwelling: Stops the operation.

(3) When an auxiliary function is being executed: Stops the operation when auxiliary function completion signal EFINg is input.

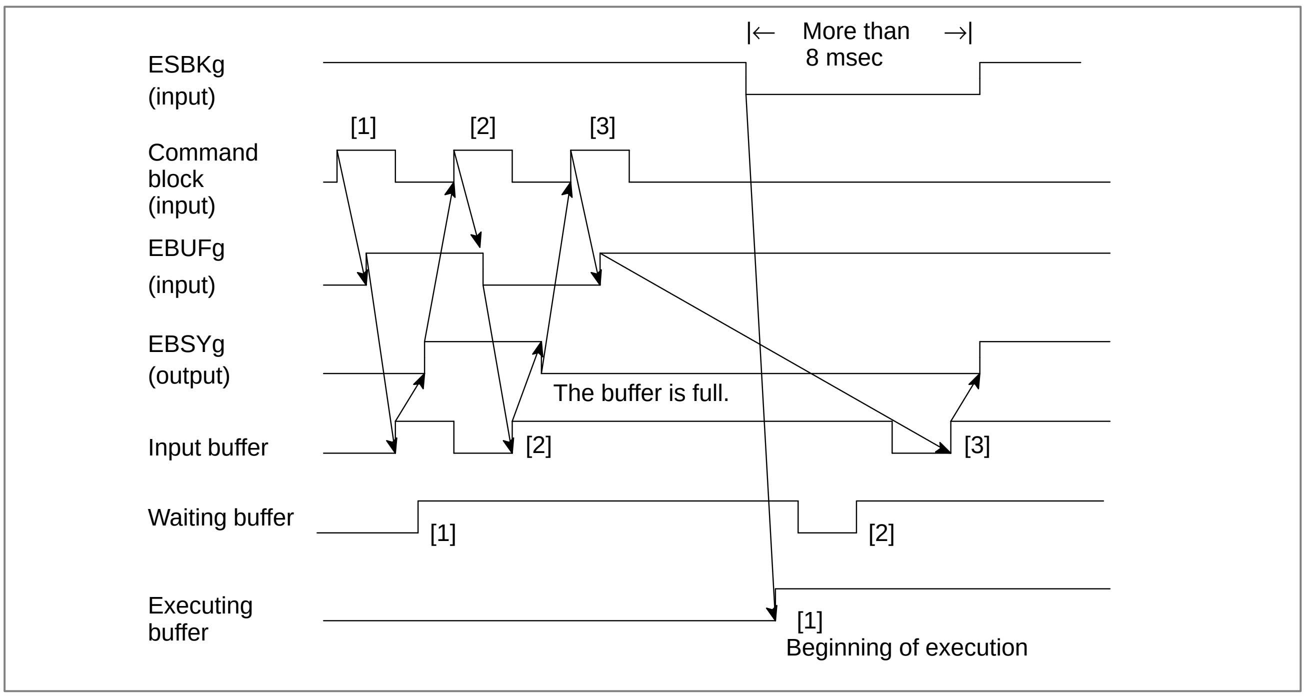
The stopped operation can be restarted by setting this signal to “0”.

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 EM48g

[Classification] Output signal

12 Auxiliary function strobe signal EMFg

[Classification] Output signal

13 Miscellaneous function 2 strobe signal EMF2g

[Classification] Output signal

14 Miscellaneous function 3 strobe signal EMF3g

[Classification] Output signal

15 Auxiliary function completion signal EFINg

[Classification] Input signal

[Function] When an auxiliary function command (EC0g to EC6g: 12h) is issued by the PMC, the auxiliary function code is specified in a byte (using signals EID0g to EID7g) or two bytes (using signals EID0g to EID15g), depending on the setting of bit 6 (AUX) of parameter No. 8001.

The CNC sends the auxiliary function code specified in signals EID0g to EID7g and EID8g to EID15g to auxiliary function code signals EM11g to EM28g and EM31g to EM48g and awaits auxiliary function completion signal EFINg. When the auxiliary function completion signal EFINg is returned, the CNC starts executing the next block.
The timings for sending the auxiliary function code signals and auxiliary function strobe signal, as well as for receiving the auxiliary function completion signal, are the same as those for the miscellaneous functions (M functions) under the control of the CNC. See "Auxiliary function executing signal" for details.

---

16 Servo–off signal ESOFg

[Classification] Input signal

[Function] When this signal is set to "1", the servo motor for the corresponding PMC–controlled axis is turned off (servo–off state).

When this signal is set to "0", the servo motor is turned on.

When a torque control command (EC0g to EC6g: 11h) is specified, entering the servo–off state cancels torque control mode, but the torque control state remains set. In such a case, set reset signal ECLRg to "1".

---

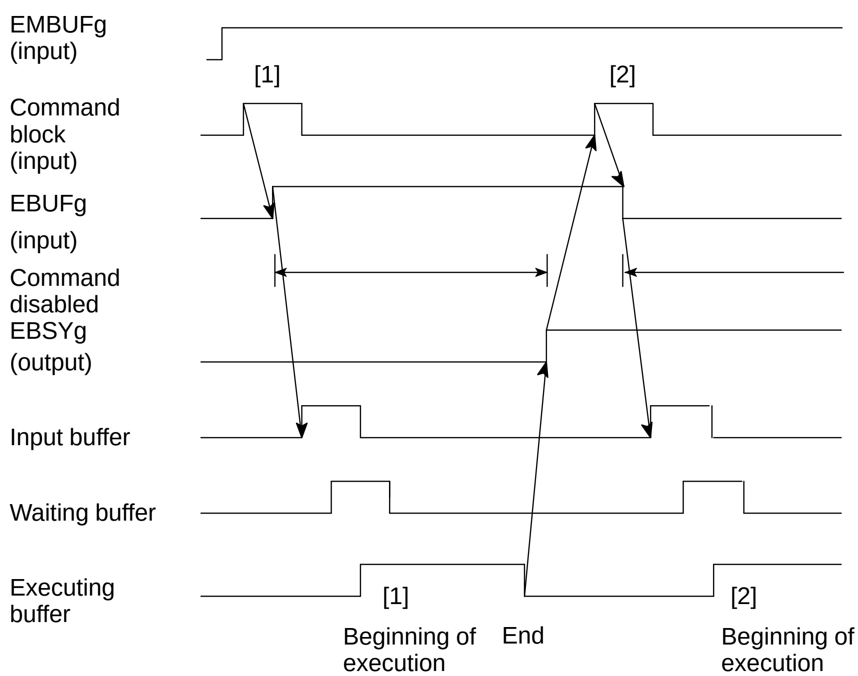
17 Buffering disable signal EMBUFg

[Classification] Input signal

[Function] When this signal is set to "1", commands from the PMC are not read while the executing, waiting, or input buffer contains a block. If this signal is set to "1" when any of these buffers contain a block, that block is executed but subsequent commands are read only when the buffers are all empty.

To discriminate the buffering disabled condition, the CNC outputs the axis control command read completion signal (EBSYg) only when a command is read when all buffers are empty.

The timing chart for the command operation is shown below.



Buffering is disabled, regardless of buffering disable signal EMBUFg, for the following commands:

(1) Skip–feed per minute (EC0g to EC6g: 03h)
(2) Reference position return (EC0g to EC6g: 05h)
(3) 1st reference position return (EC0g to EC6g: 07h)
(4) 2nd reference position return (EC0g to EC6g: 08h)
(5) 3rd reference position return (EC0g to EC6g: 09h)
(6) 4th reference position return (EC0g to EC6g: 0Ah)
(7) Machine coordinate system selection (EC0g to EC6g: 20h)

The following commands, for which reset signal ECLRg is used for termination, operate as though buffering had been disabled. That is, the subsequently specified block is not executed but canceled:

(1) Continuous feed (EC0g to EC6g: 06h)
(2) External pulse synchronization – main spindle (EC0g to EC6g: 0Bh)
(3) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)
(4) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(5) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

(6) Speed command (EC0g to EC6g: 10h)

18 Control axis selection status signal *EAXSL

[Classification] Output signal

[Function] When this signal is set to “0”, control axis selection signals EAX1 to 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 EAX8 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 EAX8 or set servo–off signal ESOFg to “0” before setting reset signal ECLRg to “1”.

When a command is issued for any of the four paths with PMC axis control, signal *EAXSL is set to “1” to disable axis selection. Thus, changing signals EAX1 to EAX8 results in the output of P/S alarm No. 139. For paths for which commands are not issued, however, axis selection is enabled if bit 5 (DSL) of parameter No. 8004 is set accordingly.

19 In–position signal EINPg

[Classification] Output signal

[Function] This signal is set to “1” when the corresponding PMC–controlled axis is in the in–position state.

When the tool is decelerated, in–position check is performed to disable the next command until the tool enters the in–position area. The in–position check, however, can be skipped using bit 6 (NCI) of parameter No. 8004 to reduce the cycle time.

> **NOTE**
> When the axis fed at a very low speed, in –position signal might turn “1”.

20 Following zero checking signal ECKZg

[Classification] Output signal

[Function] This signal is set to “1” when following zero check or in–position check is being performed for the corresponding PMC–controlled axis.

21 Alarm signal EIALg

[Classification] Output signal

[Function] This signal is set to “1” when a servo alarm, overtravel alarm, or P/S alarm No. 130 or 139 occurs for the corresponding PMC–controlled axis. This signal is set to “0” when reset signal ECLRg is set to “1” after the alarm is released, as described below.

- Servo alarm

  Eliminate the cause of the alarm, then reset the CNC.

- Overtravel alarm

  Move the tool into the area within the stored stroke limit, then reset the CNC.

  The following commands can be used to move the tool into the area within the stored stroke limit during an overtravel alarm:

(1) Rapid traverse (EC0g to EC6g: 00h)

(2) Cutting feed–feed per minute (EC0g to EC6g: 01h)

(3) Cutting feed–feed per rotation (EC0g to EC6g: 02h)

(4) Continuous feed (EC0g to EC6g: 06h)

(5) External pulse synchronization – first manual handle (EC0g to EC6g: 0Dh)

(6) External pulse synchronization – second manual handle (EC0g to EC6g: 0Eh)

(7) External pulse synchronization – third manual handle (EC0g to EC6g: 0Fh)

- P/S alarm (130 or 139)

  Reset the CNC. See “Alarms and messages” for details.

Reset signal ECLRg cannot be used to reset the CNC in the above cases. Use the reset button on the setting panel, external reset signal ERS, or emergency stop signal *ESP.

22 Axis moving signal EGENg

[Classification] Output signal

[Function] This signal is set to “1” when the tool is moving along the corresponding PMC–controlled axis according to commands such as rapid traverse (EC0g to EC6g: 00h) and cutting feed (EC0g to EC6g: 01h).

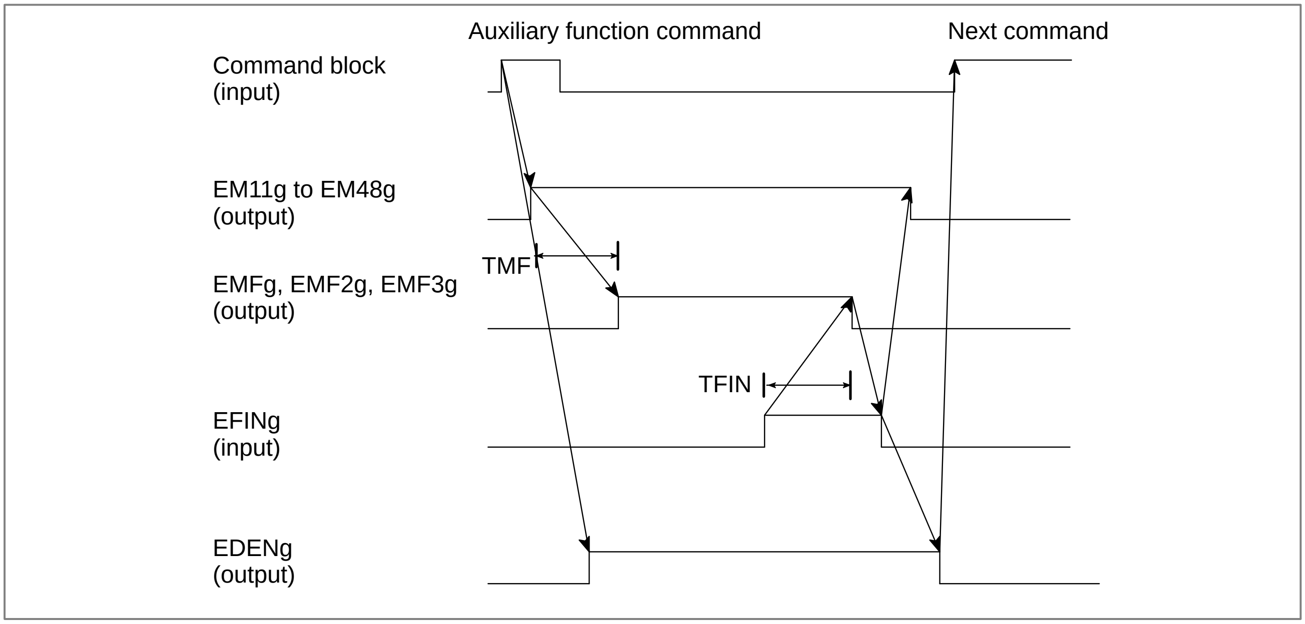
NOTE
This signal is set to "0" when distribution for the axis is completed (the signal is set to "0" during deceleration).

23 Auxiliary function executing signal EDENg

[Classification] Output signal

[Function] When an auxiliary function (EC0g to EC6g: 12h) is specified by the PMC, this signal is set to "1" during the period from when auxiliary function codes EID0g to EID15g are sent to auxiliary function code signals EM11g to EM48g until auxiliary function completion signal EFINg is returned.

The timing chart for the command operation is shown below.



TMF and TFIN are set with parameters No. 3010 and 3011.

24 Negative–direction overtravel signal EOTNg
25 Positive–direction overtravel signal EOTPg

[Classification] Output signal

[Function] These signals are set to "1" when an overtravel alarm is detected. When the stroke limit in the negative direction is exceeded, signal EOTNg is set to "1". When the stroke limit in the positive direction is exceeded, signal EOTPg is set to "1". Simultaneously, alarm signal EIALg is set to "1".

These signals are set to “0” when the overtravel alarm is released and reset signal ECLRg is set to “1”. See “Alarm signal EIALg” for details of how to release an overtravel alarm.

26 Feedrate override signals *FV0E to *FV7E

[Classification] Input signal

[Function] Like the CNC’s feedrate override signals *FV0 to *FV7, these signals can be used to select the override for the cutting feedrate, in steps of 1% from 0 to 254%, independently of the CNC using bit 2 (OVE) of parameter No. 8001.

These signals form an eight–bit binary code and correspond to the override value as follows:

$$\text{Override value} = \sum_{i=0}^{7} |2^i \times Vi|\%$$

Vi = 0 when signal *FViE is 1
Vi = 1 when signal *FViE is 0

That is, each signal has the following significance:

*FV7E = 128%, *FV3E = 8% ,
*FV6E = 64%, *FV2E = 4% ,
*FV5E = 32%, *FV1E = 2% ,
*FV4E = 16%, *FV0E = 1%

When all signals are set to “0”, the override is regarded as being 0%, as well as when all signals are “1”.

27 Override cancellation signal OVCE

[Classification] Input signal

[Function] When override is enabled, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001, setting this signal to “1” fixes the cutting feed override to 100%. This signal does not affect the rapid traverse override.

28 Rapid traverse override signals ROV1E and ROV2E

[Classification] Input signal

[Function] These signals can be used to select the override for the rapid traverse rate, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001.

Rapid traverse override signals		Override value
ROV2E	ROV1E	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the minimum feedrate specified with parameter No. 1421.

29 Dry run signal DRNE
30 Manual rapid traverse selection signal RTE

[Classification] Input signal

[Function] These signals can be used to perform dry run or manual rapid traverse, independently of the CNC, by setting bit 2 (OVE) of parameter No. 8001. When dry run signal DRNE is set to “1”, the specified rapid traverse rate and cutting feedrate are ignored and the tool moves at the dry run speed (set in parameter No. 1410) multiplied by the specified override. Bit 3 (RDE) of parameter No. 8001 can be used to specify whether to enable or disable dry run for rapid traverse.

When 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.

Feedrate of Dry run

Manual rapid traverse select signal	Command from PMC	
	Rapid traverse	Feed
1	Rapid traverse rate	Maximum cutting feedrate
0	Dry run feed rate × JV(*)	Dry run feed rate × JV

* Can also be set to the rapid traverse rate with bit 3 (RDE) of parameter No. 8001.

31 Override 0% signal EOV0

[Classification] Output signal

[Function] This signal is set to “1” when the feedrate override is 0%.

32 Skip signal ESKIP

[Classification] Input signal

[Function] When this signal is set to “1” during executing the skip cutting command, the block being executed is immediately stopped and the next block is executed. Bit 7 (SKE) of parameter No. 8001 can be used to select whether to use signal SKIP, which is the common skip signal for the PMC and CNC, or PMC–specific skip signal ESKIP.

NOTE
When two paths are controlled, the skip signal ESKIP is valid for the first path. For the second path, use the SKIP signal (X013, #7), which is used also by the CNC.

33 Distribution completion signals EADEN1 to EADEN8

[Classification] Output signal

[Function] These signals are set to “0” when the tool is moving with a command from the PMC. The signals are set to “1” when the tool is not moving, except when it is stopped by the issue of an axis control temporary stop signal ESTPg during the execution of a move command.

34 Buffer full signal EABUFg

[Classification] Output signal

[Function] This signal is set to “1” when the input buffer contains a command block.

35 Controlling signals EACNT1 to EACNT8

[Classification] Output signal

[Function] When the control axis selection status signal *EAXSL is set to “1”, signal EACNTn corresponding to the axis being controlled is set to “1”.

36 External deceleration signals *+ED1 to *+ED8/*–ED1 to *–ED8

[Classification] Input signal

[Function] These signals are also used by the CNC. The signals are provided for each direction of the individual controlled axes. The plus or minus sign in the signal name indicates the direction, and the number at the end corresponds to the axis number of the controlled axis.

```
± EDx
│   └── 1 … Feed along the first axis is decelerated.
│       :
│       8 …Feed along the eighth axis is decelerated.
│
└────── + … Feed in the positive direction is decelerated.
        :
        – … Feed in the negative direction is decelerated.
```

When any of the following axis control command is specified while the EDC bit (bit 0 of parameter 8005) is held 1, the external deceleration function becomes effective:

(1) Rapid traverse (EC0g to EC6g: 00h)
(2) Cutting feed per minute (EC0g to EC6g: 01h)
(3) Reference position return (EC0g to EC6g: 05h)
(4) Continuous feed (EC0g to EC6g: 06h)
(5) First reference position return (EC0g to EC6g: 07h)
(6) Second reference position return (EC0g to EC6g: 08h)
(7) Third reference position return (EC0g to EC6g: 09h)
(8) Fourth reference position return (EC0g to EC6g: 0Ah)
(9) Machine coordinate system selection (EC0g to EC6g: 20h)

While the signal is held “0”, the feedrate in the corresponding direction of axial movement can be reduced to the speed specified in parameter 1427 (external deceleration speed).

When the feedrate is lower than the external deceleration speed, the specified feedrate is valid.

The feedrate on other axes for which the signal is not held “0” is not affected.

37 Accumulated zero check signal ELCKZg

[Classification] Input signal

[Function] Setting this signal to 1 causes an accumulated zero check between blocks to be made at a subsequent cutting feed command.

(1) Cutting feed per minute (EC0g to EC6g: 01h)
(2) Cutting feed per rotation (EC0g to EC6g: 02h)

This enables the chopping function.

38 Torque control mode signal TRQMx

[Classification] Output signal

[Function] This signal indicates which axis is placed in torque control mode under PMC axis control.

Signal address

MT→CNC

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
X004	SKIP	ESKIP						

PMC→CNC

ADDRESS	#7	#6	#5	#4	#3	#2	#1	#0
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1
G120	*–ED8	*–ED7	*–ED6	*–ED5	*–ED4	*–ED3	*–ED2	*–ED1
G136	EAX8	EAX7	EAX6	EAX5	EAX4	EAX3	EAX2	EAX1
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E

For group A

	#7	#6	#5	#4	#3	#2	#1	#0
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA		EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

For group B

	#7	#6	#5	#4	#3	#2	#1	#0
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB		EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

For group C

	#7	#6	#5	#4	#3	#2	#1	#0
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC		EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

For group D

	#7	#6	#5	#4	#3	#2	#1	#0
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD		EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

CNC→PMC

ADDRESS

Address	#7	#6	#5	#4	#3	#2	#1	#0
F112	EADEN8	EADEN7	EADEN6	EADEN5	EADEN4	EADEN3	EADEN2	EADEN1
F129	*EAXSL		EOV0					

For group A

Address	#7	#6	#5	#4	#3	#2	#1	#0
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131							EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A

For group B

Address	#7	#6	#5	#4	#3	#2	#1	#0
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134							EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B

For group C

Address	#7	#6	#5	#4	#3	#2	#1	#0
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137							EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C

For group D

Address	#7	#6	#5	#4	#3	#2	#1	#0
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140							EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D

Group	Address	#7	#6	#5	#4	#3	#2	#1	#0
Group A	F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
Group B	F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
Group C	F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
Group D	F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D

	#7	#6	#5	#4	#3	#2	#1	#0
F182	EACNT8	EACNT7	EACNT6	EACNT5	EACNT4	EACNT3	EACNT2	EACNT1
F190	TRQM8	TRQM7	TRQM6	TRQM5	TRQM4	TRQM3	TRQM2	TRQM1

Parameter

1427	External deceleration speed of each axis

[Data type] Word axis type

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotation axis	1 deg/min	6 to 15000	6 to 12000

	#7	#6	#5	#4	#3	#2	#1	#0
1803				TRF				

[Data type] Bit type

TRF In torque control mode,

0 : Follow–up is not performed.
1 : Follow–up is performed.

1885	Permissible value of integrated travel distance under torque control

[Data type] Word axis type

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the maximum permissible value for the integrated travel distance in torque control mode. If the integrated travel distance exceeds the value set in this parameter, servo alarm No. 423 is issued.

NOTE
This parameter is valid when the TRF (bit 4 of parameter No. 1803) is set to 0.

1886	Torque control cancel limit

[Data type] Word axis type

[Unit of data] Detection unit

[Valid data range] 0 to 32767

This parameter sets the cancel limit value used to cancel torque control mode.

When the position error falls below or equals to the value set in this parameter, torque control mode is canceled and position control becomes effective.

NOTE
This parameter is valid when the TRF (bit 4 of parameter 1803) is held 0.

	#7	#6	#5	#4	#3	#2	#1	#0
2000							DGPRx	

[Data type] Bit axis type

DGPRx At power–ON, the torque constant (parameter No. 2105):
0 : Is automatically set to the standard value specific to the motor.
1 : Is not automatically set to the standard value specific to the motor.

	#7	#6	#5	#4	#3	#2	#1	#0
2007	TRQx							

[Data type] Bit axis type

TRQx Torque control:
0 : Is not performed for the axis.
1 : Is performed for the axis.

2105	Torque constant

[Data type] Word axis type

[Unit of data] 0.00001 Nm/(torque command)

[Valid data range] 1 to 32767

A torque constant is set for each motor torque characteristic.

NOTE
When a linear motor is used, the units of data are 0.001 N/(torque command).

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

[Data type] Bit type

PCF Whether movement along PMC–controlled axes is included in the actual speed display
0 : Included
1 : Not included

NOTE
This parameter is valid when the same axis is controlled alternately by the CNC and PMC.

	#7	#6	#5	#4	#3	#2	#1	#0
8001	SKE	AUX	NCC		RDE	OVE		MLE

[Data type] Bit type

MLE Whether machine lock signal MLK is valid for PMC–controlled axes

0 : Valid
1 : Invalid

> **NOTE**
> Each–axis machine lock signals MLK1 to MLK8 are always valid, regardless of the setting of this parameter.

OVE Signals related to dry run and override used in PMC axis control

0: Same signals as those used for the CNC
(1) Feedrate override signals *FV0 to *FV7
(2) Override cancellation signal OVC
(3) Rapid traverse override signals ROV1 and ROV2
(4) Dry run signal DRN
(5) Rapid traverse selection signal RT

1: Signals specific to the PMC
(1) Feedrate override signals *FV0E to *FV7E
(2) Override cancellation signal OVCE
(3) Rapid traverse override signals ROV1E and ROV2E
(4) Dry run signal DRNE
(5) Rapid traverse selection signal RTE

RDE Whether dry run is valid for rapid traverse in PMC axis control

0 : Invalid
1 : Valid

NCC When a travel command is issued for a PMC–controlled axis (selected by a controlled–axis selection signal) according to the program:

0 : P/S alarm 139 is issued while the PMC controls the axis with an axis control command. While the PMC does not control the axis, a CNC command is enabled.
1 : P/S alarm 139 is issued unconditionally.

AUX The number of bytes for the code of an auxiliary function (12H) command to be output is

0 : 1 byte type (0 to 255)
1 : 2 bytes type (0 to 65535)

SKE Skip signal during axis control by the PMC

0 : Uses the same signal SKIP (X004#7) as CNC.
1 : Uses dedicated axis control signal ESKIP (X004#6) used by the PMC.

> **NOTE**
> If SKE is set to 1 when 2–path control is being applied, it is effective only for path 1. Path 2 uses the same signal (SKIP <X013#7>) as that for the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10	SUE	DWE	RPD

[Data type] Bit type

RPD Rapid traverse rate for PMC–controlled axes

0 : Feedrate specified with parameter No. 1420
1 : Feedrate specified with the feedrate data in an axis control command

DWE Minimum time which can be specified in a dwell command in PMC axis control when the increment system is IS–C

0 : 1 ms
1 : 0.1 ms

SUE Whether acceleration/deceleration is performed for an axis that is synchronized with external pulses, for external pulse synchronization commands in PMC axis control

0 : Performed (exponential acceleration/deceleration)
1 : Not performed

F10 Least increment for the feedrate for cutting feed (per minute) in PMC axis control

F10	Metric input	Inch input
0	1 mm/min	0.01 inch/min
1	10 mm/min	0.1 inch/min

PF1, PF2 Set the the feedrate unit of feed per minute in PMC axis control

PF2	PF1	Feedrate unit
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

FR1, FR2 Set the feedrate unit for feed per rotation for an axis controlled by the PMC.

FR2	FR1	Metric input	Inch input
0	0	0.0001 mm/rev	0.000001 inch/rev
1	1		
0	1	0.001 mm/rev	0.00001 inch/rev
1	0	0.01 mm/rev	0.0001 inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

NOTE
When this parameter is set, the power must be turned off then back on again to make the setting effective.

[Data type] Bit type

PIM If a linear axis is controlled solely by the PMC, the commands for that axis are:

0 : Affected by inch/metric input.

1 : Not affected by inch/metric input.

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL	G8R	G8C	JFM	NMT	CMV
		NCI	DSL	G8R	G8C	JFM	NMT	CMV

[Data type] Bit type

CMV If the PMC issues a command for an axis after the completion of a movement along that axis according to a command issued by the CNC, but before receiving the signal indicating that the miscellaneous function is completed in the same block,

0 : P/S alarm No. 130 occurs.

1 : The axis is handled as a PMC axis and the specified movement is executed.

NMT If the CNC issues a command that does not result in any movement along a PMC–controlled axis while another command, specified for the axis, is being processed,

0 : P/S alarm No. 130 occurs.

1 : No alarm occurs.

JFM Specifies the units used to specify the feedrate for continuous feed (06H) for a PMC–controlled axis.

Increment system	JFM	Metric input	Inch input	Rotation axis
IS–B	0	1 mm/min	0.01 inch/min	0.00023 rpm
	1	200 mm/min	2.00 inch/min	0.046 rpm
IS–C	0	0.1 mm/min	0.001 inch/min	0.000023 rpm
	1	20 mm/min	0.200 inch/min	0.0046 rpm

G8C If a cutting feed command is specified for a PMC–controlled axis, look–ahead control is:

0 : Disabled.

1 : Enabled.

> **NOTE**
> The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

G8R If a rapid traverse command is specified for a PMC–controlled axis, look–ahead control is:

0 : Disabled.
1 : Enabled.

> **NOTE**
> The above setting is effective only when the NAHx bit (bit 7 of parameter No. 1819) is set to 0.

DSL If an axis exchange is attempted while the selection of a PMC–controlled axis is inhibited,

0 : The attempt fails and a P/S alarm No. 139 occurs.
1 : The axis exchange is executed for the axes that belong to an unspecified path.

NCI During deceleration along an axis controlled by the PMC, the in–position check is:

0 : Performed.
1 : Not performed.

NDI When diameter programming is selected for a PMC controlled axis, under PMC axis control:

0 : Radius programming is used to specify the travel distance and feedrate.
1 : Diameter programming is used to specify the travel distance and feedrate.

> **NOTE**
> While the CDI bit (bit 1 of parameter No. 8005) is held 0, the NDI bit is valid for an axis of diameter programming (the DIAx bit (bit 3 of parameter No. 1006) is set to 1).

	#7	#6	#5	#4	#3	#2	#1	#0
8005	MFD				DRR	R10	CDI	EDC
	MFD				DRR	R10		EDC

[Data type] Bit

EDC Under PMC axis control, the external deceleration signal is:

0 : Ineffective.
1 : Effective.

CDI When diameter programming is selected for a PMC controlled axis, under PMC axis control:

0 : Radius programming is used to specify the travel distance and feedrate.
1 : Diameter programming is used to specify the travel distance, and radius programming is used to specify the feedrate.

NOTE
1 This parameter is valid when the DIAx bit (bit 3 of parameter 1006) is held 1.
2 When the CDI bit is set to 1, the NDI bit (bit 7 of parameter 8004) is invalid.

R10 When the RPD bit (bit 0 of parameter 8002) is set to 1, the units for specifying the rapid traverse rate on the PMC axis are:
0 : 1 mm/min.
1 : 10 mm/min.

DRR During cutting feed per rotation under PMC axis control, dry run is:
0 : Not effective.
1 : Effective.

MFD : The miscellaneous function individual output of the PMC axis control function is:
0 : Not effective.
1 : Effective.

8010	Selection of the DI/DO group for each axis controlled by the PMC

[Data type] Byte axis type

[Valid data range] 1 to 4

Specify the DI/DO group to be used to specify a command for each PMC–controlled axis.

Value	Description
1	DI/DO group A (G142 to G153) is used.
2	DI/DO group B (G154 to G165) is used.
3	DI/DO group C (G166 to G177) is used.
4	DI/DO group D (G178 to G189) is used.

NOTE
If another value is specified, the axis is not PMC–controlled.

8022	Maximum feedrate for feed per rotation along a PMC–controlled axis

[Data type] Word axis type

[Unit of data]

[Valid data range]

Increment system	Unit of data	Valid data range	
		IS–A, IS–B	IS–C
Millimeter machine	1 mm/min	6 to 15000	6 to 12000
Inch machine	0.1 inch/min	6 to 6000	6 to 4800
Rotaion axis	1 deg/min	6 to 15000	6 to 12000

Specify the maximum feedrate for feed per rotation along a PMC–controlled axis

> **NOTE**
> The maximum feedrate set to first axis is valid to all axes. The data of after second axis are invalid.

8028	Linear acceleration/deceleration time constant for jog feed specified by the speed command for each PMC–controlled axis

[Data type] Word axis type

[Unit of data] msec/1000 rpm

[Valid data range] 0 to 32767

Specify, for each PMC–controlled axis, the time needed to increase or decrease the speed of the servo motor by 1000 rpm, that is, the time constant of linear acceleration/deceleration for jog feed according to the speed specified for that axis.

> **NOTE**
> If 0 is specified, the system does not control the acceleration/deceleration.

Alarm and message

A servo alarm or overtravel alarm for a PMC–controlled axis is detected in the same way as an alarm for a CNC–controlled axis.

If an alarm occurs, the alarm is handled by 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 (G12.1), a P/S alarm No. 130 occurs.

(1) P/S Alarm

Number	Message	Description
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
139	CAN NOT CHANGE PMC CONTROL AXIS	A 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.

(2) Servo alarm

Number	Message	Description
417	SERVO ALARM: n AXIS EXCESS ERR	An invalid parameter is specified for torque control. The torque constant parameter is set to 0.
422	SERVO ALARM: n AXIS EXCESS ERR	The maximum speed permitted under torque control has been exceeded.
423	SERVO ALARM: n AXIS EXCESS ERR	The maximum permissible value of integrated travel distance set in the parameter under torque control has been exceeded.

Warning

WARNING

1 The mode selection, CNC reset, and other CNC statuses have no effect.

2 Feed hold, single block stop, reset, or interlock of one or all axes, performed by the CNC, does not affect a PMC–controlled axis. Similar control is possible by using the equivalent signals issued from the PMC.

3 The mirror image functions (setting, parameter, input signal) are disabled.

Caution

CAUTION

1 Emergency stop or machine lock is enabled. Machine lock can be disabled if the MLE bit (bit 0 of parameter No. 8001) is 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 Commands for a linear axis that is controlled solely by the PMC (not used as an axis controlled by the CNC) are not affected by inch/metric input if the PIM bit (bit 0 of parameter No. 8003) is specified accordingly. The current position display, also, is not affected by inch/metric input.
4 For a PMC–controlled axis, manual absolute mode is always set. If the PMC starts control of an axis after manual intervention (manual continuous feed, manual handle feed, etc.) is performed during automatic operation while manual absolute mode is not set (*ABSM is set to 1), manual absolute mode is set.
5 Under PMC axis control, all commands are handled as axis commands. Even for the miscellaneous function, the position check is effective.
6 When the CNC specifies the command of workpiece coordinate system setting (G54 to G59) during an axial movement by the PMC, a correct coordinate system cannot be set.

Note

NOTE

1 The actual speed excluding the effect of the movement along a PMC–controlled axis can be displayed if the PCF bit (bit 1 of parameter No. 3105) 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.
4 The individual output of the miscellaneous function is provided by adding a signal for individual output. The timing diagram of controlling and specifying the miscellaneous function is not changed. The normal specifications of the miscellaneous functions for PMC axis control function are applied.

15.2 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.

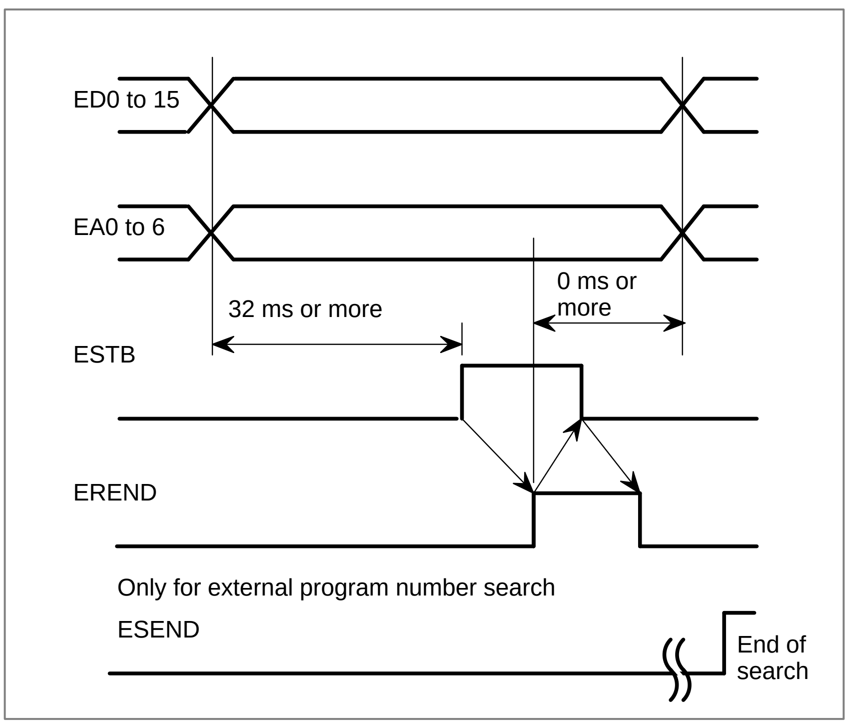
(4) After reading the address, the control unit sets read completion signal EREND to "1".

(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

No.	Item	ESTB EA6 EA5 EA4	EA3 EA2 EA1 EA0	ED15 to ED0			
				15141312	1110 9 8	7 6 5 4	3 2 1 0
1	External program number search	1 0 0 0	××××	Program number(BCD4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9
2	External tool compensa-tion	1 0 0 1	××××	Offset value(BCD 4 digits with sign)			
				±0 to 7	0 to 9	0 to 9	0 to 9
3	External workpiece coordi-nate system shift	1 0 1 0	axis code	Shift value(BCD 4 digits with sign)			
				±0 to 7	0 to 9	0 to 9	0 to 9
4	External machine coordi-nate system shift	1 0 1 1	axis code	Machine coordinate system shift value(binary) ±0 to 9999			
5	Alarm set	1 1 0 0	0 0 0 0	Alarm No.(binary)			
					0 to 999		
	Alarm clear	1 1 0 0	0 0 0 1	Alarm No.(binary)			
					0 to 999		
	Oper to tor message list	1 1 0 0	0 1 0 0	Message No.(binary)			
					0 to 999		
	Operator message clear	1 1 0 0	0 1 0 1	Message No.(binary)			
					0 to 999		
	Message	1 1 0 0	0×1 1	character(Character code)			
6	Substitute No. of parts required	1 1 1 0	0 0 0 0	No. of parts required(BDC4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9
	Substitute No. of parts machined	1 1 1 0	0 0 0 1	No. of parts machined(BDC4 digits)			
				0 to 9	0 to 9	0 to 9	0 to 9

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
5th axis	0	1	0	0
6th axis	0	1	0	1
7th axis	0	1	1	0
8th axis	0	1	1	1

WARNING

Though bits EA4 to EA6 distinguish one set of data from another, the PMC must be interlocked in order to prevent other function data being fed during a process for which they are invalid.

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 in MEM mode.

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 ESR no. 6300#4=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, P/S 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 - +/-7.999mm or +/-0.7999 inch 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 LSB ← Offset amount →														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000 to 7999)

Specify "000"

0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

Specify "001" for external tool offset

Data specification 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 LSB ← Offset amount →														

sign: 0 : + / 1 : –

Specified by BCD 4-digit code (0000A7999)

EA1	EA0	Description
0	0	Specification of X axis offset
0	1	Specification of Z axis offset
1	0	Specification of tool nose radius compensation amount

0 : Specification of tool wear offset amount
1 : Specification of tool geometry offset amount

0 : Incremental specification
(Input value is added to present compensation amount)
1 : Absolute specification
(Input value is replaced with present compensation amount)

Specify "001" for external tool offset

3) External workpiece coordinate system shift

The external workpiece coordinate system shift adjusts the workpiece coordinate system depending on the shift amount set via PMC. Each axis (parameter No. 1220) has this shift amount, and it is added to all the workpiece coordinate systems for use. The shift amount is not lost by cut off of the power supply. It is not added incrementally, but each input shift amount makes a new shift amount. The amount that can be input is 0-+/-7.999 mm or +/-0.7999 inch.

4) External machine coordinate system shift

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 alarm message

(a) 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 32 characters long. The alarm condition is reset by external data.

(b) External operator message
The external operator message sends the operator message and number from the external unit to the CNC, with a display on the CRT screen of the CNC.
Only one message can be transmitted, with a potential message length of 255 characters. The alarm number ranges from 0 to 999; from 0 to 99, the CNC adds 2000 to the number, while from 100 to 999 the number is not displayed, only the message is displayed.

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 to ED8 . . . . . . Character code in 1st character.
> ED7 to 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	m
1	1	1	1	/	?	O	_	ッ	ソ	マ	#

SP : Space code

6) Substituting No. of parts required and No. of parts, machined

Substitution is possible for the No. of parts required and the No. of parts machined.

Data specifiecation 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 LSB Value of aprt nos.		

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 <G000, G001>

[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 <G002>

[Classification] Input signal

[Function] The signals indicate the type of the entered data.

Read signal for external data input ESTB <G002#7>

[Classification] Input signal

[Function] The signal reports that the address and data are set in external data input.

When the signal is set to “1”, the control unit reads the address and data for external data input.

[Operation] The “basic procedure” describes the procedure for, and operation of, the control unit when the signal turns to “1”.

Read completion signal for external data input EREND <F060#0>

[Classification] Output signal

[Function] 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 <F060#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
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0

	#7	#6	#5	#4	#3	#2	#1	#0
F060							ESEND	EREND

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3202		PSR						

[Data type] Bit type

PSR Search for the program number of a protected program

0 : Disabled
1 : Enabled

		#7	#6	#5	#4	#3	#2	#1	#0
6300					ESR				

[Data type] Bit type

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, PN16 <G009#0 to #4>

[Classification] Input signal

[Function] Select the number of a workpiece to be machined in the memory mode. Five code signals are provided. These signals are set as binary code to designate a workpiece number as follows:

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
0	0	0	0	0	00
0	0	0	0	1	01
0	0	0	1	0	02
0	0	0	1	1	03
0	0	1	0	0	04
0	0	1	0	1	05
0	0	1	1	0	06
0	0	1	1	1	07
0	1	0	0	0	08
0	1	0	0	1	09
0	1	0	1	0	10
0	1	0	1	1	11
0	1	1	0	0	12
0	1	1	0	1	13
0	1	1	1	0	14
0	1	1	1	1	15

Workpiece number search signal					Workpiece number
PN16	PN8	PN4	PN2	PN1	
1	0	0	0	0	16
1	0	0	0	1	17
1	0	0	1	0	18
1	0	0	1	1	19
1	0	1	0	0	20
1	0	1	0	1	21
1	0	1	1	0	22
1	0	1	1	1	23
1	1	0	0	0	24
1	1	0	0	1	25
1	1	0	1	0	26
1	1	0	1	1	27
1	1	1	0	0	28
1	1	1	0	1	29
1	1	1	1	0	30
1	1	1	1	1	31

Workpiece number 00 is used for special designation “no search”. Thus, a workpiece number ranges from 01 to 31.

NOTE

These signals are also used to specify a file number for file search during external program input. See Section 13.7, “External Program Input.”

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G009				PN16	PN8	PN4	PN2	PN1

Alarm and Message

Number	Message	Description
059	PROGRAM NUMBER NOT FOUND	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 DNC operation and MDI operation.
2 Select the program number from O001 ~ O031.
3 Program numbers from O001 to O031 can be used. However, programs corresponding to all the program numbers do not have to be stored in memory.
4 When a program corresponding to the specified program number is not stored in memory, an alarm (No. 059) is activated when the start button is pressed.
5 Program search is performed only when the start button is pressed in the reset state. When the CNC is in the automatic operation stop state (single block stop, etc.) or pause state (feedhold stop, etc.), program search is not performed even if the start button is pressed and execution is started from the point specified by the present execution pointer.
6 To restart 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 PN16 are “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 PN16 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.

The first, second, and third spindles all 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 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 <G0027, #3, #4, #5> can be used to stop a PMC–controlled spindle.

The spindle motor speed data is obtained from the following expression. Its value can range from 0 to 4095:

$$\text{Spindle motor speed data} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

Remark) Usually, the spindle speed must be controlled. If a gear train is used to connect the spindle to the spindle motor, first obtain the maximum spindle speed at the maximum spindle motor speed.

$$\text{Spindle motor speed data} = \frac{\text{Spindle speed}}{\text{Maximum spindle speed}} \times 4095$$

By using this expression, the spindle motor speed data can 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 <F022 to F025> and SF signal <F007, #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.

Specify the related bits of parameter No. 3705 according to the desired application, then use the S–code and SF signals.

Twelve code signals corresponding to the S value (output)

Twelve code signals corresponding to S value R01O to R12O <F036, #0 to F037, #3> are output to the first spindle motor. The output data is calculated from the results of the CNC spindle control. (See Section 9.3.)

Even while a spindle is subject to PMC control, an S command that is issued to the CNC is converted to spindle output data and output.

The SIND signal determines whether the speed output command, issued to the spindle motor, is obtained from the twelve code signals corresponding to the S value, or from the R01I to R12I signals calculated and specified by the PMC.

The use of this signal may simplify PMC ladder processing used to enable PMC spindle control.

Sample application 1)

Controlling the first and second spindles of a lathe system

→ Share the gear stages between the first and second spindles.

(If the first spindle uses two gears, for example, specify parameters Nos. 3743 and 3744, thus enabling the use of gears 3 and 4 for the second spindle.)

Perform the necessary setting to enable control of the first and second spindles by the PMC.

To specify a rotation command for the first spindle, enter the gears for the first spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the first spindle in the PMC control interface for the first spindle.

To specify a rotation command for the second spindle, enter the gears to be used for the second spindle in GR1 and GR2 and obtain the data of the twelve code signals corresponding to the S value. Specify the data as the speed output command for the second spindle in the PMC control interface for the second spindle.

Sample application 2)

Using a lathe's orientation function with the stop position of the serial spindle specified externally, specifying the S value as the angle of the stop position for spindle orientation after the spindle positioning mode has been selected

→ Use 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. 3744 to 360.)

Specify the M code used to set the spindle to positioning mode and stop the spindle. Enter gear 4 in GR1 and GR2.

Then, specify a spindle positioning angle with the S command. (To specify the position of 145 degrees, for example, specify S145;.)

Expression $145/360 \times 4095$ is calculated and the result is output to the twelve code signals corresponding to the S value (output signal). Enter the data in external stop position commands SHA00 to SHA11 <G078, #0 to G079, #3> and perform the orientation.

Signal

PMC spindle control signals

For the first spindle: SIND, SSIN, SGN <G033#7, #6, #5>
R01I to R12I <G032#0 to G033#3>
For the second spindle: SIND2, SSIN2, SGN2 <G035#7, #6, #5>
R01I2 to R12I2 <G034#0 to G035#3>
For the third spindle: SIND3, SSIN3, SGN3 <G037#7, #6, #5>
R01I3 to R12I3 <G036#0 to G037#3>

[Classification] Input signal

[Function] The above signals enable the control of a spindle motor by issuing commands from the PMC. Both the speed and polarity of the spindle motor (direction of rotation) can be controlled.

The speed command and polarity are usually specified by the CNC. The use of these signals allows the user to select whether the speed and polarity are controlled by the CNC or PMC.

Even if the multispindle control function is not provided, these signals allow the second or third spindle to be controlled.

When the multispindle control function and type A are being used (if the MSI bit, bit 2 of parameter No. 3709, is set to 0), the signals for the second and third spindles cannot be used.

- **Details of the signals**

- Signal used to select the spindle motor speed command SINDx

→ The above signal is used to select whether the spindle motor speed is controlled by the CNC or PMC.

1: The spindle motor is controlled according to speed commands (R01Ix to R12Ix) issued by the PMC.

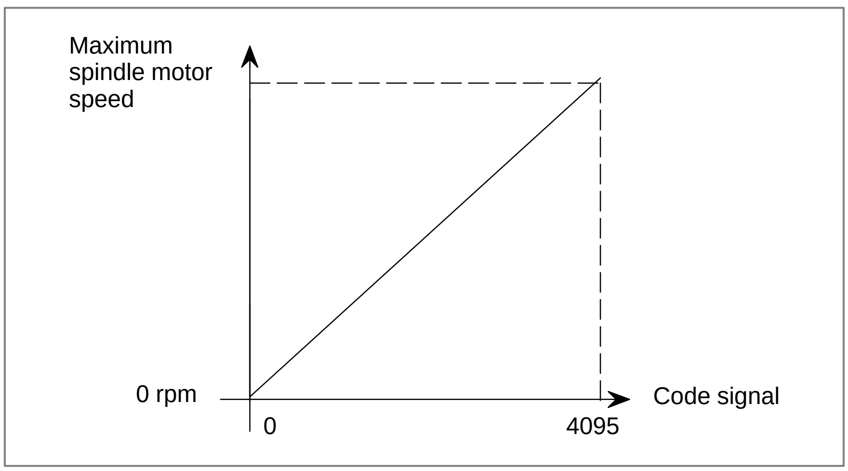
0: The spindle motor is controlled according to speed commands issued by the CNC. The spindle speed specified with the S command is output.

- Signals used to input the spindle motor speed command issued by the PMC R01Ix to R12Ix

→ If the PMC is being used to control the spindle motor speed command, specify, in binary format, the value obtained using the following expression.

$$\text{Value to be specified} = \frac{\text{Spindle motor speed}}{\text{Maximum spindle motor speed}} \times 4095$$

(Spindle motor speed)



- Signal used to select the polarity of the spindle motor speed command, SSINx

→ The above signal selects whether the output polarity of the spindle motor speed command is controlled by the CNC or PMC.

1 : The spindle motor is controlled according to the polarity command (SGNx) issued by the PMC.

0 : The CNC controls the polarity. The polarity is determined by the TCW and CWM bits (bits 7 and 6 of parameter No. 3706) and the M03 or M04 command.

- Signal used to specify the polarity of the spindle motor selected by the PMC, SGNx

→ If the PMC is used to control the output polarity of the spindle motor speed command, specify the polarity with this signal.

1 : The output polarity of the spindle is negative.

0 : The output polarity of the spindle is positive.

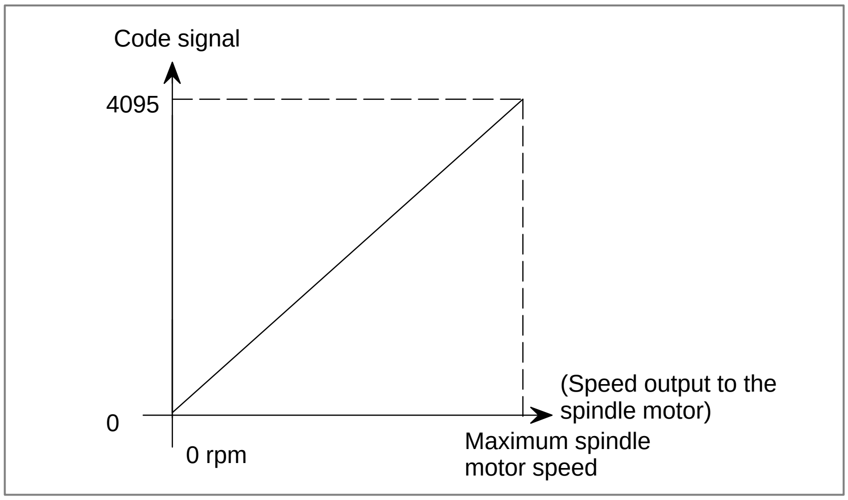
Twelve code signals corresponding to the S value R01O to R12O <G036#0 to G037#3>

[Classification] Output signal

[Function] The S value, specified in the CNC part program, is converted to the speed output 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.)



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
3701				SS2				

[Data type] Bit

SS2 The number of connections in serial spindle control

0 : 1 spindle
1 : 2 spindles

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				ESF
		SFA	NSF					ESF

[Data type] Bit type

ESF When the spindle control function (S analog output or S serial output) is used, and the constant surface speed control function is used or bit 7 (GTT) of parameter No. 3705 is set to 1:

0 : S codes and SF are output for all S commands.
1 : S codes and SF are not output for an S command in constant surface speed control mode (G96 mode) or for an S command used to specify maximum spindle speed clamping (G50S––;).

> **NOTE**
> For the T series, this parameter is enabled when bit 4 (EVS) of parameter No. 3705 is set to 1. For the M series, SF is not output:
> (1) For an S command used to specify maximum spindle speed clamping (G92S––;) in constant surface speed control mode
> (2) When bit 5 (NSF) of parameter No. 3705 is set to 1

EVS When the spindle control function (S analog output or S serial output) is used, S codes and SF are:

0 : Not output for an S command.
1 : Output for an S command.

> **NOTE**
> The output of S codes and SF for an S command in constant surface speed control mode (G96), or for an S command used to specify maximum spindle speed clamping (G50S––;) depends on the setting of bit 0 (ESF) of parameter No. 3705.

NSF: When an S code command is issued in constant surface–speed control,

0 : SF is output.
1 : SF is not output.

SFA: The SF signal is output:

0 : When gears are switched
1 : Irrespective of whether gears are switched

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		

[Data type] Bit type

MSI In multi–spindle control, the SIND signal is valid:

0 : Only when the first spindle is selected. (SIND signal for 2nd and 3rd spindle become invalid)
1 : For each spindle irrespective of whether the spindle is selected. (Each spindle has its own SIND signal.)

3821	Offset–voltage compensation value of the analog output of the third–spindle speed

[Data type] Word type

[Unit of data] Velo

[Valid data range] –1024 to 1024

Set the offset–voltage compensation value of the analog output of the third–spindle speed.

[Setting method]

(1) Set 0 (standard value)

(2) Specify a spindle speed at which the spindle speed analog output becomes 0.

(3) Measure output voltage.

(4) Set the following value to parameter No. 3731.

$$\text{Set value} = \frac{-8191 \quad \times \text{ offset voltage (V)}}{12.5}$$

(5) After the parameter has been set, command again a spindle speed where the spindle speed analog output become 0 and confirm that voltage becomes 0V.

Note

NOTE

1 If the spindle fails to move after the PMC issues the spindle motor speed command, check the following:

Type A is selected (the MSI bit, bit 2 of parameter No. 3709, is set to 0) when the multispindle control function is used.

→ The second 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 <G027, #3, #4, #5> stops the spindle.

M03/M04 is not specified when the CNC is being used to control the output polarity.

→ If the TCW bit, bit 7 of parameter No. 3706, is set to 1, the M03/M04 command issued to the CNC changes the output polarity for the spindle motor. If no M03/M04 command is specified after the CNC is turned on, the specified speed output is not sent to the spindle motor because the output polarity has not been determined.

2 The SF signal indicates that output of the S code to the PMC has been completed. The signal does not indicate the end of the command for specifying the spindle speed.

3 For an explanation of connecting the second or third spindle, see Sections 9.2 SPINDLE SERIAL OUTPUT/ SPINDLE ANALOG OUTPUT and 9.10 MULTI–SPINDLE CONTROL.

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

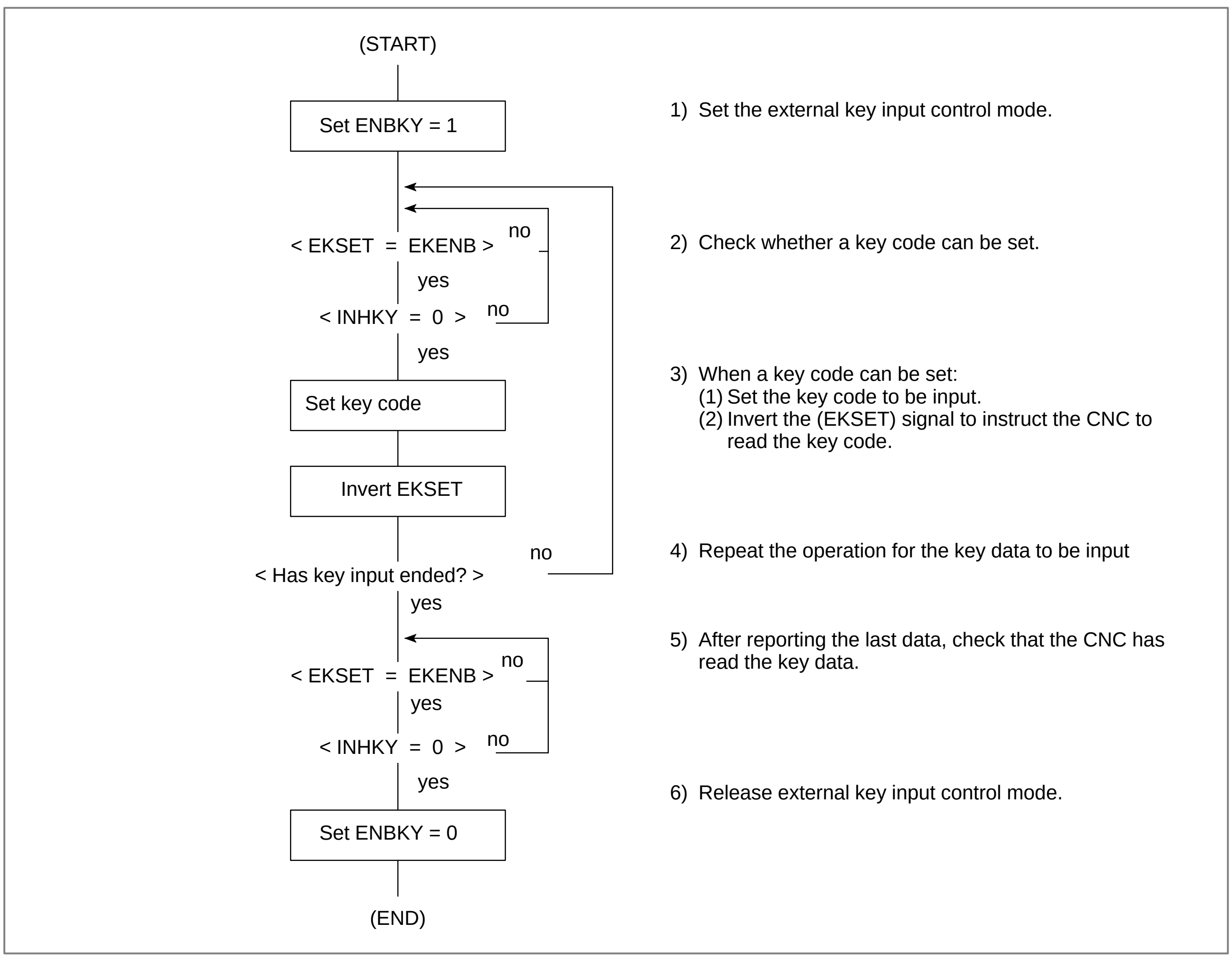
General

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)	EKC0 to EKC7
Key code read signal (input)	EKSET
Key code read completion signal (output)	EKENB
Key input disable signal (output)	INHKY
Program screen display mode signal (output)	PRGDPL

The processing flow in the PMC is shown below.



NOTE

Read processing is controlled by exclusive–ORing (XOR) the key code read signal (EKSET) with the read completion signal (EKENB). When the EKSET and EKENB signals differ in their logic, the CNC reads the input key code. Once reading has been completed, the CNC inverts the EKENB signal to match its logic with that of the EKSET signal.
In the PMC, on the other hand, a new key code cannot be set while the EKSET and EKENB signals differ in their logic.

Signals

External key input mode selection signal ENBKY <G066#1>

[Classification] Input signal

[Function] While this signal is turned on “1”, external key input control is enabled. In external key input control mode, any MDI key operations are ignored.

Key code read signal EKSET <G066#7>

[Classification] Input signal

[Function] This signal instructs the CNC to read the input key code.

Key code signals EKC0 to EKC7 <G098>

[Classification] Input signal

[Function] These signals set an input key code. (See the MDI key code table.)

Key input disable signal INHKY<F053#0>

[Classification] Output signal

[Function] While this signal is “1”, no key code is accepted in external key input control mode.

Program screen display mode signal PRGDPL <F053#1>

[Classification] Output signal

[Function] This signal is on "1" while the CNC is displaying a program screen.

Key code read completion signal EKENB <F053#7>

[Classification] Output signal

[Function] This signal reports that the CNC has read a key code.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G066	EKSET						ENBKY	
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0

	#7	#6	#5	#4	#3	#2	#1	#0
F053	EKENB						PRGDPL	INHKY

MDI key code table

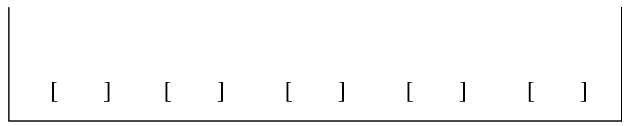
Codes in the table are given in hexadecimal. For example, [A] corresponds to 41H in hexadecimal. [RESET] corresponds to 90H in hexadecimal.

(Note 1) and (Note 2) are explained below:

NOTE

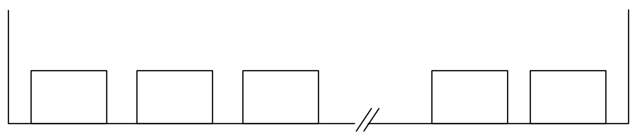
1 For the small keyboard, 0EDH is assigned to GRAPH/CUSTOM .
For the standard keyboard, 0EDH is assigned to GRAPH .
0EEH is assigned to CUSTOM .

2 Handling of the soft keys
[F0] to [F9], [FR], and [FL] in the key code table are the key codes for the soft keys. They are associated with the MDI keys as shown below.
Key configuration for 7.2″/8.4″ LCD, or etc. : 5 keys + 2 keys ([F0] to [F4] and [FR], [FL])



[FL] [F4] [F3] [F2] [F1] [F0] [FR]

Key configuration for 9.5″/10.4″ LCD, or etc.: 10 keys + 2 keys ([F0] to [F9] and [FR], [FL])



[FL] [F9] [F8] [F7] ··· [F1] [F0] [FR]

MDI Key Code Table(00H–7FH)

	0	1	2	3	4	5	6	7
0			Space	0	@	P		
1				1	A	Q		
2				2	B	R		
3			#	3	C	S		
4				4	D	T		
5				5	E	U		
6			&	6	F	V		
7				7	G	W		
8			(	8	H	X		
9			)	9	I	Y		
A	; (EOB)		*		J	Z		
B			+		K	[		
C			,		L			
D			–	=	M	]		
E			.		N			
F			/	?	O			

MDI Key Code Table(80H–0FFH)

	8	9	A	B	C	D	E	F
0		RESET						[F0] (Note2)
1								[F1] (Note2)
2								[F2] (Note2)
3								[F3] (Note2)
4		INSERT						[F4] (Note2)
5		DELETE						[F5] (Note2)
6	CAN	ALTER						[F6] (Note2)
7								[F7] (Note2)
8	Cursor →	INPUT					POS	[F8] (Note2)
9	Cursor ←						PROG	[F9] (Note2)
A	Cursor ↓	HELP					OFFSET SET-TING	
B	Cursor ↑						SYSTEM	
C							MES-SAEG	
D							GRAPH (CUSTOM) (Note1)	
E	PAGE ↓						CUSTOM (Note1)	[FR] (Note2)
F	PAGE ↑						FAPT	[FL] (Note2)

15.6 DIRECT OPERATION BY PMC/MMC

General

Activating memory operation in memory operation mode (MEM) with the direct operation select signal set to 1 enables machining (direct operation) while reading a program stored in the PMC–RC or MMC.

Signal

Direct operation select signal DMMC <G042#7>

[Classification] Input signal

[Function] Selects the mode (direct operation mode) for performing machining while reading a program stored in the PMC–RC and MMC.

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

- When memory operation mode (MEM) is not selected, the control unit ignores this signal.
- When memory operation mode (MEM) is selected, the control unit selects direct operation mode and enables direct operation.

Reference Item

FANUC PMC–MODEL RC/RC3/NB PROGRAMMING MANUAL (C LANGUAGE) (B–61863E–1)

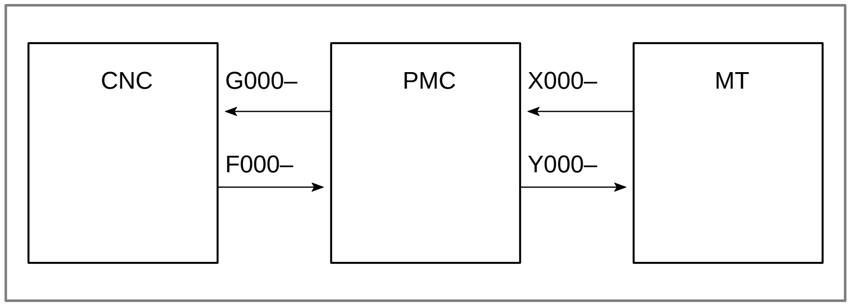
APPENDIX

A INTERFACE BETWEEN CNC AND PMC

A.1 LIST OF ADDRESSES

A.1.1 Series 16*i*/18*i*/160*i*/180*i* List of Addresses (One–Path Control)

Interface addresses among CNC, PMC and Machine Tool are as follows:



Following shows table of addresses:

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

[Example 1]

Signals EXLM and ST are common signals, STLK is for T series only and RLSOT and RVS are for M series only.

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

MT → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0	
X000									
X001									
X002									
X003									
X004	SKIP	ESKIP / SKIP6	–MIT2 / SKIP5	+MIT2 / SKIP4	–MIT1 / SKIP3	+MIT1 / SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
	SKIP	ESKIP / SKIP6	SKIP5	SKIP4	SKIP3	ZAE / SKIP2	YAE / SKIP8	XAE / SKIP7	(M series)
X005									
X006									
X007									
X008				*ESP					
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1	
X010									
X011									
X012									

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G003								
G004			MFIN3	MFIN2	FIN			
G005	BFIN	AFL		BFIN	TFIN	SFIN	EFIN	MFIN
G006		SKIPP		OVC		*ABSM		SRN
G007	RLSOT	EXLM	*FLWU	RLSOT3		ST	STLK	RVS
G008	ERS	RRW	*SP	*ESP				*IT
G009				PN16	PN8	PN4	PN2	PN1
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0
G013	*AFV7	*AFV6	*AFV5	*AFV4	*AFV3	*AFV2	*AFV1	*AFV0
G014							ROV2	ROV1
G015								
G016	F1D							MSDFON
G017								
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019	RT		MP2	MP1	HS3D	HS3C	HS3B	HS3A
G020								
G021								
G022								
G023	ALNGH	RGHTH						
G024								

	#7	#6	#5	#4	#3	#2	#1	#0
G025								
G026								
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC	SPSTP	*SCPF	*SUCPF		GR2	GR1	
G029		*SSTP	SOR	SAR		GR31		GR21
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G031	PKESS2	PKESS1						
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
G038	*BECLP	*BEUCP			SPPHS	SPSYC		*PLSST
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040	WOSET	PRC						OFN6
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042	DMMC				HS3ID	HS3IC	HS3IB	HS3IA
G043	ZRN		DNCI			MD4	MD2	MD1
G044							MLK	BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G046	DRN	KEY4	KEY3	KEY2	KEY1		SBK	
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0

	#7	#6	#5	#4	#3	#2	#1	#0
G050							*TLV9	*TLV8
G051	*CHLD	CHPST			*CHP8	*CHP4	*CHP2	*CHP0
G052	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G053	CDZ	SMZ			UINT			TMRON
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
G056								
G057								
G058		STWD	STRD		EXWT	EXSTP	EXRD	MINP
G059							TRRTN	TRESC
G060	*TSB							
G061			RGTSP2	RGTSP1				RGTAP
G062			PDT2	PDT1			*CRTOF	
G063			NOZAGC					
G064		ESRSYC					SLCSEQ	RTNCY
G065								
G066	EKSET			RTRCT			ENBKY	IGNVRY
G067								
G068								
G069								
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072	RCHHGA	MFNHGA	INCMDA	OVRIDA	DEFMDA	NRROA	ROTAA	INDXA
G073						MPOFA	SLVA	MORCMA
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB

	#7	#6	#5	#4	#3	#2	#1	#0
G075	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076	RCHHGB	MFNHGB	INCMDB	OVRIDB	DEFMDB	NRROB	ROTAB	INDXB
G077						MPOFB	SLVB	MORCMB
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08
G082	Reserve for order made macro							
G083	Reserve for order made macro							
G084								
G085								
G086					–Ja	+Ja	–Jg	+Jg
G087								
G088								
G089								
G090								
G091					SRLNI3	SRLNI2	SRLNI1	SRLNI0
G092				BGEN	BGIALM	BGION	IOLS	IOLACK
G093								
G094								
G095								
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	#7	#6	#5	#4	#3	#2	#1	#0	
G100	+J8	+J7	+J6	+J5	+J4	+J3	+J2	+J1	
G101									
G102	–J8	–J7	–J6	–J5	–J4	–J3	–J2	–J1	
G103									
G104									
G105									
G106	MI8	MI7	MI6	MI5	MI4	MI3	MI2	MI1	
G107									
G108	MLK8	MLK7	MLK6	MLK5	MLK4	MLK3	MLK2	MLK1	
G109									
G110	+LM8	+LM7	+LM6	+LM5	+LM4	+LM3	+LM2	+LM1	
G111									
G112	–LM8	–LM7	–LM6	–LM5	–LM4	–LM3	–LM2	–LM1	
G113									
G114	*+L8	*+L7	*+L6	*+L5	*+L4	*+L3	*+L2	*+L1	
G115									
G116	*–L8	*–L7	*–L6	*–L5	*–L4	*–L3	*–L2	*–L1	
G117									
G118	*+ED8	*+ED7	*+ED6	*+ED5	*+ED4	*+ED3	*+ED2	*+ED1	
G119									
G120	*–ED8	*–ED7	*–ED6	*–ED5	*–ED4	*–ED3	*–ED2	*–ED1	
G121									
G122	PK8 PKESS2	PK7 PKESS1	PK6	PK5	PK4	PK3	PK2	PK1	(T series)
	PKESS2	PKESS1							(M series)
G123									

	#7	#6	#5	#4	#3	#2	#1	#0
G124	DTCH8	DTCH7	DTCH6	DTCH5	DTCH4	DTCH3	DTCH2	DTCH1
G125								
G126	SVF8	SVF7	SVF6	SVF5	SVF4	SVF3	SVF2	SVF1
G127								
G128								
G129								
G130	*IT8	*IT7	*IT6	*IT5	*IT4	*IT3	*IT2	*IT1
G131								
G132					+MIT4	+MIT3	+MIT2	+MIT1
G133								
G134					–MIT4	–MIT3	–MIT2	–MIT1
G135								
G136	EAX8	EAX7	EAX6	EAX5	EAX4	EAX3	EAX2	EAX1
G137								
G138	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4	SYNC3	SYNC2	SYNC1
G139								
G140	SYNCJ8	SYNCJ7	SYNCJ6	SYNCJ5	SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1
G141								
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA		EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A

	#7	#6	#5	#4	#3	#2	#1	#0
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E
G152								
G153								
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB		EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G162								
G163								
G164								
G165								
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC		EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

	#7	#6	#5	#4	#3	#2	#1	#0
G174								
G175								
G176								
G177								
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD		EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D
G186								
G187								
G188								
G189								
G190								
G191								
G192	IGVRY8	IGVRY7	IGVRY6	IGVRY5	IGVRY4	IGVRY3	IGVRY2	IGVRY1
G193								
G194								
G195								
G196								
G197								
G198	NPOS8	NPOS7	NPOS6	NPOS5	NPOS4	NPOS3	NPOS2	NPOS1

	#7	#6	#5	#4	#3	#2	#1	#0
G199								
G200								
G201								
G202								
G203								
G204	MRDYC	ORCML	SFRC	SRVC	CTH1C	CTH2C	TLMHC	TLMLC
G205	RCHC	RSLC	INTGC	SOCNC	MCFNC	SPSLC	*ESPC	ARSTC
G206	RCHHGC	MFNHGC	INCMDC	OVRIDC	DEFMDC	NRROC	ROTAC	INDXC
G207						MPOFC	SLVC	MORCMC
G208	SHC07	SHC06	SHC05	SHC04	SHC03	SHC02	SHC01	SHC00
G209					SHC11	SHC10	SHC09	SHC08
G210								
G211								
G212								
G213								
G214								
G215								
G216								
G217								
G218								
G219								
G220								
G221								
G222								

CNC → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP	SA	STL	SPL				RWD
F001	MA		TAP	ENB	DEN	BAL	RST	AL
F002	MDRN	CUT		SRNMV	THRD	CSS	RPDO	INCH
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF	MAFL	MSBK	MABSM	MMLK	MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2
F006								
F007	BF			BF	TF	SF	EFD	MF
F008			MF3	MF2				EF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308
F018								
F019								
F020								
F021								
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16

	#7	#6	#5	#4	#3	#2	#1	#0
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24
F034						GR3O	GR2O	GR1O
F035								SPAL
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O
F038					ENB3	ENB2	SUCLP	SCLP
F039					CHPCYL	CHPMD		
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
F042								
F043								
F044				SYCAL	FSPPH	FSPSY	FSCSL	
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
F047							INCSTA	PC1DEA
F048								
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB

	#7	#6	#5	#4	#3	#2	#1	#0
F050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
F051							INCSTB	PC1DEB
F052								
F053	EKENB			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
F060							ESEND	EREND
F061							BCLP	BUCLP
F062	PRTSF							
F063	PSYN		RCYO			PSAR	PSE2	PSE1
F064						TLCHI	TLNW	TLCH
F065		SYNMOD		RTRCTF			RGSPM	RGSPP
F066	EXHPCC	MMPCC	PECK2		RTNMVS			G08MD
F067								
F068								
F069	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071							PSW10	PSW09
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								

	#7	#6	#5	#4	#3	#2	#1	#0
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O	RTAP		MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	– J4O	+ J4O	– J3O	+ J3O	– J2O	+ J2O	– J1O	+ J1O
F082						RVSL		
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090					ABTSP3	ABTSP2	ABTSP1	ABTQSV
F091								
F092			TRSPS		TRACT			
F093								
F094	ZP8	ZP7	ZP6	ZP5	ZP4	ZP3	ZP2	ZP1
F095								
F096	ZP28	ZP27	ZP26	ZP25	ZP24	ZP23	ZP22	ZP21
F097								
F098	ZP38	ZP37	ZP36	ZP35	ZP34	ZP33	ZP32	ZP31
F099								

	#7	#6	#5	#4	#3	#2	#1	#0
F100	ZP48	ZP47	ZP46	ZP45	ZP44	ZP43	ZP42	ZP41
F101								
F102	MV8	MV7	MV6	MV5	MV4	MV3	MV2	MV1
F103								
F104	INP8	INP7	INP6	INP5	INP4	INP3	INP2	INP1
F105								
F106	MVD8	MVD7	MVD6	MVD5	MVD4	MVD3	MVD2	MVD1
F107								
F108	MMI8	MMI7	MMI6	MMI5	MMI4	MMI3	MMI2	MMI1
F109								
F110	MDTCH8	MDTCH7	MDTCH6	MDTCH5	MDTCH4	MDTCH3	MDTCH2	MDTCH1
F111								
F112	EADEN8	EADEN7	EADEN6	EADEN5	EADEN4	EADEN3	EADEN2	EADEN1
F113								
F114	TRQL8	TRQL7	TRQL6	TRQL5	TRQL4	TRQL3	TRQL2	TRQL1
F115								
F116	FRP8	FRP7	FRP6	FRP5	FRP4	FRP3	FRP2	FRP1
F117								
F118	SYN8O	SYN7O	SYN6O	SYN5O	SYN4O	SYN3O	SYN2O	SYN1O
F119								
F120	ZRF8	ZRF7	ZRF6	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1
F121								
F122	HDO7	HDO6	HDO5	HDO4	HDO3	HDO2	HDO1	HDO0
F123								
F124								

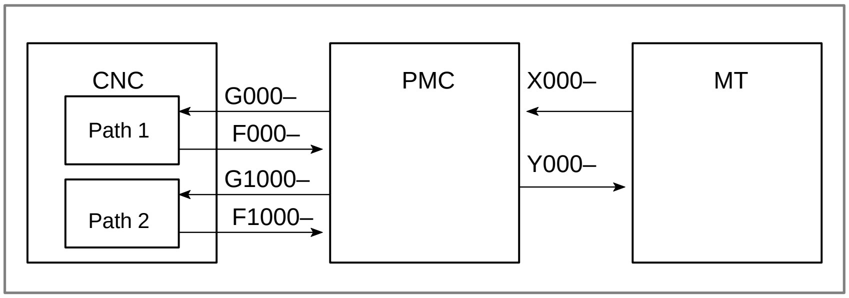
	#7	#6	#5	#4	#3	#2	#1	#0
F125								
F126								
F127								
F128								
F129	*EAXSL		EOV0					
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131							EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134							EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137							EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140							EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
F143								
F144								
F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
F146								
F147								
F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
F149								

	#7	#6	#5	#4	#3	#2	#1	#0
F150								
F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168	ORARC	TLMC	LDT2C	LDT1C	SARC	SDTC	SSTC	ALMC
F169	MORA2C	MORA1C	PORA2C	SLVSC	RCFNC	RCHPC	CFINC	CHPC
F170							INCSTC	PC1DEC
F171								
F172								
F173								
F174								

	#7	#6	#5	#4	#3	#2	#1	#0
F175								
F176								
F177	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F178					SRLNO3	SRLNO2	SRLNO1	SRLNO0
F179								
F180	CLRCH8	CLRCH7	CLRCH6	CLRCH5	CLRCH4	CLRCH3	CLRCH2	CLRCH1
F181								
F182	EACNT8	EACNT7	EACNT6	EACNT5	EACNT4	EACNT3	EACNT2	EACNT1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								

A.1.2 Series 16*i*/18*i*/160*i*/180*i* List of Addresses (Two–path Control)

Interface addresses among CNC, PMC and Machine Tool are as follows:



Signals addresses for each path are usually assigned as follows:

Signal address	Contents
G000–G255	Signals on path 1(PMC→CNC)
F000–F255	Signals on path 1(CNC→PMC)
G1000–G1255	Signals on path 2(PMC→CNC)
F1000–F1255	Signals on path 2(CNC→PMC)

However, for the signals common to both paths, those signals are assigned to path 1. Interface signals between the CNC and PMC are as shown below:

The signals with suffix #1 are those for path 1 and the signals with suffix #2 are those for path 2.

MT → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0	
X000									
X001									
X002									
X003									
X004	SKIP #1	ESKIP / SKIP6 #1	–MIT2 #1 / SKIP5 #1	+MIT2 #1 / SKIP4 #1	–MIT1 #1 / SKIP3 #1	+MIT1 #1 / SKIP2 #1	ZAE #1 / SKIP8 #1	XAE #1 / SKIP7 #1	(T series)
	SKIP #1	ESKIP / SKIP6 #1	SKIP5 #1	SKIP4 #1	SKIP3 #1	ZAE #1 / SKIP2 #1	YAE #1 / SKIP8 #1	XAE #1 / SKIP7 #1	(M series)
X005									
X006									
X007		*DEC7 #2	*DEC6 #2	*DEC5 #2	*DEC4 #2	*DEC3 #2	*DEC2 #2	*DEC1 #2	
X008				*ESP					
X009		*DEC7 #1	*DEC6 #1	*DEC5 #1	*DEC4 #1	*DEC3 #1	*DEC2 #1	*DEC1 #1	
X010									
X011									
X012									
X013	SKIP #2	SKIP6 #2	–MIT2 #2 / SKIP5 #2	+MIT2 #2 / SKIP4 #2	–MIT1 #2 / SKIP3 #2	+MIT1 #2 / SKIP2 #2	ZAE #2 / SKIP8 #2	XAE #2 / SKIP7 #2	(T series)
	SKIP #2	SKIP6 #2	SKIP5 #2	SKIP4 #2	SKIP3 #2	ZAE #2 / SKIP2 #2	YAE #2 / SKIP8 #2	XAE #2 / SKIP7 #2	(M siries)

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7 #1	ED6 #1	ED5 #1	ED4 #1	ED3 #1	ED2 #1	ED1 #1	ED0 #1
G001	ED15 #1	ED14 #1	ED13 #1	ED12 #1	ED11 #1	ED10 #1	ED9 #1	ED8 #1
G002	ESTB #1	EA6 #1	EA5 #1	EA4 #1	EA3 #1	EA2 #1	EA1 #1	EA0 #1
G003								
G004			MFIN3 #1	MFIN2 #1	FIN #1			
G005	BFIN #1	AFL #1		BFIN #1	TFIN #1	SFIN #1	EFIN #1	MFIN #1
G006		SKIPP #1		OVC #1		*ABSM #1		SRN #1
G007	RLSOT #1	EXLM #1	*FLWP #1	RLSOT3 #1		ST #1	STLK #1	RVS #1
G008	ERS #1	RRW #1	*SP #1	*ESP #1				*IT #1
G009				PN16 #1	PN8 #1	PN4 #1	PN2 #1	PN1 #1
G010	*JV7 #1	*JV6 #1	*JV5 #1	*JV4 #1	*JV3 #1	*JV2 #1	*JV1 #1	*JV0 #1
G011	*JV15 #1	*JV14 #1	*JV13 #1	*JV12 #1	*JV11 #1	*JV10 #1	*JV9 #1	*JV8 #1
G012	*FV7 #1	*FV6 #1	*FV5 #1	*FV4 #1	*FV3 #1	*FV2 #1	*FV1 #1	*FV0 #1
G013	*AFV7 #1	*AFV6 #1	*AFV5 #1	*AFV4 #1	*AFV3 #1	*AFV2 #1	*AFV1 #1	*AFV0 #1
G014							ROV2 #1	ROV1 #1
G015								
G016	F1D #1							MSDFON #1
G017								
G018	HS2D #1	HS2C #1	HS2B #1	HS2A #1	HS1D #1	HS1C #1	HS1B #1	HS1A #1
G019	RT #1		MP2 #1	MP1 #1	HS3D #1	HS3C #1	HS3B #1	HS3A #1
G020								
G021								
G022								
G023	ALNGH #1	RGHTH #1						
G024								

	#7	#6	#5	#4	#3	#2	#1	#0
G025								
G026								
G027	CON[#1]		*SSTP3[#1]	*SSTP2[#1]	*SSTP1[#1]	SWS3[#1]	SWS2[#1]	SWS1[#1]
G028	PC2SLC[#1]	SPSTP[#1]	*SCPF[#1]	*SUCPF[#1]		GR2[#1]	GR1[#1]	
G029		*SSTP[#1]	SOR[#1]	SAR[#1]		GR31[#1]		GR21[#1]
G030	SOV7[#1]	SOV6[#1]	SOV5[#1]	SOV4[#1]	SOV3[#1]	SOV2[#1]	SOV1[#1]	SOV0[#1]
G031	PKESS2[#1]	PKESS1[#1]						
G032	R08I[#1]	R07I[#1]	R06I[#1]	R05I[#1]	R04I[#1]	R03I[#1]	R02I[#1]	R01I[#1]
G033	SIND[#1]	SSIN[#1]	SGN[#1]		R12I[#1]	R11I[#1]	R10I[#1]	R09I[#1]
G034	R08I2[#1]	R07I2[#1]	R06I2[#1]	R05I2[#1]	R04I2[#1]	R03I2[#1]	R02I2[#1]	R01I2[#1]
G035	SIND2[#1]	SSIN2[#1]	SGN2[#1]		R12I2[#1]	R11I2[#1]	R10I2[#1]	R09I2[#1]
G036	R08I3[#1]	R07I3[#1]	R06I3[#1]	R05I3[#1]	R04I3[#1]	R03I3[#1]	R02I3[#1]	R01I3[#1]
G037	SIND3[#1]	SSIN3[#1]	SGN3[#1]		R12I3[#1]	R11I3[#1]	R10I3[#1]	R09I3[#1]
G038	*BECLP[#1]	*BEUCP[#1]			SPPHS / SPPHS[#1]	SPSYC / SPSYC[#1]		*PLSST[#1]
G039	GOQSM[#1]	WOQSM[#1]	OFN5[#1]	OFN4[#1]	OFN3[#1]	OFN2[#1]	OFN1[#1]	OFN0[#1]
G040	WOSET[#1]	PRC[#1]						OFN6[#1]
G041	HS2ID[#1]	HS2IC[#1]	HS2IB[#1]	HS2IA[#1]	HS1ID[#1]	HS1IC[#1]	HS1IB[#1]	HS1IA[#1]
G042					HS3ID[#1]	HS3IC[#1]	HS3IB[#1]	HS3IA[#1]
G043	ZRN[#1]		DNCI[#1]			MD4[#1]	MD2[#1]	MD1[#1]
G044							MLK[#1]	BDT1[#1]
G045	BDT9[#1]	BDT8[#1]	BDT7[#1]	BDT6[#1]	BDT5[#1]	BDT4[#1]	BDT3[#1]	BDT2[#1]
G046	DRN[#1]	KEY4[#1]	KEY3[#1]	KEY2[#1]	KEY1[#1]		SBK[#1]	
G047	TL128[#1]	TL64[#1]	TL32[#1]	TL16[#1]	TL08[#1]	TL04[#1]	TL02[#1]	TL01[#1]
G048	TLRST[#1]	TLRSTI[#1]	TLSKP[#1]					TL256[#1]
G049	*TLV7[#1]	*TLV6[#1]	*TLV5[#1]	*TLV4[#1]	*TLV3[#1]	*TLV2[#1]	*TLV1[#1]	*TLV0[#1]

	#7	#6	#5	#4	#3	#2	#1	#0
G050							*TLV9 #1	*TLV8 #1
G051	*CHLD #1	CHPST #1			*CHP8 #1	*CHP4 #1	*CHP2 #1	*CHP0 #1
G052								
G053	CDZ #1	SMZ #1			UINT #1			TMRON#1
G054	UI007 #1	UI006 #1	UI005 #1	UI004 #1	UI003 #1	UI002 #1	UI001 #1	UI000 #1
G055	UI015 #1	UI014 #1	UI013 #1	UI012 #1	UI011 #1	UI010 #1	UI009 #1	UI008 #1
G056								
G057								
G058					EXWT #1	EXSTP #1	EXRD #1	MINP#1
G059							TRRTN #1	TRESC #1
G060	*TSB #1							
G061			RGTSP2#1	RGTSP1#1				RGTAP#1
G062			PDT2#1	PDT1#1			*CRTOF#1	
G063		INFD#1	NOZAGC#1		SLSPB	SLSPA	NOWT	HEAD
G064		ESRSYC#1			SLPCB	SLPCA	SLCSEQ#1	RTNCY #1
G065								
G066	EKSET			RTRCT			ENBKY	IGNVRY#1
G067								
G068								
G069								
G070	MRDYA#1	ORCMA#1	SFRA #1	SRVA #1	CTH1A#1	CTH2A#1	TLMHA#1	TLMLA#1
G071	RCHA #1	RSLA #1	INTGA #1	SOCNA#1	MCFNA#1	SPSLA#1	*ESPA#1	ARSTA#1
G072	RCHHGA#1	MFNHGA#1	INCMDA#1	OVRIDA#1	DEFMDA#1	NRROA#1	ROTAA#1	INDXA#1
G073						MPOFA#1	SLVA #1	MORCMA#1
G074	MRDYB#1	ORCMB#1	SFRB #1	SRVB #1	CTH1B#1	CTH2B#1	TLMHB#1	TLMLB#1

	#7	#6	#5	#4	#3	#2	#1	#0
G075	RCHB#1	RSLB #1	INTGB #1	SOCNB#1	MCFNB#1	SPSLB#1	*ESPB#1	ARSTB#1
G076	RCHHGB#1	MFNHGB#1	INCMDB#1	OVRIDB#1	DEFMDB#1	NRROB#1	ROTAB#1	INDXB#1
G077						MPOFB#1	SLVB#1	MORCMB#1
G078	SHA07#1	SHA06#1	SHA05#1	SHA04#1	SHA03#1	SHA02#1	SHA01#1	SHA00#1
G079					SHA11#1	SHA10#1	SHA09#1	SHA08#1
G080	SHB07#1	SHB06#1	SHB05#1	SHB04#1	SHB03#1	SHB02#1	SHB01#1	SHB00#1
G081					SHB11#1	SHB10#1	SHB09#1	SHB08#1
G082	Reserved for order made macro							
G083	Reserved for order made macro							
G084								
G085								
G086								
G087								
G088								
G089								
G090								
G091					SRLNI3#1	SRLNI2#1	SRLNI1#1	SRLNI0#1
G092				BGEN#1	BGIALM#1	BGION#1	IOLS#1	IOLACK#1
G093								
G094								
G095								
G096	HROV#1	*HROV6#1	*HROV5#1	*HROV4#1	*HROV3#1	*HROV2#1	*HROV1#1	*HROV0#1
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	#7	#6	#5	#4	#3	#2	#1	#0	
G100		+J7[#1]	+J6[#1]	+J5[#1]	+J4[#1]	+J3[#1]	+J2[#1]	+J1[#1]	
G101									
G102		–J7[#1]	–J6[#1]	–J5[#1]	–J4[#1]	–J3[#1]	–J2[#1]	–J1[#1]	
G103									
G104									
G105									
G106		MI7[#1]	MI6[#1]	MI5[#1]	MI4[#1]	MI3[#1]	MI2[#1]	MI1[#1]	
G107									
G108		MLK7[#1]	MLK6[#1]	MLK5[#1]	MLK4[#1]	MLK3[#1]	MLK2[#1]	MLK1[#1]	
G109									
G110		+LM7[#1]	+LM6[#1]	+LM5[#1]	+LM4[#1]	+LM3[#1]	+LM2[#1]	+LM1[#1]	
G111									
G112		–LM7[#1]	–LM6[#1]	–LM5[#1]	–LM4[#1]	–LM3[#1]	–LM2[#1]	–LM1[#1]	
G113									
G114		*+L7[#1]	*+L6[#1]	*+L5[#1]	*+L4[#1]	*+L3[#1]	*+L2[#1]	*+L1[#1]	
G115									
G116		*–L7[#1]	*–L6[#1]	*–L5[#1]	*–L4[#1]	*–L3[#1]	*–L2[#1]	*–L1[#1]	
G117									
G118		*+ED7[#1]	*+ED6[#1]	*+ED5[#1]	*+ED4[#1]	*+ED3[#1]	*+ED2[#1]	*+ED1[#1]	
G119									
G120		*–ED7[#1]	*–ED6[#1]	*–ED5[#1]	*–ED4[#1]	*–ED3[#1]	*–ED2[#1]	*–ED1[#1]	
G121									
G122	PKESS2[#1]	PK7[#1] / PKESS1[#1]	PK6[#1]	PK5[#1]	PK4[#1]	PK3[#1]	PK2[#1]	PK1[#1]	(T series)
	PKESS2[#1]	PKESS1[#1]							(M series)
G123									

	#7	#6	#5	#4	#3	#2	#1	#0
G124		DTCH7#1	DTCH6#1	DTCH5#1	DTCH4#1	DTCH3#1	DTCH2#1	DTCH1#1
G125								
G126		SVF7 #1	SVF6 #1	SVF5 #1	SVF4 #1	SVF3 #1	SVF2 #1	SVF1 #1
G127								
G128		MIX7	MIX6	MIX5	MIX4	MIX3	MIX2	MIX1
G129								
G130		*IT7 #1	*IT6 #1	*IT5 #1	*IT4 #1	*IT3 #1	*IT2 #1	*IT1 #1
G131								
G132					+MIT4 #1	+MIT3 #1	+MIT2 #1	+MIT1 #1
G133								
G134					–MIT4 #1	–MIT3 #1	–MIT2 #1	–MIT1 #1
G135								
G136		EAX7 #1	EAX6 #1	EAX5 #1	EAX4 #1	EAX3 #1	EAX2 #1	EAX1 #1
G137								
G138		SYNC7#1	SYNC6#1	SYNC5#1	SYNC4#1	SYNC3#1	SYNC2#1	SYNC1#1
G139								
G140		SYNCJ7#1	SYNCJ6#1	SYNCJ5#1	SYNCJ4#1	SYNCJ3#1	SYNCJ2#1	SYNCJ1#1
G141								
G142	EBUFA#1	ECLRA#1	ESTPA#1	ESOFA#1	ESBKA#1	EMBUFA#1		EFINA#1
G143	EMSBKA#1	EC6A #1	EC5A #1	EC4A #1	EC3A #1	EC2A #1	EC1A #1	EC0A #1
G144	EIF7A #1	EIF6A #1	EIF5A #1	EIF4A #1	EIF3A #1	EIF2A #1	EIF1A #1	EIF0A #1
G145	EIF15A#1	EIF14A#1	EIF13A#1	EIF12A#1	EIF11A#1	EIF10A#1	EIF9A #1	EIF8A #1
G146	EID7A#1	EID6A#1	EID5A#1	EID4A#1	EID3A#1	EID2A#1	EID1A#1	EID0A#1
G147	EID15A#1	EID14A#1	EID13A#1	EID12A#1	EID11A#1	EID10A#1	EID9A#1	EID8A#1
G148	EID23A#1	EID22A#1	EID21A#1	EID20A#1	EID19A#1	EID18A#1	EID17A#1	EID16A#1

	#7	#6	#5	#4	#3	#2	#1	#0
G149	EID31A[#1]	EID30A[#1]	EID29A[#1]	EID28A[#1]	EID27A[#1]	EID26A[#1]	EID25A[#1]	EID24A[#1]
G150	DRNE[#1]	RTE [#1]	OVCE[#1]				ROV2E[#1]	ROV1E[#1]
G151	*FV7E[#1]	*FV6E[#1]	*FV5E[#1]	*FV4E[#1]	*FV3E[#1]	*FV2E[#1]	*FV1E[#1]	*FV0E[#1]
G152								
G153								
G154	EBUFB[#1]	ECLRB[#1]	ESTPB[#1]	ESOFB[#1]	ESBKB[#1]	EMBUFB[#1]		EFINB[#1]
G155	EMSBKB[#1]	EC6B [#1]	EC5B [#1]	EC4B [#1]	EC3B [#1]	EC2B [#1]	EC1B [#1]	EC0B [#1]
G156	EIF7B[#1]	EIF6B[#1]	EIF5B[#1]	EIF4B[#1]	EIF3B[#1]	EIF2B[#1]	EIF1B[#1]	EIF0B[#1]
G157	EIF15B[#1]	EIF14B[#1]	EIF13B[#1]	EIF12B[#1]	EIF11B[#1]	EIF10B[#1]	EIF9B[#1]	EIF8B[#1]
G158	EID7B[#1]	EID6B[#1]	EID5B[#1]	EID4B[#1]	EID3B[#1]	EID2B[#1]	EID1B[#1]	EID0B[#1]
G159	EID15B[#1]	EID14B[#1]	EID13B[#1]	EID12B[#1]	EID11B[#1]	EID10B[#1]	EID9B[#1]	EID8B[#1]
G160	EID23B[#1]	EID22B[#1]	EID21B[#1]	EID20B[#1]	EID19B[#1]	EID18B[#1]	EID17B[#1]	EID16B[#1]
G161	EID31B[#1]	EID30B[#1]	EID29B[#1]	EID28B[#1]	EID27B[#1]	EID26B[#1]	EID25B[#1]	EID24B[#1]
G162								
G163								
G164								
G165								
G166	EBUFC[#1]	ECLRC[#1]	ESTPC[#1]	ESOFC[#1]	ESBKC[#1]	EMBUFC[#1]		EFINC[#1]
G167	EMSBKC[#1]	EC6C [#1]	EC5C [#1]	EC4C [#1]	EC3C [#1]	EC2C [#1]	EC1C [#1]	EC0C [#1]
G168	EIF7C[#1]	EIF6C[#1]	EIF5C[#1]	EIF4C[#1]	EIF3C[#1]	EIF2C[#1]	EIF1C[#1]	EIF0C[#1]
G169	EIF15C[#1]	EIF14C[#1]	EIF13C[#1]	EIF12C[#1]	EIF11C[#1]	EIF10C[#1]	EIF9C[#1]	EIF8C[#1]
G170	EID7C[#1]	EID6C[#1]	EID5C[#1]	EID4C[#1]	EID3C[#1]	EID2C[#1]	EID1C[#1]	EID0C[#1]
G171	EID15C[#1]	EID14C[#1]	EID13C[#1]	EID12C[#1]	EID11C[#1]	EID10C[#1]	EID9C[#1]	EID8C[#1]
G172	EID23C[#1]	EID22C[#1]	EID21C[#1]	EID20C[#1]	EID19C[#1]	EID18C[#1]	EID17C[#1]	EID16C[#1]
G173	EID31C[#1]	EID30C[#1]	EID29C[#1]	EID28C[#1]	EID27C[#1]	EID26C[#1]	EID25C[#1]	EID24C[#1]

	#7	#6	#5	#4	#3	#2	#1	#0
G174								
G175								
G176								
G177								
G178	EBUFD#1	ECLRD#1	ESTPD#1	ESOFD#1	ESBKD#1	EMBUFD#1		EFIND#1
G179	EMSBKD#1	EC6D #1	EC5D #1	EC4D #1	EC3D #1	EC2D #1	EC1D #1	EC0D #1
G180	EIF7D#1	EIF6D#1	EIF5D#1	EIF4D#1	EIF3D#1	EIF2D#1	EIF1D#1	EIF0D#1
G181	EIF15D#1	EIF14D#1	EIF13D#1	EIF12D#1	EIF11D#1	EIF10D#1	EIF9D#1	EIF8D#1
G182	EID7D#1	EID6D#1	EID5D#1	EID4D#1	EID3D#1	EID2D#1	EID1D#1	EID0D#1
G183	EID15D#1	EID14D#1	EID13D#1	EID12D#1	EID11D#1	EID10D#1	EID9D#1	EID8D#1
G184	EID23D#1	EID22D#1	EID21D#1	EID20D#1	EID19D#1	EID18D#1	EID17D#1	EID16D#1
G185	EID31D#1	EID30D#1	EID29D#1	EID28D#1	EID27D#1	EID26D#1	EID25D#1	EID24D#1
G186								
G187								
G188								
G189								
G190		OVLS7#1	OVLS6#1	OVLS5#1	OVLS4#1	OVLS3#1	OVLS2#1	OVLS1#1
G191								
G192		IGVRY7#1	IGVRY6#1	IGVRY5#1	IGVRY4#1	IGVRY3#1	IGVRY2#1	IGVRY1#1
G193								
G194								
G195								
G196								
G197								
G198		NPOS7#1	NPOS6#1	NPOS5#1	NPOS4#1	NPOS3#1	NPOS2#1	NPOS1#1

	#7	#6	#5	#4	#3	#2	#1	#0
G199								
G200								
G201								
G202								
G203								
G204								
G205								
G206								
G207								
G208								
G209								
G210								
G211								
G212								
G213								
G214								
G215								
G216								
G217								
G218								
G219								

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
G1000	ED7#2	ED6#2	ED5#2	ED4#2	ED3#2	ED2#2	ED1#2	ED0#2
G1001	ED15#2	ED14#2	ED13#2	ED12#2	ED11#2	ED10#2	ED9#2	ED8#2
G1002	ESTB#2	EA6#2	EA5#2	EA4#2	EA3#2	EA2#2	EA1#2	EA0#2
G1003								
G1004			MFIN3#2	MFIN2#2	FIN#2			
G1005	BFIN#2	AFL#2		BFIN#2	TFIN#2	SFIN#2	EFIN#2	MFIN#2
G1006		SKIPP#2		OVC#2		*ABSM#2		SRN#2
G1007	RLSOT#2	EXLM#2	*FLWP#2	RLSOT3#2		ST#2	STLK#2	RVS#2
G1008	ERS#2	RRW#2	*SP#2	*ESP#2				*IT#2
G1009				PN16#2	PN8#2	PN4#2	PN2#2	PN1#2
G1010	*JV7#2	*JV6#2	*JV5#2	*JV4#2	*JV3#2	*JV2#2	*JV1#2	*JV0#2
G1011	*JV15#2	*JV14#2	*JV13#2	*JV12#2	*JV11#2	*JV10#2	*JV9#2	*JV8#2
G1012	*FV7#2	*FV6#2	*FV5#2	*FV4#2	*FV3#2	*FV2#2	*FV1#2	*FV0#2
G1013	*AFV7#2	*AFV6#2	*AFV5#2	*AFV4#2	*AFV3#2	*AFV2#2	*AFV1#2	*AFV0#2
G1014							ROV2#2	ROV1#2
G1015								
G1016	F1D#2							MSDFON#2
G1017								
G1018	HS2D#2	HS2C#2	HS2B#2	HS2A#2	HS1D#2	HS1C#2	HS1B#2	HS1A#2
G1019	RT#2		MP2#2	MP1#2	HS3D#2	HS3C#2	HS3B#2	HS3A#2
G1020								
G1021								
G1022								
G1023	ALNGH#2	RGHTH#2						
G1024								

	#7	#6	#5	#4	#3	#2	#1	#0
G1025								
G1026								
G1027	CON#2		*SSTP3#2	*SSTP2#2	*SSTP1#2	SWS3#2	SWS2#2	SWS1#2
G1028	PC2SLC#2	SPSTP#2	*SCPF#2	*SUCPF#2		GR2#2	GR1#2	
G1029		*SSTP#2	SOR#2	SAR#2		GR31#2		GR21#2
G1030	SOV7#2	SOV6#2	SOV5#2	SOV4#2	SOV3#2	SOV2#2	SOV1#2	SOV0#2
G1031	PKESS2#2	PKESS1#2						
G1032	R08I#2	R07I#2	R06I#2	R05I#2	R04I#2	R03I#2	R02I#2	R01I#2
GI033	SIND#2	SSIN#2	SGN#2		R12I#2	R11I#2	R10I#2	R09I#2
G1034	R08I2#2	R07I2#2	R06I2#2	R05I2#2	R04I2#2	R03I2#2	R02I2#2	R01I2#2
G1035	SIND2#2	SSIN2#2	SGN2#2		R12I2#2	R11I2#2	R10I2#2	R09I2#2
G1036	R08I3#2	R07I3#2	R06I3#2	R05I3#2	R04I3#2	R03I3#2	R02I3#2	R01I3#2
G1037	SIND3#2	SSIN3#2	SGN3#2		R12I3#2	R11I3#2	R10I3#2	R09I3#2
G1038	*BECLP #2	*BEUCP #2			SPPHS#2	SPSYC#2		*PLSST#2
G1039	GOQSM#2	WOQSM#2	OFN5#2	OFN4#2	OFN3#2	OFN2#2	OFN1#2	OFN0#2
G1040	WOSET#2	PRC#2						OFN6#2
G1041	HS2ID#2	HS2IC#2	HS2IB#2	HS2IA#2	HS1ID#2	HS1IC#2	HS1IB#2	HS1IA#2
G1042					HS3ID#2	HS3IC#2	HS3IB#2	HS3IA#2
G1043	ZRN#2		DNCI#2			MD4#2	MD2#2	MD1#2
G1044							MLK#2	BDT1#2
G1045	BDT9#2	BDT8#2	BDT7#2	BDT6#2	BDT5#2	BDT4#2	BDT3#2	BDT2#2
G1046	DRN#2	KEY4#2	KEY3#2	KEY2#2	KEY1#2		SBK#2	
G1047	TL128#2	TL64#2	TL32#2	TL16#2	TL08#2	TL04#2	TL02#2	TL01#2
G1048	TLRST#2	TLRSTI#2	TLSKP#2					TL256#2
G1049	*TLV7 #2	*TLV6 #2	*TLV5 #2	*TLV4#2	*TLV3 #2	*TLV2 #2	*TLV1 #2	*TLV0 #2

	#7	#6	#5	#4	#3	#2	#1	#0
G1050							*TLV9 #2	*TLV8 #2
G1051	*CHLD #2	CHPST #2			*CHP8 #2	*CHP4 #2	*CHP2 #2	*CHP0 #2
G1052								
G1053	CDZ#2	SMZ#2			UINT#2			TMRON#2
G1054	UI007#2	UI006#2	UI005#2	UI004#2	UI003#2	UI002#2	UI001#2	UI000#2
G1055	UI015#2	UI014#2	UI013#2	UI012#2	UI011#2	UI010#2	UI009#2	UI008#2
G1056								
G1057								
G1058					EXWT#2	EXSTP#2	EXRD#2	MINP#2
G1059							TRRTN#2	TRESC#2
G1060	*TSB#2							
G1061			RGTSP2#2	RGTSP1#2				RGTAP#2
G1062			PDT2#2	PDT2#1			*CRTOF#2	
G1063		INFD#2	NOZAGC#2					
G1064		ESRSYC#2					SLCSEQ #2	RTNCY #2
G1065								
G1066				RTRCT#2				IGNVRY#2
G1067								
G1068								
G1069								
G1070	MRDYA#2	ORCMA#2	SFRA#2	SRVA#2	CTH1A#2	CTH2A#2	TLMHA#2	TLMLA#2
G1071	RCHA#2	RSLA#2	INTGA #2	SOCNA#2	MCFNA#2	SPSLA#2	*ESPA#2	ARSTA#2
G1072	RCHHGA#2	MFNHGA#2	INCMDA#2	OVRIDA#2	DEFMDA#2	NRROA#2	ROTAA#2	INDXA#2
G1073						MPOFA#2	SLVA #2	MORCMA#2
G1074	MRDYB#2	ORCMB#2	SFRB#2	SRVB#2	CTH1B#2	CTH2B#2	TLMHB#2	TLMLB#2

	#7	#6	#5	#4	#3	#2	#1	#0
G1075	RCHB#2	RSLB#2	INTGB #2	SOCNB#2	MCFNB#2	SPSLB#2	*ESPB#2	ARSTB#2
G1076	RCHHGB#2	MFNHGB#2	INCMDB#2	OVRIDB#2	DEFMDB#2	NRROB#2	ROTAB#2	INDXB#2
G1077						MPOFB#2	SLVB#2	MORCMB#2
G1078	SHA07#2	SHA06#2	SHA05#2	SHA04#2	SHA03#2	SHA02#2	SHA01#2	SHA00#2
G1079					SHA11#2	SHA10#2	SHA09#2	SHA08#2
G1080	SHB07#2	SHB06#2	SHB05#2	SHB04#2	SHB03#2	SHB02#2	SHB01#2	SHB00#2
G1081					SHB11#2	SHB10#2	SHB09#2	SHB08#2
G1082	Reserved for order made macro							
G1083	Reserved for order made macro							
G1084								
G1085								
G1086								
G1087								
G1088								
G1089								
G1090								
G1091					SRLNI3#2	SRLNI2#2	SRLNI1#2	SRLNI0#2
G1092				BGEN#2	BGIALM#2	BGION#2	IOLS#2	IOLACK#2
G1093								
G1094								
G1095								
G1096	HROV#2	*HROV6#2	*HROV5#2	*HROV4#2	*HROV3#2	*HROV2#2	*HROV1#2	*HROV0#2
G1097								
G1098								
G1099								

	#7	#6	#5	#4	#3	#2	#1	#0	
G1100		+J7#2	+J6#2	+J5#2	+J4#2	+J3#2	+J2#2	+J1#2	
G1101									
G1102		–J7#2	–J6#2	–J5#2	–J4#2	–J3#2	–J2#2	–J1#2	
G1103									
G1104									
G1105									
G1106		MI7#2	MI6#2	MI5#2	MI4#2	MI3#2	MI2#2	MI1#2	
G1107									
G1108		MLK7#2	MLK6#2	MLK5#2	MLK4#2	MLK3#2	MLK2#2	MLK1#2	
G1109									
G1110		+LM7 #2	+LM6 #2	+LM5 #2	+LM4 #2	+LM3 #2	+LM2 #2	+LM1 #2	
G1111									
G1112		–LM7 #2	–LM6 #2	–LM5 #2	–LM4 #2	–LM3 #2	–LM2 #2	–LM1 #2	
G1113									
G1114		*+L7#2	*+L6#2	*+L5#2	*+L4#2	*+L3#2	*+L2#2	*+L1#2	
G1115									
G1116		*–L7#2	*–L6#2	*–L5#2	*–L4#2	*–L3#2	*–L2#2	*–L1#2	
G1117									
G1118		*+ED7#2	*+ED6#2	*+ED5#2	*+ED4#2	*+ED3#2	*+ED2#2	*+ED1#2	
G1119									
G1120		*–ED7#2	*–ED6#2	*–ED5#2	*–ED4#2	*–ED3#2	*–ED2#2	*–ED1#2	
G1121									
G1122	PKESS2#2	PK7#2 / PKESS1#2	PK6 #2	PK5 #2	PK4 #2	PK3 #2	PK2 #2	PK1 #2	(T series)
	PKESS2#2	PKESS1#2							(M series)
G1123									

	#7	#6	#5	#4	#3	#2	#1	#0
G1124		DTCH7#2	DTCH6#2	DTCH5#2	DTCH4#2	DTCH3#2	DTCH2#2	DTCH1#2
G1125								
G1126		SVF7#2	SVF6#2	SVF5#2	SVF4#2	SVF3#2	SVF2#2	SVF1#2
G1127								
G1128								
G1129								
G1130		*IT7#2	*IT6#2	*IT5#2	*IT4#2	*IT3#2	*IT2#2	*IT1#2
G1131								
G1132					+MIT4 #2	+MIT3#2	+MIT2 #2	+MIT1 #2
G1133								
G1134					–MIT4 #2	–MIT3 #2	–MIT2 #2	–MIT1 #2
G1135								
G1136		EAX7#2	EAX6#2	EAX5#2	EAX4#2	EAX3#2	EAX2#2	EAX1#2
G1137								
G1138		SYNC7#2	SYNC6#2	SYNC5#2	SYNC4#2	SYNC3#2	SYNC2#2	SYNC1#2
G1139								
G1140		SYNCJ7#2	SYNCJ6#2	SYNCJ5#2	SYNCJ4#2	SYNCJ3#2	SYNCJ2#2	SYNCJ1#2
G1141								
G1142	EBUFA#2	ECLRA#2	ESTPA#2	ESOFA#2	ESBKA#2	EMBUFA#2		EFINA#2
G1143	EMSBKA#2	EC6A#2	EC5A#2	EC4A#2	EC3A#2	EC2A#2	EC1A#2	EC0A#2
G1144	EIF7A#2	EIF6A#2	EIF5A#2	EIF4A#2	EIF3A#2	EIF2A#2	EIF1A#2	EIF0A#2
G1145	EIF15A#2	EIF14A#2	EIF13A#2	EIF12A#2	EIF11A#2	EIF10A#2	EIF9A#2	EIF8A#2
G1146	EID7A#2	EID6A#2	EID5A#2	EID4A#2	EID3A#2	EID2A#2	EID1A#2	EID0A#2
G1147	EID15A#2	EID14A#2	EID13A#2	EID12A#2	EID11A#2	EID10A#2	EID9A#2	EID8A#2
G1148	EID23A#2	EID22A#2	EID21A#2	EID20A#2	EID19A#2	EID18A#2	EID17A#2	EID16A#2

	#7	#6	#5	#4	#3	#2	#1	#0
G1149	EID31A#2	EID30A#2	EID29A#2	EID28A#2	EID27A#2	EID26A#2	EID25A#2	EID24A#2
G1150	DRNE#2	RTE#2	OVCE#2				ROV2E#2	ROV1E#2
G1151	*FV7E#2	*FV6E#2	*FV5E#2	*FV4E#2	*FV3E#2	*FV2E#2	*FV1E#2	*FV0E#2
G1152								
G1153								
G1154	EBUFB#2	ECLRB#2	ESTPB#2	ESOFB#2	ESBKB#2	EMBUFB#2		EFINB#2
G1155	EMSBKB#2	EC6B#2	EC5B#2	EC4B#2	EC3B#2	EC2B#2	EC1B#2	EC0B#2
G1156	EIF7B#2	EIF6B#2	EIF5B#2	EIF4B#2	EIF3B#2	EIF2B#2	EIF1B#2	EIF0B#2
G1157	EIF15B#2	EIF14B#2	EIF13B#2	EIF12B#2	EIF11B#2	EIF10B#2	EIF9B#2	EIF8B#2
G1158	EID7B#2	EID6B#2	EID5B#2	EID4B#2	EID3B#2	EID2B#2	EID1B#2	EID0B#2
G1159	EID15B#2	EID14B#2	EID13B#2	EID12B#2	EID11B#2	EID10B#2	EID9B#2	EID8B#2
G1160	EID23B#2	EID22B#2	EID21B#2	EID20B#2	EID19B#2	EID18B#2	EID17B#2	EID16B#2
G1161	EID31B#2	EID30B#2	EID29B#2	EID28B#2	EID27B#2	EID26B#2	EID25B#2	EID24B#2
G1162								
G1163								
G1164								
G1165								
G1166	EBUFC#2	ECLRC#2	ESTPC#2	ESOFC#2	ESBKC#2	EMBUFC#2		EFINC#2
G1167	EMSBKC#2	EC6C#2	EC5C#2	EC4C#2	EC3C#2	EC2C#2	EC1C#2	EC0C#2
G1168	EIF7C#2	EIF6C#2	EIF5C#2	EIF4C#2	EIF3C#2	EIF2C#2	EIF1C#2	EIF0C#2
G1169	EIF15C#2	EIF14C#2	EIF13C#2	EIF12C#2	EIF11C#2	EIF10C#2	EIF9C#2	EIF8C#2
G1170	EID7C#2	EID6C#2	EID5C#2	EID4C#2	EID3C#2	EID2C#2	EID1C#2	EID0C#2
G1171	EID15C#2	EID14C#2	EID13C#2	EID12C#2	EID11C#2	EID10C#2	EID9C#2	EID8C#2
G1172	EID23C#2	EID22C#2	EID21C#2	EID20C#2	EID19C#2	EID18C#2	EID17C#2	EID16C#2
G1173	EID31C#2	EID30C#2	EID29C#2	EID28C#2	EID27C#2	EID26C#2	EID25C#2	EID24C#2

	#7	#6	#5	#4	#3	#2	#1	#0
G1174								
G1175								
G1176								
G1177								
G1178	EBUFD#2	ECLRD#2	ESTPD#2	ESOFD#2	ESBKD#2	EMBUFD#2		EFIND#2
G1179	EMSBKD#2	EC6D#2	EC5D#2	EC4D#2	EC3D#2	EC2D#2	EC1D#2	EC0D#2
G1180	EIF7D#2	EIF6D#2	EIF5D#2	EIF4D#2	EIF3D#2	EIF2D#2	EIF1D#2	EIF0D#2
G1181	EIF15D#2	EIF14D#2	EIF13D#2	EIF12D#2	EIF11D#2	EIF10D#2	EIF9D#2	EIF8D#2
G1182	EID7D#2	EID6D#2	EID5D#2	EID4D#2	EID3D#2	EID2D#2	EID1D#2	EID0D#2
G1183	EID15D#2	EID14D#2	EID13D#2	EID12D#2	EID11D#2	EID10D#2	EID9D#2	EID8D#2
G1184	EID23D#2	EID22D#2	EID21D#2	EID20D#2	EID19D#2	EID18D#2	EID17D#2	EID16D#2
G1185	EID31D#2	EID30D#2	EID29D#2	EID28D#2	EID27D#2	EID26D#2	EID25D#2	EID24D#2
G1186								
G1187								
G1188								
G1189								
G1190		OVLS7#2	OVLS6#2	OVLS5#2	OVLS4#2	OVLS3#2	OVLS2#2	OVLS1#2
G1191								
G1192		IGVRY7#2	IGVRY6#2	IGVRY5#2	IGVRY4#2	IGVRY3#2	IGVRY2#2	IGVRY1#2
G1193								
G1194								
G1195								
G1196								
G1197								
G1198		NPOS7#2	NPOS6#2	NPOS5#2	NPOS4#2	NPOS3#2	NPOS2#2	NPOS1#2

	#7	#6	#5	#4	#3	#2	#1	#0
G1199								
G1200								
G1201								
G1202								
G1203								
G1204								
G1205								
G1206								
G1207								
G1208								
G1209								
G1210								
G1211								
G1212								
G1213								
G1214								
G1215								
G1216								
G1217								
G1218								
G1219								

CNC → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP[#1]	SA[#1]	STL[#1]	SPL[#1]				RWD[#1]
F001	MA[#1]		TAP[#1]	ENB[#1]	DEN[#1]	BAL[#1]	RST[#1]	AL[#1]
F002	MDRN[#1]	CUT[#1]		SRNMV[#1]	THRD[#1]	CSS[#1]	RPDO[#1]	INCH[#1]
F003	MTCHIN[#1]	MEDT[#1]	MMEM[#1]	MRMT[#1]	MMDI[#1]	MJ[#1]	MH[#1]	MINC[#1]
F004			MREF[#1]	MAFL[#1]	MSBK[#1]	MABSM[#1]	MMLK[#1]	MBDT1[#1]
F005	MBDT9[#1]	MBDT8[#1]	MBDT7[#1]	MBDT6[#1]	MBDT5[#1]	MBDT4[#1]	MBDT3[#1]	MBDT2[#1]
F006								
F007	BF[#1]			BF[#1]	TF[#1]	SF[#1]	EFD[#1]	MF[#1]
F008			MF3[#1]	MF2[#1]				EF[#1]
F009	DM00[#1]	DM01[#1]	DM02[#1]	DM30[#1]				
F010	M07[#1]	M06[#1]	M05[#1]	M04[#1]	M03[#1]	M02[#1]	M01[#1]	M00[#1]
F011	M15[#1]	M14[#1]	M13[#1]	M12[#1]	M11[#1]	M10[#1]	M09[#1]	M08[#1]
F012	M23[#1]	M22[#1]	M21[#1]	M20[#1]	M19[#1]	M18[#1]	M17[#1]	M16[#1]
F013	M31[#1]	M30[#1]	M29[#1]	M28[#1]	M27[#1]	M26[#1]	M25[#1]	M24[#1]
F014	M207[#1]	M206[#1]	M205[#1]	M204[#1]	M203[#1]	M202[#1]	M201[#1]	M200[#1]
F015	M215[#1]	M214[#1]	M213[#1]	M212[#1]	M211[#1]	M210[#1]	M209[#1]	M208[#1]
F016	M307[#1]	M306[#1]	M305[#1]	M304[#1]	M303[#1]	M302[#1]	M301[#1]	M300[#1]
F017	M315[#1]	M314[#1]	M313[#1]	M312[#1]	M311[#1]	M310[#1]	M309[#1]	M308[#1]
F018								
F019								
F020								
F021								
F022	S07[#1]	S06[#1]	S05[#1]	S04[#1]	S03[#1]	S02[#1]	S01[#1]	S00[#1]
F023	S15[#1]	S14[#1]	S13[#1]	S12[#1]	S11[#1]	S10[#1]	S09[#1]	S08[#1]
F024	S23[#1]	S22[#1]	S21[#1]	S20[#1]	S19[#1]	S18[#1]	S17[#1]	S16[#1]

	#7	#6	#5	#4	#3	#2	#1	#0
F025	S31#1	S30#1	S29#1	S28#1	S27#1	S26#1	S25#1	S24#1
F026	T07#1	T06#1	T05#1	T04#1	T03#1	T02#1	T01#1	T00#1
F027	T15#1	T14#1	T13#1	T12#1	T11#1	T10#1	T09#1	T08#1
F028	T23#1	T22#1	T21#1	T20#1	T19#1	T18#1	T17#1	T16#1
F029	T31#1	T30#1	T29#1	T28#1	T27#1	T26#1	T25#1	T24#1
F030	B07#1	B06#1	B05#1	B04#1	B03#1	B02#1	B01#1	B00#1
F031	B15#1	B14#1	B13#1	B12#1	B11#1	B10#1	B09#1	B08#1
F032	B23#1	B22#1	B21#1	B20#1	B19#1	B18#1	B17#1	B16#1
F033	B31#1	B30#1	B29#1	B28#1	B27#1	B26#1	B25#1	B24#1
F034						GR3O#1	GR2O#1	GR1O#1
F035								SPAL#1
F036	R08O#1	R07O#1	R06O#1	R05O#1	R04O#1	R03O#1	R02O#1	R01O#1
F037					R12O#1	R11O#1	R10O#1	R09O#1
F038					ENB3#1	ENB2#1	SUCLP#1	SCLP#1
F039					CHPCYL#1	CHPMD#1		
F040	AR7#1	AR6#1	AR5#1	AR4#1	AR3#1	AR2#1	AR1#1	AR0#1
F041	AR15#1	AR14#1	AR13#1	AR12#1	AR11#1	AR10#1	AR09#1	AR08#1
F042								
F043								
F044				SYCAL SYCAL #1	FSPPH FSPPH #1	FSPSY FSPSY #1	FSCSL#1	
F045	ORARA#1	TLMA#1	LDT2A#1	LDT1A#1	SARA#1	SDTA#1	SSTA#1	ALMA#1
F046	MORA2A#1	MORA1A#1	PORA2A#1	SLVSA#1	RCFNA#1	RCHPA#1	CFINA#1	CHPA#1
F047							INCSTA#1	PC1DEA#1
F048								
F049	ORARB#1	TLMB#1	LDT2B#1	LDT1B#1	SARB#1	SDTB#1	SSTB#1	ALMB#1

	#7	#6	#5	#4	#3	#2	#1	#0
F050	MORA2B[#1]	MORA1B[#1]	PORA2B[#1]	SLVSB[#1]	RCFNB[#1]	RCHPB[#1]	CFINB[#1]	CHPB[#1]
F051							INCSTB[#1]	PC1DEB[#1]
F052								
F053	EKENB			BGEACT[#1]	RPALM[#1]	RPBSY[#1]	PRGDPL	INHKY
F054	UO007[#1]	UO006[#1]	UO005[#1]	UO004[#1]	UO003[#1]	UO002[#1]	UO001[#1]	UO000[#1]
F055	UO015[#1]	UO014[#1]	UO013[#1]	UO012[#1]	UO011[#1]	UO010[#1]	UO009[#1]	UO008[#1]
F056	UO107[#1]	UO106[#1]	UO105[#1]	UO104[#1]	UO103[#1]	UO102[#1]	UO101[#1]	UO100[#1]
F057	UO115[#1]	UO114[#1]	UO11¿[#1]	UO112[#1]	UO111[#1]	UO110[#1]	UO109[#1]	UO108[#1]
F058	UO123[#1]	UO122[#1]	UO121[#1]	UO120[#1]	UO119[#1]	UO118[#1]	UO117[#1]	UO116[#1]
F059	UO131[#1]	UO130[#1]	UO129[#1]	UO128[#1]	UO127[#1]	UO126[#1]	UO125[#1]	UO124[#1]
F060							ESEND[#1]	EREND[#1]
F061							BCLP[#1]	BUCLP[#1]
F062	PRTSF[#1]							
F063	PSYN[#1]	WATO[#1]	RCYO[#1]			PSAR[#1]	PSE2[#1]	PSE1[#1]
F064	TIALM	TICHK	COSP			TLCHI[#1]	TLNW[#1]	TLCH[#1]
F065		SYNMOD[#1]		RTRCTF[#1]			RGSPM[#1]	RGSPP[#1]
F066			PECK2[#1]		RTNMVS[#1]			G08MD[#1]
F067								
F068								
F069								
F070	PSW08[#1]	PSW07[#1]	PSW06[#1]	PSW05[#1]	PSW04[#1]	PSW03[#1]	PSW02[#1]	PSW01[#1]
F071							PSW10[#1]	PSW09[#1]
F072	OUT7[#1]	OUT6[#1]	OUT5[#1]	OUT4[#1]	OUT3[#1]	OUT2[#1]	OUT1[#1]	OUT0[#1]
F073				ZRNO[#1]		MD4O[#1]	MD2O[#1]	MD1O[#1]
F074								

	#7	#6	#5	#4	#3	#2	#1	#0
F075	SPO[#1]	KEYO[#1]	DRNO[#1]	MLKO[#1]	SBKO[#1]	BDTO[#1]		
F076			ROV2O[#1]	ROV1O[#1]	RTAP[#1]		MP2O[#1]	MP1O[#1]
F077		RTO[#1]			HS1DO[#1]	HS1CO[#1]	HS1BO[#1]	HS1AO[#1]
F078	*FV7O[#1]	*FV6O[#1]	*FV5O[#1]	*FV4O[#1]	*FV3O[#1]	*FV2O[#1]	*FV1O[#1]	*FV0O[#1]
F079	*JV7O[#1]	*JV6O[#1]	*JV5O[#1]	*JV4O[#1]	*JV3O[#1]	*JV2O[#1]	*JV1O[#1]	*JV0O[#1]
F080	*FV15O[#1]	*FV14O[#1]	*FV13O[#1]	*FV12O[#1]	*FV11O[#1]	*FV10O[#1]	*FV9O[#1]	*FV8O[#1]
F081	–J4O[#1]	+J4O[#1]	–J3O[#1]	+J3O[#1]	–J2O[#1]	+J2O[#1]	–J1O[#1]	+J1O[#1]
F082						RVSL[#1]		
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090						ABTSP2[#1]	ABTSP1[#1]	ABTQSV[#1]
F091								
F092			TRSPS[#1]		TRACT[#1]			
F093								
F094		ZP7[#1]	ZP6[#1]	ZP5[#1]	ZP4[#1]	ZP3[#1]	ZP2[#1]	ZP1[#1]
F095								
F096		ZP27[#1]	ZP26[#1]	ZP25[#1]	ZP24[#1]	ZP23[#1]	ZP22[#1]	ZP21[#1]
F097								
F098		ZP37[#1]	ZP36[#1]	ZP35[#1]	ZP34[#1]	ZP33[#1]	ZP32[#1]	ZP31[#1]
F099								

	#7	#6	#5	#4	#3	#2	#1	#0
F100		ZP47#1	ZP46#1	ZP45#1	ZP44#1	ZP43#1	ZP42#1	ZP41#1
F101								
F102		MV7#1	MV6#1	MV5#1	MV4#1	MV3#1	MV2#1	MV1#1
F103								
F104		INP7#1	INP6#1	INP5#1	INP4#1	INP3#1	INP2#1	INP1#1
F105								
F106		MVD7#1	MVD6#1	MVD5#1	MVD4#1	MVD3#1	MVD2#1	MVD1#1
F107								
F108		MMI7#1	MMI6#1	MMI5#1	MMI4#1	MMI3#1	MMI2#1	MMI1#1
F109								
F110		MDTCH7#1	MDTCH6#1	MDTCH5#1	MDTCH4#1	MDTCH3#1	MDTCH2#1	MDTCH1#1
F111								
F112		EADEN7#1	EADEN6#1	EADEN5#1	EADEN4#1	EADEN3#1	EADEN2#1	EADEN1#1
F113								
F114		TRQL7#1	TRQL6#1	TRQL5#1	TRQL4#1	TRQL3#1	TRQL2#1	TRQL1#1
F115								
F116		FRP7#1	FRP6#1	FRP5#1	FRP4#1	FRP3#1	FRP2#1	FRP1#1
F117								
F118		SYN7O#1	SYN6O#1	SYN5O#1	SYN4O#1	SYN3O#1	SYN2O#1	SYN1O#1
F119								
F120		ZRF7#1	ZRF6#1	ZRF5#1	ZRF4#1	ZRF3#1	ZRF2#1	ZRF1#1
F121								
F122	HDO7#1	HDO6#1	HDO5#1	HDO4#1	HDO3#1	HDO2#1	HDO1#1	HDO0#1
F123								
F124								

	#7	#6	#5	#4	#3	#2	#1	#0
F125								
F126								
F127								
F128								
F129	*EAXSL#1		EOV0#1					
F130	EBSYA#1	EOTNA#1	EOTPA#1	EGENA#1	EDENA#1	EIALA#1	ECKZA#1	EINPA#1
F131							EABUFA#1	EMFA#1
F132	EM28A#1	EM24A#1	EM22A#1	EM21A#1	EM18A#1	EM14A#1	EM12A#1	EM11A#1
F133	EBSYB#1	EOTNB#1	EOTPB#1	EGENB#1	EDENB#1	EIALB#1	ECKZB#1	EINPB#1
F134							EABUFB#1	EMFB#1
F135	EM28B#1	EM24B#1	EM22B#1	EM21B#1	EM18B#1	EM14B#1	EM12B#1	EM11B#1
F136	EBSYC#1	EOTNC#1	EOTPC#1	EGENC#1	EDENC#1	EIALC#1	ECKZC#1	EINPC#1
F137							EABUFC#1	EMFC#1
F138	EM28C#1	EM24C#1	EM22C#1	EM21C#1	EM18C#1	EM14C#1	EM12C#1	EM11C#1
F139	EBSYD#1	EOTND#1	EOTPD#1	EGEND#1	EDEND#1	EIALD#1	ECKZD#1	EINPD#1
F140							EABUFD#1	EMFD#1
F141	EM28D#1	EM24D#1	EM22D#1	EM21D#1	EM18D#1	EM14D#1	EM12D#1	EM11D#1
F142	EM48A#1	EM44A#1	EM42A#1	EM41A#1	EM38A#1	EM34A#1	EM32A#1	EM31A#1
F143								
F144								
F145	EM48B#1	EM44B#1	EM42B#1	EM41B#1	EM38B#1	EM34B#1	EM32B#1	EM31B#1
F146								
F147								
F148	EM48C#1	EM44C#1	EM42C#1	EM41C#1	EM38C#1	EM34C#1	EM32C#1	EM31C#1
F149								

	#7	#6	#5	#4	#3	#2	#1	#0
F150								
F151	EM48D#1	EM44D#1	EM42D#1	EM41D#1	EM38D#1	EM34D#1	EM32D#1	EM31D#1
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168								
F169								
F170								
F171								
F172								
F173								
F174								

	#7	#6	#5	#4	#3	#2	#1	#0
F175								
F176								
F177	EDGN#1	EPARM#1	EVAR#1	EPRG#1	EWTIO#1	ESTPIO#1	ERDIO#1	IOLNK#1
F178					SRLNO3#1	SRLNO2#1	SRLNO1#1	SRLNO0#1
F179								
F180	CLRCH8#1	CLRCH7#1	CLRCH6#1	CLRCH5#1	CLRCH4#1	CLRCH3#1	CLRCH2#1	CLRCH1#1
F181								
F182		EACNT7#1	EACNT6#1	EACNT5#1	EACNT4#1	EACNT3#1	EACNT2#1	EACNT1#1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
F1000	OP[#2]	SA[#2]	STL[#2]	SPL[#2]				RWD[#2]
F1001	MA[#2]		TAP[#2]	ENB[#2]	DEN[#2]	BAL[#2]	RST[#2]	AL[#2]
F1002	MDRN[#2]	CUT[#2]		SRNMV[#2]	THRD[#2]	CSS[#2]	RPDO[#2]	INCH[#2]
F1003	MTCHIN[#2]	MEDT[#2]	MMEM[#2]	MRMT[#2]	MMDI[#2]	MJ[#2]	MH[#2]	MINC[#2]
F1004			MREF[#2]	MAFL[#2]	MSBK[#2]	MABSM[#1]	MMLK[#2]	MBDT1[#2]
F1005	MBDT9[#2]	MBDT8[#2]	MBDT7[#2]	MBDT6[#2]	MBDT5[#2]	MBDT4[#2]	MBDT3[#2]	MBDT2[#2]
F1006								
F1007	BF[#2]			BF[#2]	TF[#2]	SF[#2]	EFD[#2]	MF[#2]
F1008			MF3[#2]	MF2[#2]				EF[#2]
F1009	DM00[#2]	DM01[#2]	DM02[#2]	DM30[#2]				
F010	M07[#2]	M06[#2]	M05[#2]	M04[#2]	M03[#2]	M02[#2]	M01[#2]	M00[#2]
F1011	M15[#2]	M14[#2]	M13[#2]	M12[#2]	M11[#2]	M10[#2]	M09[#2]	M08[#2]
F1012	M23[#2]	M22[#2]	M21[#2]	M20[#2]	M19[#2]	M18[#2]	M17[#2]	M16[#2]
F1013	M31[#2]	M30[#2]	M29[#2]	M28[#2]	M27[#2]	M26[#2]	M25[#2]	M24[#2]
F1014	M207[#2]	M206[#2]	M205[#2]	M204[#2]	M203[#2]	M202[#2]	M201[#2]	M200[#2]
F1015	M215[#2]	M214[#2]	M213[#2]	M212[#2]	M211[#2]	M210[#2]	M209[#2]	M208[#2]
F1016	M307[#2]	M306[#2]	M305[#2]	M304[#2]	M303[#2]	M302[#2]	M301[#2]	M300[#2]
F1017	M315[#2]	M314[#2]	M313[#2]	M312[#2]	M311[#2]	M310[#2]	M309[#2]	M308[#2]
F1018								
F1019								
F1020								
F1021								
F1022	S07[#2]	S06[#2]	S05[#2]	S04[#2]	S03[#2]	S02[#2]	S01[#2]	S00[#2]
F1023	S15[#2]	S14[#2]	S13[#2]	S12[#2]	S11[#2]	S10[#2]	S09[#2]	S08[#2]
F1024	S23[#2]	S22[#2]	S21[#2]	S20[#2]	S19[#2]	S18[#2]	S17[#2]	S16[#2]

	#7	#6	#5	#4	#3	#2	#1	#0
F1025	S31[#2]	S30[#2]	S29[#2]	S28[#2]	S27[#2]	S26[#2]	S25[#2]	S24[#2]
F1026	T07[#2]	T06[#2]	T05[#2]	T04[#2]	T03[#2]	T02[#2]	T01[#2]	T00[#2]
F1027	T15[#2]	T14[#2]	T13[#2]	T12[#2]	T11[#2]	T10[#2]	T09[#2]	T08[#2]
F1028	T23[#2]	T22[#2]	T21[#2]	T20[#2]	T19[#2]	T18[#2]	T17[#2]	T16[#2]
F1029	T31[#2]	T30[#2]	T29[#2]	T28[#2]	T27[#2]	T26[#2]	T25[#2]	T24[#2]
F1030	B07[#2]	B06[#2]	B05[#2]	B04[#2]	B03[#2]	B02[#2]	B01[#2]	B00[#2]
F1031	B15[#2]	B14[#2]	B13[#2]	B12[#2]	B11[#2]	B10[#2]	B09[#2]	B08[#2]
F1032	B23[#2]	B22[#2]	B21[#2]	B20[#2]	B19[#2]	B18[#2]	B17[#2]	B16[#2]
F1033	B31[#2]	B30[#2]	B29[#2]	B28[#2]	B27[#2]	B26[#2]	B25[#2]	B24[#2]
F1034						GR3O[#2]	GR2O[#2]	GR1O[#2]
F1035								SPAL[#2]
F1036	R08O[#2]	R07O[#2]	R06O[#2]	R05O[#2]	R04O[#2]	R03O[#2]	R02O[#2]	R01O[#2]
F1037					R12O[#2]	R11O[#2]	R10O[#2]	R09O[#2]
F1038					ENB3[#2]	ENB2[#2]	SUCLP[#2]	SCLP[#2]
F1039					CHPCYL[#2]	CHPMD[#2]		
F1040	AR7[#2]	AR6[#2]	AR5[#2]	AR4[#2]	AR3[#2]	AR2[#2]	AR1[#2]	AR0[#2]
F1041	AR15[#2]	AR14[#2]	AR13[#2]	AR12[#2]	AR11[#2]	AR10[#2]	AR09[#2]	AR08[#2]
F1042								
F1043								
F1044					FSPPH[#2]	FSPSY[#2]	FSCSL[#2]	
F1045	ORARA[#2]	TLMA[#2]	LDT2A[#2]	LDT1A[#2]	SARA[#2]	SDTA[#2]	SSTA[#2]	ALMA[#2]
F1046	MORA2A[#2]	MORA1A[#2]	PORA2A[#2]	SLVSA[#2]	RCFNA[#2]	RCHPA[#2]	CFINA[#2]	CHPA[#2]
F1047							INCSTA[#2]	PC1DEA[#2]
F1048								
F1049	ORARB[#2]	TLMB[#2]	LDT2B[#2]	LDT1B[#2]	SARB[#2]	SDTB[#2]	SSTB[#2]	ALMB[#2]

	#7	#6	#5	#4	#3	#2	#1	#0
F1050	MORA2B#2	MORA1B#2	PORA2B#2	SLVSB#2	RCFNB#2	RCHPB#2	CFINB#2	CHPB#2
F051							INCSTB#2	PC1DEB#2
F1052								
F1053				BGEACT#2	RPALM#2	RPBSY#2		
F1054	UO007#2	UO006#2	UO005#2	UO004#2	UO003#2	UO002#2	UO001#2	UO000#2
F1055	UO015#2	UO014#2	UO013#2	UO012#2	UO011#2	UO010#2	UO009#2	UO008#2
F1056	UO107#2	UO106#2	UO105#2	UO104#2	UO103#2	UO102#2	UO101#2	UO100#2
F1057	UO115#2	UO114#2	UO11¿#2	UO112#2	UO111#2	UO110#2	UO109#2	UO108#2
F1058	UO123#2	UO122#2	UO121#2	UO120#2	UO119#2	UO118#2	UO117#2	UO116#2
F1059	UO131#2	UO130#2	UO129#2	UO128#2	UO127#2	UO126#2	UO125#2	UO124#2
F1060							ESEND#2	EREND#2
F1061							BCLP#2	BUCLP#2
F1062	PRTSF#2							
F1063	PSYN#2	WATO#2	RCYO#2			PSAR#2	PSE2#2	PSE1#2
F1064						TLCHI#2	TLNW#2	TLCH#2
F1065		SYNMOD#2		RTRCTF#2			RGSPM#2	RGSPP#2
F1066			PECK2#2		RTNMVS#2			G08MD#2
F1067								
F1068								
F1069								
F1070	PSW08#2	PSW07#2	PSW06#2	PSW05#2	PSW04#2	PSW03#2	PSW02#2	PSW01#2
F1071							PSW10#2	PSW09#2
F1072	OUT7#2	OUT6#2	OUT5#2	OUT4#2	OUT3#2	OUT2#2	OUT1#2	OUT0#2
F1073				ZRNO#2		MD4O#2	MD2O#2	MD1O#2
F1074								

	#7	#6	#5	#4	#3	#2	#1	#0
F1075	SPO#2	KEYO#2	DRNO#2	MLKO#2	SBKO#2	BDTO#2		
F1076			ROV2O#2	ROV1O#2	RTAP#2		MP2O#2	MP1O#2
F1077		RTO#2			HS1DO#2	HS1CO#2	HS1BO#2	HS1AO#2
F1078	*FV7O#2	*FV6O#2	*FV5O#2	*FV4O#2	*FV3O#2	*FV2O#2	*FV1O#2	*FV0O#2
F1079	*JV7O#2	*JV6O#2	*JV5O#2	*JV4O#2	*JV3O#2	*JV2O#2	*JV1O#2	*JV0O#2
F1080	*FV15O#2	*FV14O#2	*FV13O#2	*FV12O#2	*FV11O#2	*FV10O#2	*FV09O#2	*FV08O#2
F1081	–J4O#2	+J4O#2	–J3O#2	+J3O#2	–J2O#2	+J2O#2	–J1O#2	+J1O#2
F1082						RVSL#2		
F1083								
F1084								
F1085								
F1086								
F1087								
F1088								
F1089								
F1090						ABTSP2#2	ABTSP1#2	ABTQSV#2
F1091								
F1092			TRSPS#2		TRACT#2			
F1093								
F1094		ZP7#2	ZP6#2	ZP5#2	ZP4#2	ZP3#2	ZP2#2	ZP1#2
F1095								
F1096		ZP27#2	ZP26#2	ZP25#2	ZP24#2	ZP23#2	ZP22#2	ZP21#2
F1097								
F1098		ZP37#2	ZP36#2	ZP35#2	ZP34#2	ZP33#2	ZP32#2	ZP31#2
F1099								

	#7	#6	#5	#4	#3	#2	#1	#0
F1100		ZP47[#2]	ZP46[#2]	ZP45[#2]	ZP44[#2]	ZP43[#2]	ZP42[#2]	ZP41[#2]
F1101								
F1102		MV7[#2]	MV6[#2]	MV5[#2]	MV4[#2]	MV3[#2]	MV2[#2]	MV1[#2]
F1103								
F1104		INP7[#2]	INP6[#2]	INP5[#2]	INP4[#2]	INP3[#2]	INP2[#2]	INP1[#2]
F1105								
F1106		MVD7[#2]	MVD6[#2]	MVD5[#2]	MVD4[#2]	MVD3[#2]	MVD2[#2]	MVD1[#2]
F1107								
F1108		MMI7[#2]	MMI6[#2]	MMI5[#2]	MMI4[#2]	MMI3[#2]	MMI2[#2]	MMI1[#2]
F1109								
F1110		MDTCH7[#2]	MDTCH6[#2]	MDTCH5[#2]	MDTCH4[#2]	MDTCH3[#2]	MDTCH2[#2]	MDTCH1[#2]
F1111								
F1112		EADEN7[#2]	EADEN6[#2]	EADEN5[#2]	EADEN4[#2]	EADEN3[#2]	EADEN2[#2]	EADEN1[#2]
F1113								
F1114		TRQL7[#2]	TRQL6[#2]	TRQL5[#2]	TRQL4[#2]	TRQL3[#2]	TRQL2[#2]	TRQL1[#2]
F1115								
F1116		FRP7[#2]	FRP6[#2]	FRP5[#2]	FRP4[#2]	FRP3[#2]	FRP2[#2]	FRP1[#2]
F1117								
F1118		SYN7O[#2]	SYN6O[#2]	SYN5O[#2]	SYN4O[#2]	SYN3O[#2]	SYN2O[#2]	SYN1O[#2]
F1119								
F1120		ZRF7[#2]	ZRF6[#2]	ZRF5[#2]	ZRF4[#2]	ZRF3[#2]	ZRF2[#2]	ZRF1[#2]
F1121								
F1122	HDO7[#2]	HDO6[#2]	HDO5[#2]	HDO4[#2]	HDO3[#2]	HDO2[#2]	HDO1[#2]	HDO0[#2]
F1123								
F1124								

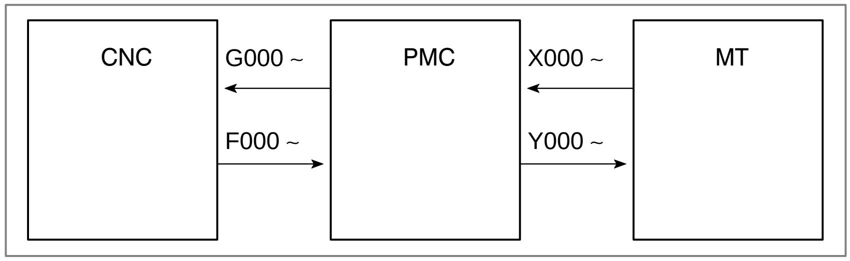
	#7	#6	#5	#4	#3	#2	#1	#0
F1125								
F1126								
F1127								
F1128								
F1129	*EAXSL[#2]		EOV0[#2]					
F1130	EBSYA[#2]	EOTNA[#2]	EOTP[#2]	EGENA[#2]	EDENA[#2]	EIALA[#2]	ECKZA[#2]	EINPA[#2]
F1131							EABUFA[#2]	EMFA[#2]
F1132	EM28A[#2]	EM24A[#2]	EM22A[#2]	EM21A[#2]	EM18A[#2]	EM14A[#2]	EM12A[#2]	EM11A[#2]
F1133	EBSYB[#2]	EOTNB[#2]	EOTB[#2]	EGENB[#2]	EDENB[#2]	EIALB[#2]	ECKZB[#2]	EINPB[#2]
F1134							EABUFB[#2]	EMFB[#2]
F1135	EM28B[#2]	EM24B[#2]	EM22B[#2]	EM21B[#2]	EM18B[#2]	EM14B[#2]	EM12B[#2]	EM11B[#2]
F1136	EBSYC[#2]	EOTNC[#2]	EOTC[#2]	EGENC[#2]	EDENC[#2]	EIALC[#2]	ECKZC[#2]	EINPC[#2]
F1137							EABUFC[#2]	EMFC[#2]
F1138	EM28C[#2]	EM24C[#2]	EM22C[#2]	EM21C[#2]	EM18C[#2]	EM14C[#2]	EM12C[#2]	EM11C[#2]
F1139	EBSYD[#2]	EOTND[#2]	EOTD[#2]	EGEND[#2]	EDEND[#2]	EIALD[#2]	ECKZD[#2]	EINPD[#2]
F1140							EABUFD[#2]	EMFD[#2]
F1141	EM28D[#2]	EM24D[#2]	EM22D[#2]	EM21D#[2]	EM18D[#2]	EM14D[#2]	EM12D[#2]	EM11D[#2]
F1142	EM48A[#2]	EM44A[#2]	EM42A[#2]	EM41A[#2]	EM38A[#2]	EM34A[#2]	EM32A[#2]	EM31A[#2]
F1143								
F1144								
F1145	EM48B[#2]	EM44B[#2]	EM42B[#2]	EM41B[#2]	EM38B[#2]	EM34B[#2]	EM32B[#2]	EM31B[#2]
F1146								
F1147								
F1148	EM48C[#2]	EM44C[#2]	EM42C[#2]	EM41C[#2]	EM38C[#2]	EM34C[#2]	EM32C[#2]	EM31C[#2]
F1149								

	#7	#6	#5	#4	#3	#2	#1	#0
F1150								
F1151	EM48D#2	EM44D#2	EM42D#2	EM41D#2	EM38D#2	EM34D#2	EM32D#2	EM31D#2
F1152								
F1153								
F1154								
F1155								
F1156								
F1157								
F1158								
F1159								
F1160								
F1161								
F1162								
F1163								
F1164								
F1165								
F1166								
F1167								
F1168								
F1169								
F1170								
F1171								
F1172								
F1173								
F1174								

	#7	#6	#5	#4	#3	#2	#1	#0
F1175								
F1176								
F1177	EDGN#2	EPARM#2	EVAR#2	EPRG#2	EWTIO#2	ESTPIO#2	ERDIO#2	IOLNK#2
F1178					SRLNO3#2	SRLNO2#2	SRLNO1#2	SRLNO0#2
F1179								
F1180	CLRCH8#2	CLRCH7#2	CLRCH6#2	CLRCH5#2	CLRCH4#2	CLRCH3#2	CLRCH2#2	CLRCH1#2
F1181								
F1182		EACNT7#2	EACNT6#2	EACNT5#2	EACNT4#2	EACNT3#2	EACNT2#2	EACNT1#2
F1183								
F1184								
F1185								
F1186								
F1187								
F1188								
F1189								
F1190								
F1191								
F1192								
F1193								
F1194								

A.1.3 Series 21*i*/210*i* Address List

The figure below illustrates the addresses of interface signals between the CNC and PMC.



Following shows table of addresses:

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

[Example 1]

Signals EXLM and ST are common signals, STLK is for T series only and RLSOT and RVS are for M series only.

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

MT → PMC

Address / Bit number

Address	#7	#6	#5	#4	#3	#2	#1	#0	
X000									
X001									
X002									
X003									
X004	SKIP	ESKIP / SKIP6	–MIT2 / SKIP5	+MIT2 / SKIP4	–MIT1 / SKIP3	+MIT1 / SKIP2	ZAE / SKIP8	XAE / SKIP7	(T series)
	SKIP	ESKIP				ZAE	YAE	XAE	(M series)
X005									
X006									
X007									
X008				*ESP					
X009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1	
X010									
X011									
X012									

● T series/M series

PMC → CNC

Address	#7	#6	#5	#4	#3	#2	#1	#0
G000	ED7	ED6	ED5	ED4	ED3	ED2	ED1	ED0
G001	ED15	ED14	ED13	ED12	ED11	ED10	ED9	ED8
G002	ESTB	EA6	EA5	EA4	EA3	EA2	EA1	EA0
G003								
G004			MFIN3	MFIN2	FIN			
G005	BFIN	AFL		BFIN	TFIN	SFIN	EFIN	MFIN
G006		SKIPP		OVC		*ABSM		SRN
G007	RLSOT	EXLM	*FLWU	RLSOT3		ST	STLK	
G008	ERS	RRW	*SP	*ESP				*IT
G009				PN16	PN8	PN4	PN2	PN1
G010	*JV7	*JV6	*JV5	*JV4	*JV3	*JV2	*JV1	*JV0
G011	*JV15	*JV14	*JV13	*JV12	*JV11	*JV10	*JV9	*JV8
G012	*FV7	*FV6	*FV5	*FV4	*FV3	*FV2	*FV1	*FV0
G013								
G014							ROV2	ROV1
G015								
G016	F1D							
G017								
G018	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A
G019	RT		MP2	MP1	HS3D	HS3C	HS3B	HS3A
G020								
G021								
G022								
G023								
G024								

	#7	#6	#5	#4	#3	#2	#1	#0
G025								
G026								
G027	CON		*SSTP3	*SSTP2	*SSTP1	SWS3	SWS2	SWS1
G028	PC2SLC	SPSTP	*SCPF	*SUCPF		GR2	GR1	
G029		*SSTP	SOR	SAR		GR31		GR21
G030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
G031	PKESS2	PKESS1						
G032	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G033	SIND	SSIN	SGN		R12I	R11I	R10I	R09I
G034	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G035	SIND2	SSIN2	SGN2		R12I2	R11I2	R10I2	R09I2
G036	R08I3	R07I3	R06I3	R05I3	R04I3	R03I3	R02I3	R01I3
G037	SIND3	SSIN3	SGN3		R12I3	R11I3	R10I3	R09I3
G038	*BECLP	*BEUCP			SPPHS	SPSYC		
G039	GOQSM	WOQSM	OFN5	OFN4	OFN3	OFN2	OFN1	OFN0
G040	WOSET	PRC						
G041	HS2ID	HS2IC	HS2IB	HS2IA	HS1ID	HS1IC	HS1IB	HS1IA
G042	DMMC				HS3ID	HS3IC	HS3IB	HS3IA
G043	ZRN		DNCI			MD4	MD2	MD1
G044							MLK	BDT1
G045	BDT9	BDT8	BDT7	BDT6	BDT5	BDT4	BDT3	BDT2
G046	DRN	KEY4	KEY3	KEY2	KEY1		SBK	
G047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01
G048	TLRST	TLRSTI	TLSKP					TL256
G049	*TLV7	*TLV6	*TLV5	*TLV4	*TLV3	*TLV2	*TLV1	*TLV0

	#7	#6	#5	#4	#3	#2	#1	#0
G050							*TLV9	*TLV8
G051								
G052	RMTDI7	RMTDI6	RMTDI5	RMTDI4	RMTDI3	RMTDI2	RMTDI1	RMTDI0
G053	CDZ	SMZ			UINT			TMRON
G054	UI007	UI006	UI005	UI004	UI003	UI002	UI001	UI000
G055	UI015	UI014	UI013	UI012	UI011	UI010	UI009	UI008
G056								
G057								
G058					EXWT	EXSTP	EXRD	MINP
G059								
G060	*TSB							
G061			RGTSP2	RGTSP1				RGTAP
G062							*CRTOF	
G063			NOZAGC					
G064		ESRSYC						
G065								
G066	EKSET						ENBKY	IGNVRY
G067								
G068								
G069								
G070	MRDYA	ORCMA	SFRA	SRVA	CTH1A	CTH2A	TLMHA	TLMLA
G071	RCHA	RSLA	INTGA	SOCNA	MCFNA	SPSLA	*ESPA	ARSTA
G072	RCHHGA	MFNHGA	INCMDA	OVRIDA	DEFMDA	NRROA	ROTAA	INDXA
G073						MPOFA	SLVA	MORCMA
G074	MRDYB	ORCMB	SFRB	SRVB	CTH1B	CTH2B	TLMHB	TLMLB

	#7	#6	#5	#4	#3	#2	#1	#0
G075	RCHB	RSLB	INTGB	SOCNB	MCFNB	SPSLB	*ESPB	ARSTB
G076	RCHHGB	MFNHGB	INCMDB	OVRIDB	DEFMDB	NRROB	ROTAB	INDXB
G077						MPOFB	SLVB	MORCMB
G078	SHA07	SHA06	SHA05	SHA04	SHA03	SHA02	SHA01	SHA00
G079					SHA11	SHA10	SHA09	SHA08
G080	SHB07	SHB06	SHB05	SHB04	SHB03	SHB02	SHB01	SHB00
G081					SHB11	SHB10	SHB09	SHB08
G082	Reserved for order–made macro							
G083	Reserved for order–made macro							
G084								
G085								
G086								
G087								
G088								
G089								
G090								
G091					SRLNI3	SRLNI2	SRLNI1	SRLNI0
G092				BGEN	BGIALM	BGION	IOLS	IOLACK
G093								
G094								
G095								
G096	HROV	*HROV6	*HROV5	*HROV4	*HROV3	*HROV2	*HROV1	*HROV0
G097								
G098	EKC7	EKC6	EKC5	EKC4	EKC3	EKC2	EKC1	EKC0
G099								

	#7	#6	#5	#4	#3	#2	#1	#0
G100					+J4	+J3	+J2	+J1
G101								
G102					–J4	–J3	–J2	–J1
G103								
G104								
G105								
G106					MI4	MI3	MI2	MI1
G107								
G108					MLK4	MLK3	MLK2	MLK1
G109								
G110					+LM4	+LM3	+LM2	+LM1
G111								
G112					–LM4	–LM3	–LM2	–LM1
G113								
G114					*+L4	*+L3	*+L2	*+L1
G115								
G116					*–L4	*–L3	*–L2	*–L1
G117								
G118					*+ED4	*+ED3	*+ED2	*+ED1
G119								
G120					*–ED4	*–ED3	*–ED2	*–ED1
G121								
G122	PKESS2	PKESS1						
G123								

	#7	#6	#5	#4	#3	#2	#1	#0
G124					DTCH4	DTCH3	DTCH2	DTCH1
G125								
G126					SVF4	SVF3	SVF2	SVF1
G127								
G128								
G129								
G130					*IT4	*IT3	*IT2	*IT1
G131								
G132					+MIT4	+MIT3	+MIT2	+MIT1
G133								
G134					–MIT4	–MIT3	–MIT2	–MIT1
G135								
G136					EAX4	EAX3	EAX2	EAX1
G137								
G138					SYNC4	SYNC3	SYNC2	SYNC1
G139								
G140					SYNCJ4	SYNCJ3	SYNCJ2	SYNCJ1
G141								
G142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA		EFINA
G143	EMSBKA	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A
G146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A

	#7	#6	#5	#4	#3	#2	#1	#0
G149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A
G150	DRNE	RTE	OVCE				ROV2E	ROV1E
G151	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E
G152								
G153								
G154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB		EFINB
G155	EMSBKB	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B
G158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B
G162								
G163								
G164								
G165								
G166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC		EFINC
G167	EMSBKC	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C
G170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

	#7	#6	#5	#4	#3	#2	#1	#0
G174								
G175								
G176								
G177								
G178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD		EFIND
G179	EMSBKD	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D
G180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D
G182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D
G186								
G187								
G188								
G189								
G190								
G191								
G192					IGVRY4	IGVRY3	IGVRY2	IGVRY1
G193								
G194								
G195								
G196								
G197								
G198					NPOS4	NPOS3	NPOS2	NPOS1

	#7	#6	#5	#4	#3	#2	#1	#0
G199								
G200								
G201								
G202								
G203								
G204								
G205								
G206								
G207								
G208								
G209								
G210								
G211								
G212								
G213								
G214								
G215								
G216								
G217								
G218								
G219								
G220								
G221								
G222								

CNC → PMC

Address	#7	#6	#5	#4	#3	#2	#1	#0
F000	OP	SA	STL	SPL				RWD
F001	MA		TAP	ENB	DEN	BAL	RST	AL
F002	MDRN	CUT		SRNMV	THRD	CSS	RPDO	INCH
F003	MTCHIN	MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F004			MREF	MAFL	MSBK	MABSM	MMLK	MBDT1
F005	MBDT9	MBDT8	MBDT7	MBDT6	MBDT5	MBDT4	MBDT3	MBDT2
F006								
F007	BF			BF	TF	SF	EFD	MF
F008			MF3	MF2				EF
F009	DM00	DM01	DM02	DM30				
F010	M07	M06	M05	M04	M03	M02	M01	M00
F011	M15	M14	M13	M12	M11	M10	M09	M08
F012	M23	M22	M21	M20	M19	M18	M17	M16
F013	M31	M30	M29	M28	M27	M26	M25	M24
F014	M207	M206	M205	M204	M203	M202	M201	M200
F015	M215	M214	M213	M212	M211	M210	M209	M208
F016	M307	M306	M305	M304	M303	M302	M301	M300
F017	M315	M314	M313	M312	M311	M310	M309	M308
F018								
F019								
F020								
F021								
F022	S07	S06	S05	S04	S03	S02	S01	S00
F023	S15	S14	S13	S12	S11	S10	S09	S08
F024	S23	S22	S21	S20	S19	S18	S17	S16

	#7	#6	#5	#4	#3	#2	#1	#0
F025	S31	S30	S29	S28	S27	S26	S25	S24
F026	T07	T06	T05	T04	T03	T02	T01	T00
F027	T15	T14	T13	T12	T11	T10	T09	T08
F028	T23	T22	T21	T20	T19	T18	T17	T16
F029	T31	T30	T29	T28	T27	T26	T25	T24
F030	B07	B06	B05	B04	B03	B02	B01	B00
F031	B15	B14	B13	B12	B11	B10	B09	B08
F032	B23	B22	B21	B20	B19	B18	B17	B16
F033	B31	B30	B29	B28	B27	B26	B25	B24
F034						GR3O	GR2O	GR1O
F035								SPAL
F036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F037					R12O	R11O	R10O	R09O
F038					ENB3	ENB2	SUCLP	SCLP
F039								
F040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F041	AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
F042								
F043								
F044				SYCAL	FSPPH	FSPSY	FSCSL	
F045	ORARA	TLMA	LDT2A	LDT1A	SARA	SDTA	SSTA	ALMA
F046	MORA2A	MORA1A	PORA2A	SLVSA	RCFNA	RCHPA	CFINA	CHPA
F047							INCSTA	PC1DEA
F048								
F049	ORARB	TLMB	LDT2B	LDT1B	SARB	SDTB	SSTB	ALMB

	#7	#6	#5	#4	#3	#2	#1	#0
F050	MORA2B	MORA1B	PORA2B	SLVSB	RCFNB	RCHPB	CFINB	CHPB
F051							INCSTB	PC1DEB
F052								
F053	EKENB			BGEACT	RPALM	RPBSY	PRGDPL	INHKY
F054	UO007	UO006	UO005	UO004	UO003	UO002	UO001	UO000
F055	UO015	UO014	UO013	UO012	UO011	UO010	UO009	UO008
F056	UO107	UO106	UO105	UO104	UO103	UO102	UO101	UO100
F057	UO115	UO114	UO113	UO112	UO111	UO110	UO109	UO108
F058	UO123	UO122	UO121	UO120	UO119	UO118	UO117	UO116
F059	UO131	UO130	UO129	UO128	UO127	UO126	UO125	UO124
F060							ESEND	EREND
F061							BCLP	BUCLP
F062	PRTSF							
F063	PSYN							
F064						TLCHI	TLNW	TLCH
F065							RGSPM	RGSPP
F066			PECK2					G08MD
F067								
F068								
F069	RMTDO7	RMTDO6	RMTDO5	RMTDO4	RMTDO3	RMTDO2	RMTDO1	RMTDO0
F070	PSW08	PSW07	PSW06	PSW05	PSW04	PSW03	PSW02	PSW01
F071							PSW10	PSW09
F072	OUT7	OUT6	OUT5	OUT4	OUT3	OUT2	OUT1	OUT0
F073				ZRNO		MD4O	MD2O	MD1O
F074								

	#7	#6	#5	#4	#3	#2	#1	#0
F075	SPO	KEYO	DRNO	MLKO	SBKO	BDTO		
F076			ROV2O	ROV1O	RTAP		MP2O	MP1O
F077		RTO			HS1DO	HS1CO	HS1BO	HS1AO
F078	*FV7O	*FV6O	*FV5O	*FV4O	*FV3O	*FV2O	*FV1O	*FV0O
F079	*JV7O	*JV6O	*JV5O	*JV4O	*JV3O	*JV2O	*JV1O	*JV0O
F080	*JV15O	*JV14O	*JV13O	*JV12O	*JV11O	*JV10O	*JV9O	*JV8O
F081	– J4O	+ J4O	– J3O	+ J3O	– J2O	+ J2O	– J1O	+ J1O
F082								
F083								
F084								
F085								
F086								
F087								
F088								
F089								
F090						ABTSP2	ABTSP1	ABTQSV
F091								
F092								
F093								
F094					ZP4	ZP3	ZP2	ZP1
F095								
F096					ZP24	ZP23	ZP22	ZP21
F097								
F098					ZP34	ZP33	ZP32	ZP31
F099								

	#7	#6	#5	#4	#3	#2	#1	#0
F100					ZP44	ZP43	ZP42	ZP41
F101								
F102					MV4	MV3	MV2	MV1
F103								
F104					INP4	INP3	INP2	INP1
F105								
F106					MVD4	MVD3	MVD2	MVD1
F107								
F108					MMI4	MMI3	MMI2	MMI1
F109								
F110					MDTCH4	MDTCH3	MDTCH2	MDTCH1
F111								
F112					EADEN4	EADEN3	EADEN2	EADEN1
F113								
F114					TRQL4	TRQL3	TRQL2	TRQL1
F115								
F116								
F117								
F118								
F119								
F120					ZRF4	ZRF3	ZRF2	ZRF1
F121								
F122								HDO0
F123								
F124								

	#7	#6	#5	#4	#3	#2	#1	#0
F125								
F126								
F127								
F128								
F129	*EAXSL		EOV0					
F130	EBSYA	EOTNA	EOTPA	EGENA	EDENA	EIALA	ECKZA	EINPA
F131							EABUFA	EMFA
F132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F133	EBSYB	EOTNB	EOTPB	EGENB	EDENB	EIALB	ECKZB	EINPB
F134							EABUFB	EMFB
F135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F136	EBSYC	EOTNC	EOTPC	EGENC	EDENC	EIALC	ECKZC	EINPC
F137							EABUFC	EMFC
F138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
F139	EBSYD	EOTND	EOTPD	EGEND	EDEND	EIALD	ECKZD	EINPD
F140							EABUFD	EMFD
F141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
F142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A
F143								
F144								
F145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B
F146								
F147								
F148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C
F149								

	#7	#6	#5	#4	#3	#2	#1	#0
F150								
F151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D
F152								
F153								
F154								
F155								
F156								
F157								
F158								
F159								
F160								
F161								
F162								
F163								
F164								
F165								
F166								
F167								
F168								
F169								
F170								
F171								
F172								
F173								
F174								

	#7	#6	#5	#4	#3	#2	#1	#0
F175								
F176								
F177	EDGN	EPARM	EVAR	EPRG	EWTIO	ESTPIO	ERDIO	IOLNK
F178					SRLNO3	SRLNO2	SRLNO1	SRLNO0
F179								
F180					CLRCH4	CLRCH3	CLRCH2	CLRCH1
F181								
F182					EACNT4	EACNT3	EACNT2	EACNT1
F183								
F184								
F185								
F186								
F187								
F188								
F189								
F190								
F191								
F192								
F193								
F194								
F195								

A.2 SIGNAL SUMMARY

A.2.1 Signal Summary (In Order of Functions)

○ : Available
● : Available only with 2–path control
– : Unavailable

Function	Signal name	Symbol	Address	T series	M series	Section
Data input/output with I/O Link	Power Mate background operation signal	BGEN	G092#4	○	○	13.8
	Power Mate read/write alarm signal	BGIALM	G092#3	○	○	13.8
	Power Mate read/write busy signal	BGION	G092#2	○	○	13.8
	Slave diagnosis selection signal	EDGN	F177#7	○	○	13.8
	Slave parameter selection signal	EPARM	F177#6	○	○	13.8
	Slave program selection signal	EPRG	F177#4	○	○	13.8
	Slave external read start signal	ERDIO	F177#1	○	○	13.8
	Slave read/write stop signal	ESTPIO	F177#2	○	○	13.8
	Slave macro variable selection signal	EVAR	F177#5	○	○	13.8
	Slave external write start signal	EWTIO	F177#3	○	○	13.8
	External read start signal	EXRD	G058#1	○	○	13.8, 13.5
	External read/punch stop signal	EXSTP	G058#2	○	○	13.8, 13.5
	External punch start signal	EXWT	G058#3	○	○	13.8, 13.5
	I/O Link check signal	IOLACK	G092#0	○	○	13.8
	Slave I/O Link selection signal	IOLINK	F177#0	○	○	13.8
	I/O Link specification signal	IOLS	G092#1	○	○	13.8
	Read/punch alarm signal	RPALM	F053#3	○	○	13.8, 13.5
	Read/punch busy signal	PRBSY	F053#2	○	○	13.8, 13.5
	Group number specification signals	SRLNI0 to SRLNI3	G091#0 to #3	○	○	13.8
	Group number output signals	SRLNO0 to SRLNO3	F178#0 to #3	○	○	13.8
External I/O device control	External read start signal	EXRD	G058#1	○	○	13.5
	External punch start signal	EXWT	G058#3	○	○	
	External read/punch stop signal	EXSTP	G058#2	○	○	
	Background editing signal	BGEACT	F053#4	○	○	
	Read/punch busy signal	RPBSY	F053#2	○	○	
	Read/punch alarm signal	RPALM	F053#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Alarm signal	Alarm signal	AL	F001#0	○	○	2.4
	Battery alarm signal	BAL	F001#2	○	○	
Abnormal load detection	Servo axis abnormal load detected signal	ABTQSV	F090#0	○	○	2.10
	First–spindle abnormal load detected signal	ABTSP1	F090#1	○	○	
	Second–spindle abnormal load detected signal	ABTSP2	F090#2	○	○	
	Third–spindle abnormal load detected signal	ABTSP3	F090#3	○	○	9.15
Position display neglect	Position display neglect signals	NPOS1 to NPOS8	G198	○	○	12.1.10
Multiple M commands in a single block	2nd M function code signals	M200 to M215	F014 to F015	○	○	8.3
	3rd M function code signals	M300 to M315	F016 to F017	○	○	
	2nd M function strobe signal	MF2	F008#4	○	○	
	3rd M function strobe signal	MF3	F008#5	○	○	
Inch/metric conversion	Inch input signal	INCH	F002#0	○	○	11.4
Index table indexing (M series)	B–axis clamp signal	BCLP	F061#1	—	○	11.11
	B–axis clamp completion signal	*BECLP	G038#7	—	○	
	B–axis unclamp signal	BUCLP	F061#0	—	○	
	B–axis unclamp completion signal	*BEUCP	G038#6	—	○	
In–position check	In–position signals	INP1 to INP8	F104	○	○	7.2.6.1
F 1–digit feed (M series)	F 1–digit feed selection signal	F1D	G016#7	—	○	7.1.5
Error detection (T series)	Error detection signal	SMZ	G053#6	○	—	7.2.6.3
Overtravel signal	Overtravel signals	*+L1 to *+L8	G114	○	○	2.3.1
		*–L1 to *–L8	G116	○	○	
Override cancel	Override cancel signal	OVC	G006#4	○	○	7.1.7.4
Feedrate override	Feedrate override signals	*FV0 to *FV7	G012	○	○	7.1.7.2
Optional block skip/addition of optional block skip	Optional block skip signals	BDT1, BDT2 to BDT9	G044#0, G045	○	○	5.5
	Optional block skip check signals	MBDT1, MBDT2 to MBDT9	G004#0, F005	○	○	
External key input	External key input mode selection signal	ENBKY	G066#1	○	○	15.5
	Key code signals	EKC0 to EKC7	G098	○	○	
	Key code read signal	EKSET	G066#7	○	○	
	Key code read completion signal	EKENB	F053#7	○	○	
	Key input disable signal	INHKY	F053#0	○	○	
	Program screen display mode signal	PRGDPL	F053#1	○	○	
External deceleration	External deceleration signals	*+ED1 to *+ED8	G118	○	○	7.1.9
		*–ED1 to *–ED8	G120	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
External data input	Data signals for external data input	ED0 to ED15	G000, G001	○	○	15.2
	Address signals for external data input	EA0 to EA6	G002#0 to #6	○	○	
	Read signal for external data input	ESTB	G002#7	○	○	
	Read completion signal for external data input	EREND	F060#0	○	○	
	Search completion signal for external data input	ESEND	F060#1	○	○	
External motion function (M series)	External operation signal	EF	F008#0	—	○	11.8
External program input	External program input start signal	MINP	G058#0	○	○	13.7
External workpiece number search	Workpiece number search signals	PN1, PN2, PN4, PN8, PN16	G009#0 to 4	○	○	15.3
Machining return and restart (M series)	Machining restart start signal	RTNCY	G064#0	—	○	5.13
	Machining restart point selection signal	SLCSEQ	G064#1	—	○	5.13
	Machining restart completion signal	RCYO	F063#5	—	○	5.13
	Machining start point signal	RTNMVS	F066#3	—	○	5.13
Custom macro signal	Input signals for custom macro	UI000 to UI015	G054, G055	○	○	11.6.1
	Output signals for custom macro	UO000 to UO015	F054, F055	○	○	
		UO100 to UO131	F056 to F059	○	○	
Run hour and part count display	Target part count reached signal	PRTSF	F062#7	○	○	12.1.11
	General–purpose integrating meter start signal	TMRON	G053#0	○	○	
Screen erase/automatic screen erase	Automatic screen erase disable signal	*CRTOF	G062#1	○	○	12.1.19
Simple electronic gearbox (M series)	Retract signal	RTRCT	G066#4	—	○	1.13
	Retract completed signal	RTRCTF	F065#4	—	○	1.13
	EGB mode signal	SYNMOD	F065#6	—	○	1.13
Simple synchronous control	Simple synchronous axis selection signals	SYNC1 to SYNC8	G138	○	○	1.6
	Simple synchronous manual feed axis selection signals	SYNCJ1 to SYNCJ8	G140	—	○	
Angular axis control/arbitrary–axis angular axis control	Angular axis control disable signal for perpendicular axis	NOZAGC	G063#5	○	○	1.11
Path selection	Path selection signal (tool post selection signal)	HEAD	G063#0	●	●	2.7
Tool axis direction handle feed function/tool axis direction handle feed B (M series)	Tool axis direction handle feed mode signal	ALNGH	G023#7	—	○	3.4.1
	Tool axis right–angle direction handle feed mode signal	RGHTH	G023#6	—	○	3.4.2

Function	Signal name	Symbol	Address	T series	M series	Section
Tool life management	Tool change signal	TLCH	F064#0	○	○	10.3
	Tool change reset signal	TLRST	G048#7	○	○	
	Individual tool change signal	TLCHI	F064#2	—	○	
	Individual tool change reset signal	TLRST	IG048#6	—	○	
	Tool skip signal	TLSKP	G048#5	○	○	
	New tool selection signal	TLNW	F064#1	○	○	
	Tool group number selection signals	TL01 to TL256	G047#0 to G048#0	—	○	
		TL01 to TL64	G047#0 to #6	○	—	
	Tool life count override signals	*TLV0 to *TLV9	G049#0 to G050#1	—	○	
Tool retraction and return	Tool retraction signal	TRESC	G059#0	○	○	5.8
	Tool retraction mode signal	TRACT	F092#3	○	○	
	Tool return signal	TRRTN	F059#1	○	○	
	Tool return completion signal	TRSPS	F092#5	○	○	
Automatic tool length measurement (M series)/automatic tool offset (T series)	Measuring position reached signals	XAE	X004#0	○	○	14.2
		YAE	X004#1	—	○	
		ZAE	X004#2	—	○	
		ZAE	X004#1	○	—	
Direct input of tool offset value measured B (T series)	Tool offset value write mode selection signal	GOQSM	G039#7	○	—	14.4.2
	Tool offset value write signals	+MIT1, +MIT2	X004#2, #4	○	—	
		–MIT1, –MIT2	X004#3, #5	○	—	
	Tool offset number selection signals	OFN0 to OFN5, OFN6	G039#0 to #5, G040#0	○	—	
	Workpiece coordinate system shift value write mode selection signal	WOQSM	G039#6	○	—	
	Workpiece coordinate system shift value write signal	WOSET	G040#7	○	—	
Input of tool offset value measured A (T series)	Position record signal	PRC	G040#6	○	—	14.4.1
High–speed M/S/T/B interface	Miscellaneous function completion signal	MFIN	G005#0	○	○	8.4
	Spindle function completion signal	SFIN	G005#2	○	○	
	Tool function completion signal	TFIN	G005#3	○	○	
	2nd auxiliary function completion signal	BFIN	G005#4	○	—	
		BFIN	G005#7	—	○	
	2nd M function completion signal	MFIN2	G004#4	○	○	
	3rd M function completion signal	MFIN3	G004#5	○	○	
	External operation signal for high–speed interface	EFD	F007#1	—	○	
	External operation function completion signal	EFIN	G005#1	—	○	

Function	Signal name	Symbol	Address	T series	M series	Section
High–speed skip signal	High–speed skip status signals	HDO0 to HDO7	F122	○	○	14.3.2
Canned cycle (M series)/canned cycle for hole machining (T series)	Tapping signal	TAP	F001#5	○	○	11.7
Servo off (mechanical handle)	Servo off signals	SVF1 to SVF8	G126	○	○	1.2.8
Servo/spindle motor velocity detection	Motor velocity detection function enable signal	MSDFON	G016#0	○	○	2.11
	Servo motor velocity detection signals	DSV1 to DSV8	Y(n+0)	○	○	2.11
	Spindle motor velocity detection signals	DSP1, DSP2, DSP3	Y(n+1)#0 to #2	○	○	2.11
Cycle start/feed hold	Cycle start signal	ST	G007#2	○	○	5.1
	Feed hold signal	*SP	G008#5	○	○	
	Automatic operation signal	OP	F000#7	○	○	
	Cycle start lamp signal	STL	F000#5	○	○	
	Feed hold lamp signal	SPL	F000#4	○	○	
Cs contour control	Cs contour control change signal	CON	G027#7	○	○	9.9
	Cs contour control change completion signal	FSCSL	F044#1	○	○	
Outputting the movement state of an axis	Axis moving signals	MV1 to MV8	F102	○	○	1.2.5
	Axis moving direction signals	MVD1 to MVD8	F106	○	○	
Actual spindle speed output	Actual spindle speed signals	AR0 to AR15	F040, F041	○	—	9.7
Constant surface speed control	Constant surface speed signal	CSS	F002#2	○	○	9.5
Spindle positioning	Spindle stop completion signal	SPSTP	G028#6	○	—	9.8
	Spindle unclamp signal	SUCLP	F038#1	○	—	
	Spindle unclamp completion signal	*SUCPF	G028#4	○	—	
	Spindle clamp signal	SCLP	F038#0	○	—	
	Spindle clamp completion signal	*SCPF	G028#5	○	—	
Spindle orientation	Spindle orientation external stop position command signals	SHA00 to SHA11	G078#0 to G079#3	○	○	9.13
		SHB00 to SBH11	G080#0 to G081#3	○	○	
		SHC00 to SHC11	G208#0 to G209#3	○	○	9.15
Spindle simple synchronous control	Spindle simple synchronous control signal	ESRSYC	G064#6	○	○	9.16
	First spindle parking signal	PKESS1	G122#6 (G031#6)	○	○	9.16
	Second spindle parking signal	PKESS2	G122#7 (G031#7)	○	○	9.16
	Phase error monitor signal	SYCAL	F044#4	○	○	9.16

Function	Signal name	Symbol	Address	T series	M series	Section
Polygonal turning with two spindles	Polygon spindle stop signal	*PLSST	G038#0	○	—	6.10.2
	Spindle polygon speed reached signal	PSAR	F063#2	○	—	
	Master axis not reached signal	PSE1	F063#0	○	—	
	Polygon synchronous axis not reached signal	PSE2	F063#1	○	—	
Spindle serial output/spindle analog output	Torque limit command LOW signals (serial spindle)	TLMLA	G070#0	○	○	9.2 9.15
		TLMLB	G074#0	○	○	
		TLMLC	G204#0	○	○	
	Torque limit command HIGH signals (serial spindle)	TLMHA	G070#1	○	○	
		TLMHB	G074#1	○	○	
		TLMHC	G204#1	○	○	
	Clutch/gear signals (serial spindle)	CTH1A, CTH2A	G070#3, #2	○	○	
		CTH1B, CTH2B	G074#3, #2	○	○	
		CTH1C, CTH2C	G204#3, #2	○	○	
	CCW command signals (serial spindle)	SRVA	G070#4	○	○	
		SRVB	G074#4	○	○	
		SRVC	G204#4	○	○	
	CW command signals (serial spindle)	SFRA	G070#5	○	○	
		SFRB	G074#5	○	○	
		SFRC	G204#5	○	○	
	Orientation command signals (serial spindle)	ORCMA	G070#6	○	○	
		ORCMB	G074#6	○	○	
		ORCMC	G204#6	○	○	
	Machine ready signals (serial spindle)	MRDYA	G070#7	○	○	
		MRDYB	G074#7	○	○	
		MRDYC	G204#7	○	○	
	Alarm reset signals (serial spindle)	ARSTA	G071#0	○	○	
		ARSTB	G075#0	○	○	
		ARSTC	G205#0	○	○	
	Emergency stop signals (serial spindle)	*ESPA	G071#1	○	○	
		*ESPB	G075#1	○	○	
		*ESPC	G205#1	○	○	
	Spindle selection signals (serial spindle)	SPSLA	G071#2	○	○	
		SPSLB	G075#2	○	○	
		SPSLC	G205#2	○	○	
	Power line switch completion signals (serial spindle)	MCFNA	G071#3	○	○	
		MCFNB	G075#3	○	○	
		MCFNC	G205#3	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Soft start/stop cancel signals (serial spindle)	SOCNA	G071#4	○	○	9.2 9.15
		SOCNB	G075#4	○	○	
		SOCNC	G205#4	○	○	
	Speed integral signals (serial spindle)	INTGA	G071#5	○	○	
		INTGB	G075#5	○	○	
		INTGC	G205#5	○	○	
	Output switch request signals (serial spindle)	RSLA	G071#6	○	○	
		RSLB	G075#6	○	○	
		RSLC	G205#6	○	○	
	Power line status check signals (serial spindle)	RCHA	G071#7	○	○	
		RCHB	G075#7	○	○	
		RCHC	G205#7	○	○	
	Orientation stop position change command signals (serial spindle)	INDXA	G072#0	○	○	
		INDXB	G076#0	○	○	
		INDXC	G206#0	○	○	
	Rotational direction command signals for orientation stop position change (serial spindle)	ROTAA	G072#1	○	○	
		ROTAB	G076#1	○	○	
		ROTAC	G206#1	○	○	
	Shortcut command signals for orientation stop position change (serial spindle)	NRROA	G072#2	○	○	
		NRROB	G076#2	○	○	
		NRROC	G206#2	○	○	
	Differential speed mode command signals (serial spindle)	DEFMDA	G072#3	○	○	
		DEFMDB	G076#3	○	○	
		DEFMDC	G206#3	○	○	
	Analog override signals (serial spindle)	OVRIDA	G072#4	○	○	
		OVRIDB	G076#4	○	○	
		OVRIDC	G206#4	○	○	
	Incremental command externally set orientation signals (serial spindle)	INCMDA	G072#5	○	○	
		INCMDB	G076#5	○	○	
		INCMDC	G206#5	○	○	
	Spindle switch MAIN MCC contact status signals (serial spindle)	MFNHGA	G072#6	○	○	
		MFNHGB	G076#6	○	○	
		MFNHGC	G206#6	○	○	
	Spindle switch HIGH MCC contact status signals (serial spindle)	RCHHGA	G072#7	○	○	
		RCHHGB	G076#7	○	○	
		RCHHGC	G206#7	○	○	
	Magnetic sensor orientation command signals (serial spindle)	MORCMA	G073#0	○	○	
		MORCMB	G077#0	○	○	
		MORCMC	G207#0	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Subordinate operation mode command signals (serial spindle)	SLVA	G073#1	○	○	9.2 9.15
		SLVB	G077#1	○	○	
		SLVC	G207#1	○	○	
	Motor power cutoff command signals (serial spindle)	MPOFA	G073#2	○	○	
		MPOFB	G077#2	○	○	
		MPOFC	G207#2	○	○	
	Alarm signals (serial spindle)	ALMA	F045#0	○	○	
		ALMB	F049#0	○	○	
		ALMC	F168#0	○	○	
	Speed zero signals (serial spindle)	SSTA	F045#1	○	○	
		SSTB	F049#1	○	○	
		SSTC	F168#1	○	○	
	Speed detection signals (serial spindle)	SDTA	F045#2	○	○	
		SDTB	F049#2	○	○	
		SDTC	F168#2	○	○	
	Speed arrival signals (serial spindle)	SARA	F045#3	○	○	
		SARB	F049#3	○	○	
		SARC	F168#3	○	○	
	Load detection signals 1 (serial spindle)	LDT1A	F045#4	○	○	
		LDT1B	F049#4	○	○	
		LDT1C	F168#4	○	○	
	Load detection signals 2 (serial spindle)	LDT2A	F045#5	○	○	
		LDT2B	F049#5	○	○	
		LDT2C	F168#5	○	○	
	Torque limit signals (serial spindle)	TLMA	F045#6	○	○	
		TLMB	F049#6	○	○	
		TLMC	F168#6	○	○	
	Orientation completion signals (serial spindle)	ORARA	F045#7	○	○	
		ORARB	F049#7	○	○	
		ORARC	F168#7	○	○	
	Power line switch signals (serial spindle)	CHPA	F046#0	○	○	
		CHPB	F050#0	○	○	
		CHPC	F169#0	○	○	
	Spindle switch completion signals (serial spindle)	CFINA	F046#1	○	○	
		CFINB	F050#1	○	○	
		CFINC	F169#1	○	○	
	Output switch signals (serial spindle)	RCHPA	F046#2	○	○	
		RCHPB	F050#2	○	○	
		RCHPC	F169#2	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Spindle serial output/spindle analog output	Output switch completion signals (serial spindle)	RCFNA	F046#3	○	○	9.2 9.15
		RCFNB	F050#3	○	○	
		RCFNC	F169#3	○	○	
	Subordinate operation status signals (serial spindle)	SLVSA	F046#4	○	○	
		SLVSB	F050#4	○	○	
		SLVSC	F169#4	○	○	
	Position coder orientation proximity signal (serial spindle)	PORA2A	F046#5	○	○	
		PORA2B	F050#5	○	○	
		PORA2C	F169#5	○	○	
	Magnetic sensor orientation completion signals (serial spindle)	MORA1A	F046#6	○	○	
		MORA1B	F050#6	○	○	
		MORA1C	F169#6	○	○	
	Magnetic sensor orientation proximity signals (serial spindle)	MORA2A	F046#7	○	○	
		MORA2B	F050#7	○	○	
		MORA2C	F169#7	○	○	
	Position coder one–rotation signal detection status signals (serial spindle)	PC1DEA	F047#0	○	○	
		PC1DEB	F051#0	○	○	
		PC1DEC	F170#0	○	○	
	Incremental orientation mode signals (serial spindle)	INCSTA	F047#1	○	○	
		INCSTB	F051#1	○	○	
		INCSTC	F170#1	○	○	
Spindle speed control	Spindle stop signal	*SSTP	G029#6	○	○	9.3
	Spindle orientation signal	SOR	G029#5	○	○	
	Spindle speed override signals	SOV0 to SOV7	G030	○	○	
	Spindle speed arrival signal	SAR	G029#4	○	○	
	Spindle enable signal	ENB	F001#4	○	○	
	Gear selection signals (output)	GR1O, GR2O, GR3O	F034#0 to #2	—	○	
	Gear selection signals (input)	GR1, GR2	G028#1, #2	○	○	
	S 12–bit code signals	R01O to R12O	F036#0 to F037#3	○	○	
Spindle speed fluctuation detection	Spindle fluctuation detection alarm signal	SPAL	F035#0	○	○	9.6
Spindle synchronous control	Spindle synchronous control signal	SPSYC	G038#2	○	○	9.12
	Spindle phase synchronous control signal	SPPHS	G038#3	○	○	
	Spindle synchronous speed control completion signal	FSPSY	F044#2	○	○	
	Spindle phase synchronous control completion signal	FSPPH	F044#3	○	○	
	Spindle synchronous control alarm signal	SYCAL	F044#4	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Manual linear/circular interpolation	Feed axis and direction selection signals	+Jg, –Jg, +Ja, –Ja	G086#0 to #3	○	○	3.5
Manual handle feed	Manual handle feed axis selection signals	HS1A to HS1D	G018#0 to #3	○	○	3.2
		HS2A to HS2D	G018#4 to #7	○	○	
		HS3A to HS3D	G019#0 to #3	—	○	
	Manual handle feed amount selection signals (incremental feed signals)	MP1, MP2	G019#4, #5	○	○	
Manual handle interrupt	Manual handle interrupt axis selection signals	HS1IA to HS1ID	G041#0 to #3	○	○	3.3
		HS2IA to HS2ID	G041#4 to #7	○	○	
		HS3IA to HS3ID	G042#0 to #3	—	○	
Manual reference position return	Manual reference position return selection signal	ZRN	G043#7	○	○	4.1
	Manual reference position return selection check signal	MREF	F004#5	○	○	
	Reference position return deceleration signal	*DEC1 to *DEC8	X009	○	○	
	Reference position return end signal	ZP1 to ZP8	F094	○	○	
	Reference position establishment signal	ZRF1 to ZRF8	F120	○	○	
Jog feed/incremental feed	Feed axis and direction selection signals	+J1 to +J8	G100	○	○	3.1
		–J1 to –J8	G102	○	○	
	Manual feedrate override signals	*JV0 to *JV15	G010, G011	○	○	
	Manual rapid traverse selection signals	RT	G019#7	○	○	
CNC ready signal	CNC ready signal	MA	F001#7	○	○	2.2
	Servo ready signal	SA	F000#6	○	○	
Small hole peck drilling cycle (M series)	Overload torque signal	SKIP	X004#7	—	○	11.17
	Small hole peck drilling–in–progress signal	PECK2	F066#5	—	○	
Status output signal	Rapid traversing signal	RPDO	F002#1	○	○	2.8
	Cutting feed signal	CUT	F002#6	○	○	
Single block	Single block signal	SBK	G046#1	○	○	5.3.3
	Single block check signal	MSBK	F004#3	○	○	
Skip function	Skip signals	SKIP	X004#7	○	○	14.3.1
		SKIPP	G006#6	○	—	

Function	Signal name	Symbol	Address	T series	M series	Section
Start lock/interlock	Start lock signal	STLK	G007#1	○	—	2.5
	Interlock signal	*IT	G008#0	○	○	
	Interlock signal for each axis	*IT1 to *IT8	G130	○	○	
	Manual feed interlock signal for each axis and direction	+MIT1, +MIT2	X004#2, #4	○	—	
	Manual feed interlock signal for each axis and direction	–MIT1, –MIT2	X004#3, #5	○	—	
	Interlock signal for each axis and direction	+MIT1 to +MIT4	G132#0 to #3	—	○	
		–MIT1 to –MIT4	G134#0 to #3	—	○	
Stored stroke limit	Stored stroke limit change signal	EXLM	G007#6	○	○	2.3.2
	Stroke limit external setting signals	+LM1 to +LM8	G110	—	○	
		–LM1 to –LM8	G110	—	○	
	Stroke limit release signal	RLSOT	G007#7	—	○	
Stored stroke limit 2, 3	Stroke limit 3 release signal	RLSOT3	G007#4	○	○	2.3.3
Controlled axis detach	Controlled axis detach signals	DTCH1 to DTCH8	G124	○	○	1.2.4
	Controlled axis detach status signals	MDTCH1 to MDTCH8	F110	○	○	
Look–ahead control (M series)	Look–ahead control mode signal	G08MD	F066#0	—	○	7.1.13
Software operator's panel	Software operator's panel signal (MD1)	MD1O	F073#0	○	○	12.1.15
	Software operator's panel signal (MD2)	MD2O	F073#1	○	○	
	Software operator's panel signal (MD4)	MD4O	F073#2	○	○	
	Software operator's panel signal (ZRN)	ZRNO	F073#4	○	○	
	Software operator's panel signals (+J1 to +J4)	+J1O to +J4O	F081#0, #2, #4, #6	○	○	
	Software operator's panel signals (–J1 to –J4)	–J1O to –J4O	F081#1, #3, #5, #7	○	○	
	Software operator's panel signal (RT)	RTO	F077#6	○	○	
	Software operator's panel signal (HS1A)	HS1AO	F077#0	○	○	
	Software operator's panel signal (HS1B)	HS1BO	F077#1	○	○	
	Software operator's panel signal (HS1C)	HS1CO	F077#2	○	○	
	Software operator's panel signal (HS1D)	HS1DO	F077#3	○	○	
	Software operator's panel signal (MP1)	MP1O	F076#0	○	○	
	Software operator's panel signal (MP2)	MP2O	F076#1	○	○	
	Software operator's panel signals (*JV0 to *JV15)	*JVO0 to *JV15O	F079, F080	○	○	
	Software operator's panel signals (*FV0 to *FV7)	*FV0O to *FV7O	F078	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Software operator's panel	Software operator's panel signal (ROV1)	ROV1O	F076#4	○	○	12.1.15
	Software operator's panel signal (ROV2)	ROV2O	F076#5	○	○	
	Software operator's panel signal (BDT)	BDTO	F075#2	○	○	
	Software operator's panel signal (SBK)	SBKO	F075#3	○	○	
	Software operator's panel signal (MLK)	MLKO	F075#4	○	○	
	Software operator's panel signal (DRN)	DRNO	F075#5	○	○	
	Software operator's panel signals (KEY1 to KEY4)	KEYO	F075#6	○	○	
	Software operator's panel signal (*SP)	SPO	F075#7	○	○	
	Software operator's panel general–purpose switch signals	OUT0 to OUT7	F072	○	○	
Second feedrate override	Second feedrate override signals	*AFV0 to *AVF7	G013	○	○	7.1.7.3
Second reference position return/3rd, 4th reference position return	2nd reference position return completion signals	ZP21 to ZP28	F096	○	○	4.5
	3rd reference position return completion signals	ZP31 to ZP38	F098	○	○	
	4th reference position return completion signals	ZP41 to ZP48	F100	○	○	
Multi–stage skip	Skip signals	SKIP2 to SKIP6, SKIP7, SKIP8	X004#2 to #6, #0, #1	○	○	14.3.3
Canned cycle (M series)/multiple repetitive turning canned cycle (T series)	Chamfering signal	CDZ	G053#7	○	—	11.9
Chuck/tailstock barrier (T series)	Tailstock barrier selection signal	*TSB	G060#7	○	—	2.3.4
Chopping function	Chopping pause signal	*CHLD	G051#7	—	○	1.12
	Chopping start signal	CHPST	G051#6	—	○	1.12
	Chopping speed override signals	*CHP8 to *CHP0	G051#0 to #3	—	○	1.12
	Chopping mode signal	CHPMD	F039#2	—	○	1.12
	Chopping operation signal	CHPCYL	F039#3	—	○	1.12
Butt–type reference position return setting	Torque limit reached signals for butt–type reference position setting	CLRCH1 to CLRCH8	F180	○	○	4.7
DNC operation	DNC operation selection signal	DNCI	G043#5	○	○	5.11
	DNC operation selection confirmation signal	MRMT	F003#4	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Synchronous control	Synchronous control axis selection signals	SYNC1 to SYNC8	G138	○	—	1.8
	Parking signals	PK1 to PK8	G122	○	—	
	Synchronous control under way signals	SYN10 to SYN80	F118	○	—	
	Synchronous control axis selection signals	SYNC1 to SYNC7	G138#0 to #6	●	—	1.9
	Synchronous/composite/superimposed control under way signals	SYN10 to SYN70	F118#0 to #6	●	—	
Dry run	Dry run signal	DRN	G046#7	○	○	5.3.2
	Dry run check signal	MDRN	F002#7	○	○	
Torque limit skip (T series)	Torque limit reached signals	TRQL1 to TRQL8	F114	○	—	14.3.4
Spindle speed control for two–path lathe	Spindle command selection signals	SLSPA, SLSPB	G063#2, #3	●	—	9.4
	Spindle feedback selection signals	SLPCA, SLPCB	G064#2, #3	●	—	
	Spindle command signal	COSP	F064#5	●	—	
Simultaneous input and output operation (M series)	Input and run simultaneous mode selection signal	STRD	G058#5	—	○	13.6
	Output and run simultaneous mode selection signal	STWD	G058#6	—	○	
Thread cutting	Thread cutting signal	THRD	F002#3	○	○	6.4.1
Tool post interference check (T series, two–path control)	Tool post interference check signal	TICHK	F064#6	●	—	2.3.5
	Tool post interference alarm signal	TIALM	F064#7	●	—	
Rapid traverse override	Rapid traverse override signals	ROV1, ROV2	G014#0, #1	○	○	7.1.7.1
	1% step rapid traverse override selection signals	HROV	G096#7	○	○	
	1% step rapid traverse override signals	*HROV0 to *HROV6	G096#0 to #6	○	○	
Direct operation by PMC/MMC	Direct operation selection signal	DMMC	G042#7	○	○	15.6
PMC axis control/PMC axis speed control function	Control axis selection signals (PMC axis control)	EAX1 to EAX8	G136	○	○	15.1
	Axis control command signals (PMC axis control)	EC0A to EC6A	G143#0 to #6	○	○	
		EC0B to EC6B	G155#0 to #6	○	○	
		EC0C to EC6C	G167#0 to #6	○	○	
		EC0D to EC6D	G179#0 to #6	○	○	
	Axis control feedrate signals (PMC axis control)	EIF0A to EIF15A	G144, G145	○	○	
		EIF0B to EIF15B	G156, G157	○	○	
		EIF0C to EIF15C	G168, G169	○	○	
		EIF0D to EIF15D	G180, G181	○	○	
	Axis control command read signals (PMC axis control)	EBUFA	G142#7	○	○	
		EBUFB	G154#7	○	○	
		EBUFC	G166#7	○	○	
		EBUFD	G178#7	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
PMC axis control/PMC axis speed control function	Axis control data signals (PMC axis control)	EID0A to EID31A	G146 to G149	○	○	15.1
		EID0B to EID31B	G158 to G161	○	○	
		EID0C to EID31C	G170 to G173	○	○	
		EID0D to EID31D	G182 to G185	○	○	
	Axis control command read completion signals (PMC axis control)	EBSYA	F130#7	○	○	
		EBSYB	F133#7	○	○	
		EBSYC	F136#7	○	○	
		EBSYD	F139#7	○	○	
	Reset signals (PMC axis control)	ECLRA	G142#6	○	○	
		ECLRB	G154#6	○	○	
		ECLRC	G166#6	○	○	
		ECLRD	G178#6	○	○	
	Axis control temporary stop signals (PMC axis control)	ESTPA	G142#5	○	○	
		ESTPB	G154#5	○	○	
		ESTPC	G166#5	○	○	
		ESTPD	G178#5	○	○	
	Block stop signals (PMC axis control)	ESBKA	G142#3	○	○	
		ESBKB	G154#3	○	○	
		ESBKC	G166#3	○	○	
		ESBKD	G178#3	○	○	
	Block stop disable signals (PMC axis control)	EMSBKA	G143#7	○	○	
		EMSBKB	G155#7	○	○	
		EMSBKC	G167#7	○	○	
		EMSBKD	G179#7	○	○	
	Auxiliary function code signals (PMC axis control)	EM11A to EM48A	F132, F142	○	○	
		EM11B to EM48B	F135, F145	○	○	
		EM11C to EM48C	F138, F148	○	○	
		EM11D to EM48D	F141, F151	○	○	
	Auxiliary function strobe signals (PMC axis control)	EMFA	F131#0	○	○	
		EMFB	F134#0	○	○	
		EMFC	F137#0	○	○	
		EMFD	F140#0	○	○	
	Auxiliary function completion signals (PMC axis control)	EFINA	G142#0	○	○	
		EFINB	G154#0	○	○	
		EFINC	G166#0	○	○	
		EFIND	G178#0	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
PMC axis control/PMC axis speed control function	Servo off signals (PMC axis control)	ESOFA	G142#4	○	○	15.1
		ESOFB	G154#4	○	○	
		ESOFC	G166#4	○	○	
		ESOFD	G178#4	○	○	
	Buffering disable signals (PMC axis control)	EMBUFA	G142#2	○	○	
		EMBUFB	G154#2	○	○	
		EMBUFC	G166#2	○	○	
		EMBUFD	G178#2	○	○	
	Control axis selection status signals (PMC axis control)	*EAXSL	F129#7	○	○	
	In–position signals (PMC axis control)	EINPA	F130#0	○	○	
		EINPB	F133#0	○	○	
		EINPC	F136#0	○	○	
		EINPD	F139#0	○	○	
	Following zero checking signals (PMC axis control)	ECKZA	F130#1	○	○	
		ECKZB	F133#1	○	○	
		ECKZC	F136#1	○	○	
		ECKZD	F139#1	○	○	
	Alarm signals (PMC axis control)	EIALA	F130#2	○	○	
		EIALB	F133#2	○	○	
		EIALC	F136#2	○	○	
		EIALD	F139#2	○	○	
	Axis moving signals (PMC axis control)	EGENA	F130#4	○	○	
		EGENB	F133#4	○	○	
		EGENC	F136#4	○	○	
		EGEND	F139#4	○	○	
	Auxiliary function executing signals (PMC axis control)	EDENA	F130#3	○	○	
		EDENB	F133#3	○	○	
		EDENC	F136#3	○	○	
		EDEND	F139#3	○	○	
	Negative–direction overtravel signals (PMC axis control)	EOTNA	F130#6	○	○	
		EOTNB	F133#6	○	○	
		EOTNC	F136#6	○	○	
		EOTND	F139#6	○	○	
	Positive–direction overtravel signals (PMC axis control)	EOTPA	F130#5	○	○	
		EOTPB	F133#5	○	○	
		EOTPC	F136#5	○	○	
		EOTPD	F139#5	○	○	
	Feedrate override signals (PMC axis control)	*FV0E to *FV7E	G151	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
PMC axis control/PMC axis speed control function	Override cancellation signal (PMC axis control)	OVCE	G150#5	○	○	15.1
	Rapid traverse override signals (PMC axis control)	ROV1E, ROV2E	G150#0, #1	○	○	
	Dry run signal (PMC axis control)	DRNE	G150#7	○	○	
	Manual rapid traverse selection signal (PMC axis control)	RTE	G150#6	○	○	
	Override 0% signal (PMC axis control)	EOV0	F129#5	○	○	
	Skip signal (PMC axis control)	ESKIP	X004#6	○	○	
	Distribution completion signals (PMC axis control)	EADEN1 to EADEN8	F112	○	○	
	Buffer full signals (PMC axis control)	EABUFA	F131#1	○	○	
		EABUFB	F134#1	○	○	
		EABUFC	F137#1	○	○	
		EABUFD	F140#1	○	○	
	Controlling signals (PMC axis control)	EACNT1 to EACNT8	F182	○	○	
Spindle output control by the PMC	Spindle motor speed command selection signals	SIND	G033#7	○	○	15.4
		SIND2	G035#7	○	○	
		SIND3	G037#7	○	○	
	Spindle motor speed command signals	R01I to R12I	G032#0 to G033#3	○	○	
		R01I2 to R12I2	G034#0 to G035#3	○	○	
		R01I3 to R12I3	G036#0 to G037#3	○	○	
	Spindle motor command polarity selection signals	SSIN	G033#6	○	○	
		SSIN2	G035#6	○	○	
		SSIN3	G037#6	○	○	
	Spindle motor command polarity command signals	SGN	G033#5	○	○	
		SGN2	G035#5	○	○	
		SGN3	G037#5	○	○	
Emergency stop	Emergency stop signal	*ESP	G008#4	○	○	2.1
		*ESP	X008#4	○	○	
VRDY off alarm ignore signal	All–axis VRDY off alarm ignore signal	IGNVRY	G066#0	○	○	2.9
	Each–axis VRDY off alarm ignore signal	IGVRY1 to IGVRY8	G192	○	○	
Follow–up	Follow–up signal	*FLWU	G007#5	○	○	1.2.7
Floating reference position return	Floating reference position return end signals	FRP1 to FRP8	F116	○	○	4.6
Program restart	Program restart signal	SRN	G006#0	○	○	5.7
	Program restart under way signal	SRNMV	F002#4	○	○	
Position switch	Position switch signals	PSW01 to PSW10	F070#0 to F071#1	○	○	1.2.9

Function	Signal name	Symbol	Address	T series	M series	Section
Miscellaneous function/2nd auxiliary function	Miscellaneous function code signals	M00 to M31	F010 to F013	○	○	8.1
	Miscellaneous function strobe signal	MF	F007#0	○	○	
	Decode M signals	DM00	F009#7	○	○	
		DM01	F009#6	○	○	
		DM02	F009#5	○	○	
		DM30	F009#4	○	○	
	Spindle function code signals	S00 to S31	F022 to F025	○	○	
	Spindle function strobe signal	SF	F007#2	○	○	
	Tool function code signals	T00 to T31	F026 to F029	○	○	
	Tool function strobe signal	TF	F007#3	○	○	
	2nd auxiliary function code signals	B00 to B31	F030 to F033	○	○	
	2nd auxiliary function strobe signal	BF	F007#4	○	—	
		BF	F007#7	—	○	
	End signal	FIN	G004#3	○	○	
	Distribution end signal	DEN	F001#3	○	○	
Auxiliary function lock	Auxiliary function lock signal	AFL	G005#6	○	○	8.2
	Auxiliary function lock check signal	MAFL	F004#4	○	○	
Polygonal turning	Polygon synchronization under way signal	PSYN	F063#7	○	—	6.10.1
Machine lock	All–axis machine lock signal	MLK	G044#1	○	○	5.3.1
	Each–axis machine lock signal	MLK1 to MLK8	G108	○	○	
	All–axis machine lock check signal	MMLK	F004#1	○	○	
Waiting M code (two–path control)	No–wait signal	NOWT	G063#1	●	●	8.5
	Waiting signal	WATO	G063#6	●	●	
Manual absolute on/off	Manual absolute signal	*ABSM	G006#2	○	○	5.4
	Manual absolute check signal	MABSM	F004#2	○	○	
Multi–spindle control (T series)	Spindle selection signals	SWS1	G027#0	○	—	9.10
		SWS2	G027#1	○	—	
		SWS3	G027#2	○	—	
	Individual spindle stop signals	*SSTP1	G027#3	○	—	
		*SSTP2	G027#4	○	—	
		*SSTP3	G027#5	○	—	
	Gear selection signals (input)	GR21	G029#0	○	—	
		GR31	G029#2	○	—	
	2nd position coder selection signal	PC2SLC	G028#7	○	—	
	Spindle enable signals	ENB2	F038#2	○	—	
		ENB3	F038#3	○	—	
Mirror image	Mirror image signals	MI1 to MI8	G106	○	○	1.2.6
	Mirror image check signals	MMI1 to MMI8	F108	○	○	

Function	Signal name	Symbol	Address	T series	M series	Section
Memory protection key	Memory protection signals	KEY1 to KEY4	G046#3 to #6	○	○	12.2.3
Mode selection	Mode selection signals	MD1, MD2, MD4	G043#3 to #2	○	○	2.6
	Manual data input selection check signal	MMDI	F003#3	○	○	
	Automatic operation selection check signal	MMEM	F003#5	○	○	
	Memory edit selection check signal	MEDT	F003#6	○	○	
	Manual handle feed selection check signal	MH	F003#1	○	○	
	Incremental feed selection check signal	MINC	F003#0	○	○	
	Jog feed selection check signal	MJ	F003#2	○	○	
	TEACH IN selection check signal	MTCHIN	F003#7	○	○	
Rigid tapping	Rigid tapping signal	RGTAP	G061#0	○	○	9.11
		RGSPP	F065#0	—	○	
	Spindle rotation direction signals	RGSPM	F065#1	—	○	
	Rigid tapping–in–progress signal	RTAP	F076#3	○	○	
	Rigid tapping spindle selection signals	RGTSP1, RGTSP2	G061#4, #5	○	—	
High–precision contour control by RISC (M series)	HPCC mode signal	MHPCC	F066#6	—	○	7.1.14
	HPCC operation signal	EXHPCC	F066#7	—	○	
Reset and rewind	External reset signal	ERS	G008#7	○	○	5.2
	Reset & rewind signal	RRW	G008#6	○	○	
	Resetting signal	RST	F001#1	○	○	
	Rewinding signal	RWD	F000#0	○	○	
Retrace (M series)	Retrace signal	RVS	G007#0	—	○	11.15
	Retrace–in–progress signal	RVSL	F082#2	—	○	
Remote buffer	Input signals for remote buffer	RMTDI0 to RMTDI7	G052	○	○	13.2
	Output signals for remote buffer	RMTDO0 to RMTDO7	F069	○	○	
Interrupt type custom macro	Interrupt signal for custom macro	UINT	G053#3	○	○	11.6.2

A.2.2 List of Signals (In Order of Symbols)

○ : Available
● : Available only with 2–path control
– : Unavailable

Group	Symbol	Signal name	Address	T series	M series	Reference item
*	*+ED1 to *+ED8	External deceleration signal	G118	○	○	7.1.9
	*+L1 to *+L8	Overtravel signal	G114	○	○	2.3.1
	*-ED1 to *-ED8	External deceleration signal	G120	○	○	7.1.9
	*-L1 to *-L8	Overtravel signal	G116	○	○	2.3.1
	*ABSM	Manual absolute signal	G006#2	○	○	5.4
	*AFV0 to *AFV7	2nd feedrate override signal	G013	○	○	7.1.7.3
	*BECLP	B–axis clamp completion signal	G038#7	-	○	11.11
	*BEUCP	B–axis unclamp completion signal	G038#6	-	○	11.11
	*CHLD	Chopping hold signal	G051#7	–	○	1.12
	*CHP8 to *CHP0	Chopping feedrate override signals	G051#0 to #3	–	○	1.12
	*CRTOF	Automatic erase CRT screen display cancel signal	G062#1	○	○	12.1.19
	*DEC1 to *DEC8	Deceleration signal for reference position return	X009	○	○	4.1
	*EAXSL	Control axis selection status signal(PMC axis control)	F129#7	○	○	15.1
	*ESP	Emergency stop signal	X008#4	○	○	2.1
	*ESP		G008#4	○	○	
	*ESPA	Emergency stop signal (serial spindle)	G071#1	○	○	9.2, 9.15
	*ESPB		G075#1	○	○	
	*ESPC		G205#1	○	○	
	*FLWU	Follow–up signal	G007#5	○	○	1.2.7
	*FV0 to *FV7	Feedrate override signal	G012	○	○	7.1.7.2
	*FV0E to *FV7E	Feedrate override signal (PMC axis control)	G151	○	○	15.1
	*FV0O to *FV7O	Software operator's panel signal(*FV0 to *FV7)	F078	○	○	12.1.15
	*HROV0 to *HROV6	1% step rapid traverse override signal	G096#0 to #6	○	○	7.1.7.1
	*IT	Interlock signal	G008#0	○	○	2.5
	*IT1 to *IT8	Interlock signal for each axis	G130	○	○	2.5
	*JV0 to *JV15	Manual feedrate override signal	G010,G011	○	○	3.1
	*JV0O to *JV15O	Software operator's panel signal(*JV0 to *JV15)	F079,F080	○	○	12.1.15
	*PLSST	Polygon spindle stop signal	G038#0	○	-	6.10.2

Group	Symbol	Signal name	Address	T series	M series	Reference item
*	*SCPF	Spindle clamp completion signal	G028#5	○	-	9.8
	*SP	Feed hold signal	G008#5	○	○	5.1
	*SSTP	Spindle stop signal	G029#6	○	○	9.3
	*SSTP1	Stop signal in each spindle	G027#3	○	-	9.10
	*SSTP2		G027#4	○	-	
	*SSTP3		G027#5	○	-	
	*SUCPF	Spindle unclamp completion signal	G028#4	○	-	9.8
	*TLV0 to *TLV9	Tool life count override signal	G049#0 to G050#1	-	○	10.3
	*TSB	Tailstock barrier select signal	G060#7	○	-	2.3.4
+	+J1 to +J8	Feed axis and direction selection signal	G100	○	○	3.1
	+J1O to +J4O	Software operator's panel signal(+J1 to +J4)	F081#0,#2,#4,#6	○	○	12.1.15
	+Jg, –Jg, +Ja, –Ja	Feed axis and direction selection signals	G086#0 to #3	○	○	3.5
	+LM1 to +LM8	Stroke limit external setting signal	G110	-	○	2.3.2
	+MIT1,+MIT2	Manual feed interlock signal for each axis	X004#2,#4	○	-	2.5
	+MIT1,+MIT2	Tool offset write signal	X004#2,#4	○	-	14.4.2
	+MIT1 to +MIT4	Interlock signal for each axis and direction	G132#0 to #3	-	○	2.5
–	–J1 to -J8	Feed axis and direction selection signal	G102	○	○	3.1
	–J1O to –J4O	Software operator's panel signal(–J1 to –J4)	F081#1,#3,#5,#7	○	○	12.1.15
	–LM1 to –LM8	Stroke limit external setting signal	G112	-	○	2.3.2
	–MIT1,–MIT2	Manual feed interlock signal for each axis	X004#3,#5	○	-	2.5
	–MIT1,–MIT2	Tool offset write signal		○	-	14.4.2
	–MIT1 to –MIT4	Interlock signal for each axis and direction	G134#0 to #3	-	○	2.5
A	ABTQSV	Servo axis abnormal load detected signal	F090#0	○	○	2.10
	ABTSP1	First–spindle abnormal load detected signal	F090#1	○	○	2.10
	ABTSP2	Second–spindle abnormal load detected signal	F090#2	○	○	2.10
	AFL	Miscellaneous function lock signal	G005#6	○	○	8.2
	AL	Alarm signal	F001#0	○	○	2.4
	ALMA	Alarm signal (serial spindle)	F045#0	○	○	9.2, 9.15
	ALMB		F049#0	○	○	
	ALMC		F168#0	○	○	
	ALNGH	Tool axis direction handle feed mode signal	G023#7	-	○	3.4.1
	AR0 to AR15	Actual spindle speed signal	F040,F041	○	-	9.7
	ARSTA	Alarm reset signal (serial spindle)	G071#0	○	○	9.2, 9.15
	ARSTB		G075#0	○	○	
	ARSTC		G205#0	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
B	B00 to B31	2nd auxiliary function code signal	F030 to F033	○	○	8.1
	BAL	Battery alarm signal	F001#2	○	○	2.4
	BCLP	B–axis clamp signal	F061#1	-	○	11.11
	BDT1,BDT2 to BDT9	Optional block skip signal	G044#0,G045	○	○	5.5
	BDTO	Software operator's panel signal(BDT)	F075#2	○	○	12.1.15
	BF	2nd auxiliary function strobe signal	F007#4	○	-	8.1
	BF		F007#7	-	○	
	BFIN	2nd auxiliary function completion signal	G005#4	○	-	8.4
	BFIN		G005#7	-	○	
	BGEACT	Background busy signal	F053#4	○	○	13.8, 13.5
	BGEN	Power Mate background busy signal	G092#4	○	○	13.8
	BGIALM	Power Mate read/write alarm signal	G092#3	○	○	13.8
	BGION	Power Mate read/write inprogress signal	G092#2	○	○	13.8
	BUCLP	B–axis unclamp signal	F061#0	-	○	11.11
C	CDZ	Chamferring signal	G053#7	○	-	11.9
	CFINA	Spindle switch completion signal (serial spindle)	F046#1	○	○	9.2, 9.15
	CFINB		F050#1	○	○	
	CFINC		F169#1	○	○	
	CHPA	Power line switch signal (serial spindle)	F046#0	○	○	9.2, 9.15
	CHPB		F050#0	○	○	
	CHPC		F269#0	○	○	
	CHPCYL	Chopping cycle signal	F039#3	–	○	1.12
	CHPMD	Chopping–in–progress signal	F039#2	–	○	1.12
	CHPST	Chopping start signal	G051#6	–	○	1.12
	CLRCH1 to CLRCH8	Torque limit reach signals for butt–type reference position setting	F180	○	○	4.7
	CON	Cs contour control change signal	G027#7	○	○	9.9
	COSP	Spindle command signal	F064#5	●	-	9.4
	CSS	Constant surface speed signal	F002#2	○	○	9.5
	CTH1A,CTH2A	Clutch/gear signal (serial spindle)	G070#3,#2	○	○	9.2, 9.15
	CTH1B,CTH2B		G074#3,#2	○	○	
	CTH1C,CTH2C		G204#3,#2	○	○	
	CUT	Cutting feed signal	F002#6	○	○	2.8

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
D	DEFMDA	Differential mode command signal (serial spindle)	G072#3	○	○	9.2, 9.15
	DEFMDB		G076#3	○	○	
	DEFMDC		G206#3	○	○	
	DEN	Distribution end signal	F001#3	○	○	8.1
	DM00	Decode M signal	F009#7	○	○	8.1
	DM01		F009#6	○	○	
	DM02		F009#5	○	○	
	DM30		F009#4	○	○	
	DMMC	Direct operation select signal	G042#7	○	○	15.6
	DNCI	DNC operation select signal	G043#5	○	○	5.11
	DRN	Dry run signal	G046#7	○	○	5.3.2
	DRNE	Dry run signal (PMC axis control)	G150#7	○	○	15.1
	DRNO	Software operator's panel signal(DRN)	F075#5	○	○	12.1.15
	DSP1, DSP2, DSP3	Spindle motor speed detection signals	Y(n+1)#0 to #2	○	○	2.11
	DSV1 to DSV8	Servo motor speed detection signals	Y(n+0)	○	○	2.11
	DTCH1 to DTCH8	Controlled axis detach signal	G124	○	○	1.2.4
E	EA0 to EA6	Address signal for external data input	G002#0 to #6	○	○	15.2
	EABUFA	Buffer full signal (PMC axis control)	F131#1	○	○	15.1
	EABUFB		F134#1	○	○	
	EABUFC		F137#1	○	○	
	EABUFD		F140#1	○	○	
	EACNT1 to EACNT8	Controlling signal (PMC axis control)	F182	○	○	15.1
	EADEN1 to EADEN8	Distribution completion signal(PMC axis control)	F112	○	○	15.1
	EAX1 to EAX8	Control axis select signal (PMC axis control)	G136	○	○	15.1
	EBSYA	Axis control command read completion signal (PMC axis control)	F130#7	○	○	15.1
	EBSYB		F133#7	○	○	15.1
	EBSYC		F136#7	○	○	15.1
	EBSYD		F139#7	○	○	15.1

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EBUFA	Axis control command read signal(PMC axis control)	G142#7	○	○	15.1
	EBUFB		G154#7	○	○	
	EBUFC		G166#7	○	○	
	EBUFD		G178#7	○	○	
	EC0A to EC6A	Axis control command signal (PMC axis control)	G143#0 to #6	○	○	15.1
	EC0B to EC6B		G155#0 to #6	○	○	
	EC0C to EC6C		G167#0 to #6	○	○	
	EC0D to EC6D		G179#0 to #6	○	○	
	ECKZA	Following zero checking signal (PMC axis control)	F130#1	○	○	15.1
	ECKZB		F133#1	○	○	
	ECKZC		F136#1	○	○	
	ECKZD		F139#1	○	○	
	ECLRA	Reset signal (PMC axis control)	G142#6	○	○	15.1
	ECLRB		G154#6	○	○	
	ECLRC		G166#6	○	○	
	ECLRD		G178#6	○	○	
	ED0 to ED15	Data signal for external data input	G000,G001	○	○	15.2
	EDENA	Auxiliary function executing signal (PMC axis control)	F130#3	○	○	15.1
	EDENB		F133#3	○	○	
	EDENC		F136#3	○	○	
	EDEND		F139#3	○	○	
	EDGN	Slave diagnosis selection signal	F177#7	○	○	13.8
	EF	External operation signal	F008#0	-	○	11.8
	EFD	External operation signal for high–speed interface	F007#1	-	○	8.4
	EFIN	External operation function completion signal	G005#1	-	○	8.4
	EFINA	Auxiliary function completion signal (PMC axis control)	G142#0	○	○	15.1
	EFINB		G154#0	○	○	
	EFINC		G166#0	○	○	
	EFIND		G178#0	○	○	
	EGENA	Axis moving signal (PMC axis control)	F130#4	○	○	15.1
	EGENB		F133#4	○	○	
	EGENC		F136#4	○	○	
	EGEND		F139#4	○	○	
	EIALA	Alarm signal (PMC axis control)	F130#2	○	○	15.1
	EIALB		F133#2	○	○	
	EIALC		F136#2	○	○	
	EIALD		F139#2	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EID0A to EID31A	Axis control data signal (PMC axis control)	G146 to G149	○	○	15.1
	EID0B to EID31B		G158 to G161	○	○	
	EID0C to EID31C		G170 to G173	○	○	
	EID0D to EID31D		G182 to G185	○	○	
	EIF0A to EIF15A	Axis control feedrate signal (PMC axis control)	G144,G145	○	○	15.1
	EIF0B to EIF15B		G156,G157	○	○	
	EIF0C to EIF15C		G168,G169	○	○	
	EIF0D to EIF15D		G180,G181	○	○	
	EINPA	In–position signal (PMC axis control)	F130#0	○	○	15.1
	EINPB		F133#0	○	○	
	EINPC		F136#0	○	○	
	EINPD		F139#0	○	○	
	EKC0 to EKC7	Key code signal	G098	○	○	15.5
	EKENB	Key code read completion signal	F053#7	○	○	15.5
	EKSET	key code read signal	G066#7	○	○	15.5
	EM11A to EM48A	Auxiliary function code signal (PMC axis control)	F132,F142	○	○	15.1
	EM11B to EM48B		F135,F145	○	○	
	EM11C to EM48C		F138,F148	○	○	
	EM11D to EM48D		F141,F151	○	○	
	EMBUFA	Buffering disable signal (PMC axis control)	G142#2	○	○	15.1
	EMBUFB		G154#2	○	○	
	EMBUFC		G166#2	○	○	
	EMBUFD		G178#2	○	○	
	EMFA	Auxiliary function strobe signal (PMC axis control)	F131#0	○	○	15.1
	EMFB		F134#0	○	○	
	EMFC		F137#0	○	○	
	EMFD		F140#0	○	○	
	EMSBKA	Block stop disable signal (PMC axis control)	G143#7	○	○	15.1
	EMSBKB		G155#7	○	○	
	EMSBKC		G167#7	○	○	
	EMSBKD		G179#7	○	○	
	ENB	Spindle enable signal	F001#4	○	○	9.3
	ENB2		F038#2	○	-	9.10
	ENB3		F038#3	○	-	
	ENBKY	External key input mode selection signal	G066#1	○	○	15.5

Group	Symbol	Signal name	Address	T series	M series	Reference item
E	EOTNA	Negative–direction overtravel signal (PMC axis control)	F130#6	○	○	15.1
	EOTNB		F133#6	○	○	
	EOTNC		F136#6	○	○	
	EOTND		F139#6	○	○	
	EOTPA	Positive–direction overtravel signal (PMC axis control)	F130#5	○	○	15.1
	EOTPB		F133#5	○	○	
	EOTPC		F136#5	○	○	
	EOTPD		F139#5	○	○	
	EOV0	Override 0% signal (PMC axis control)	F129#5	○	○	15.1
	EPARM	Slave parameter selection signal	F177#6	○	○	13.8
	EPRG	Slave program selection signal	F177#4	○	○	13.8
	ERDIO	Slave external read start signal	F177#1	○	○	13.8
	EREND	Read completion signal for external data input	F060#0	○	○	15.2
	ERS	External reset signal	G008#7	○	○	5.2
	ESBKA	Block stop signal (PMC axis control)	G142#3	○	○	15.1
	ESBKB		G154#3	○	○	
	ESBKC		G166#3	○	○	
	ESBKD		G178#3	○	○	
	ESEND	Search completion signal for external data input	F060#1	○	○	15.2
	ESKIP	Skip signal (PMC axis control)	X004#6	○	○	15.1
	ESOFA	Servo off signal (PMC axis control)	G142#4	○	○	15.1
	ESOFB		G154#4	○	○	
	ESOFC		G166#4	○	○	
	ESOFD		G178#4	○	○	
	ESRSYC	Simple spindle synchronous control signal	G064#6	○	○	9.16
	ESTB	Read signal for external data input	G002#7	○	○	15.2
	ESTPA	Axis control temporary stop signal (PMC axis control)	G142#5	○	○	15.1
	ESTPB		G154#5	○	○	
	ESTPC		G166#5	○	○	
	ESTPD		G178#5	○	○	
	ESTPIO	Slave read/write stop signal	F177#2	○	○	13.8
	EVAR	Slave macro variable selection signal	F177#5	○	○	13.8
	EXHPCC	HPCC operation signal	F066#7	-	○	7.1.14
	EXLM	Stored stroke limit select signal	G007#6	○	○	2.3.2
	EXRD	External read start signal	G058#1	○	○	13.8, 13.5
	EXSTP	External read/punch stop signal	G058#2	○	○	13.8, 13.5
	EXWT	External punch start signal	G058#3	○	○	13.8, 13.5
	EWTIO	Slave external write start signal	F177#3	○	○	13.8

Group	Symbol	Signal name	Address	T series	M series	Reference item
F	F1D	F1–digit feed select signal	G016#7	-	○	7.1.5
	FIN	Completion signal	G004#3	○	○	8.1
	FRP1 to FRP8	Floating reference position return end signal	F116	○	○	4.6
	FSCSL	Cs contour control change completion signal	F044#1	○	○	9.9
	FSPPH	Spindle phase synchronous control completion signal	F044#3	○	○	9.12
	FSPSY	Spindle synchronous speed control completion signal	F044#2	○	○	9.12
G	G08MD	Lock–ahead control mode signal	F066#0	-	○	7.1.13
	GOQSM	Tool offset value write mode select signal	G039#7	○	-	14.4.2
	GR1,GR2	Gear selection signal (input)	G028#1,#2	○	○	9.3
	GR1O,GR2O,GR3O	Gear selection signal (output)	F034#0 to #2	-	○	9.3
	GR21	Gear selection signal (input)	G029#0	○	-	9.10
	GR31		G029#2	○	-	
H	HDO0 to HDO7	High–speed skip status signal	F122	○	○	14.3.2
	HEAD	Path selection signal (Tool post selection signal)	G063#0	●	●	2.7
	HROV	1% step rapid traverse override select signal	G096#7	○	○	7.1.7.1
	HS1A to HS1D	Manual handle feed axis selection signal	G018#0 to #3	○	○	3.2
	HS1AO	Software operator’s panel signal(HS1A)	F077#0	○	○	12.1.15
	HS1BO	Software operator’s panel signal(HS1B)	F077#1	○	○	12.1.15
	HS1CO	Software operator’s panel signal(HS1C)	F077#2	○	○	12.1.15
	HS1DO	Software operator’s panel signal(HS1D)	F077#3	○	○	12.1.15
	HS1IA to HS1ID	Manual handle interruption axis select signal	G041#0 to #3	○	○	3.3
	HS2A to HS2D	Manual handle feed axis selection signal	G018#4 to #7	○	○	3.2
	HS2IA to HS2ID	Manual handle interruption axis select signal	G041#4 to #7	○	○	3.3
	HS3A to HS3D	Manual handle feed axis selection signal	G019#0 to #3	-	○	3.2
	HS3IA to HS3ID	Manual handle interruption axis select signal	G042#0 to #3	-	○	3.3

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
I	IGNVRY	All–axis VRDY OFF alarm ignore signal	G066#0	○	○	2.9
	IGVRY1 to IGVRY8	Each–axis VRDY OFF alarm ignore signal	G192	○	○	2.9
	INCH	Inch input signal	F002#0	○	○	11.4
	INCMDA	Incremental command external setting type orientation signal (serial spindle)	G072#5	○	○	9.2, 9.15
	INCMDB		G076#5	○	○	
	INCMDC		G206#5	○	○	
	INCSTA	Incremental method orientation signal (serial spindle)	F047#1	○	○	9.2, 9.15
	INCSTB		F051#1	○	○	
	INCSTC		F170#1	○	○	
	INDXA	Orientation stop position change signal (serial spindle)	G072#0	○	○	9.2, 9.15
	INDXB		G076#0	○	○	
	INDXC		G206#0	○	○	
	INHKY	Key input disable signal	F053#0	○	○	15.5
	INP1 to INP8	In–position signal	F104	○	○	7.2.6.1
	INTGA	Signal for controlling velocity integration (serial spindle)	G071#5	○	○	9.2, 9.15
	INTGB		G075#5	○	○	
	INTGC		G205#5	○	○	
	IOLACK	I/O Link confirmation signal	G092#0	○	○	13.8
	IOLNK	Slave I/O Link selection signal	F177#0	○	○	13.8
	IOLS	I/O Link specification signal	G092#1	○	○	13.8
K	KEY1 to KEY4	Memory protect signal	G046#3 to #6	○	○	12.2.3
	KEYO	Software operator's panel signal(KEY1 to KEY4)	F075#6	○	○	12.1.15
L	LDT1A	Load detection signal 1 (serial spindle)	F045#4	○	○	9.2, 9.15
	LDT1B		F049#4	○	○	
	LDT1C		F168#4	○	○	
	LDT2A	Load detection signal 2 (serial spindle)	F045#5	○	○	9.2, 9.15
	LDT2B		F049#5	○	○	
	LDT2C		F168#5	○	○	
M	M00 to M31	Miscellaneous function code signal	F010 to F013	○	○	8.1
	M200 to M215	2nd M function code signal	F014 to F015	○	○	8.3
	M300 to M315	3rd M function code signal	F016 to F017	○	○	8.3
	MA	CNC ready signal	F001#7	○	○	2.2
	MABSM	Manual absolute check signal	F004#2	○	○	5.4
	MAFL	Miscellaneous function lock check signal	F004#4	○	○	8.2
	MBDT1,MBDT2 to MBDT9	Optional block skip check signal	F004#0,F005	○	○	5.5

Group	Symbol	Signal name	Address	T series	M series	Reference item
M	MCFNA	Power line switch completion signal (serial spindle)	G071#3	○	○	9.2, 9.15
	MCFNB		G075#3	○	○	
	MCFNC		G205#3	○	○	
	MD1,MD2,MD4	Mode selection signal	G043#0 to #2	○	○	2.6
	MD1O	Software operator’s panel signal(MD1)	F073#0	○	○	12.1.15
	MD2O	Software operator’s panel signal(MD2)	F073#1	○	○	12.1.15
	MD4O	Software operator’s panel signal(MD4)	F073#2	○	○	12.1.15
	MDRN	Dry run check signal	F002#7	○	○	5.3.2
	MDTCH1 to MDTCH8	Controlled axis detach status signal	F110	○	○	1.2.4
	MEDT	Memory edit select check signal	F003#6	○	○	2.6
	MF	Auxiliary function strobe signal	F007#0	○	○	8.1
	MF2	2nd M function strobe signal	F008#4	○	○	8.3
	MF3	3rd M function strobe signal	F008#5	○	○	8.3
	MFIN	Auxiliary function completion signal	G005#0	○	○	8.4
	MFIN2	2nd M function completion signal	G004#4	○	○	8.4
	MFIN3	3rd M function completion signal	G004#5	○	○	8.4
	MFNHGA	Main spindle MCC status signal while changing spindles signal (serial spindle)	G072#6	○	○	9.2, 9.15
	MFNHGB		G076#6	○	○	
	MFNHGC		G206#6	○	○	
	MH	Manual handle feed select check signal	F003#1	○	○	2.6
	MHPCC	HPCC mode signal	F066#6	-	○	7.1.14
	MI1 to MI8	Mirror image signal	G106	○	○	1.2.6
	MINC	Incremental feed select check signal	F003#0	○	○	2.6
	MINP	External program input start signal	G058#0	○	○	13.7
	MIX1 to MIX7	Composite control axis selection signals	G128#0 to #6	●	–	1.9
	MJ	JOG feed select check signal	F003#2	○	○	2.6
	MLK	All–axis machine lock signal	G044#1	○	○	5.3.1
	MLK1 to MLK8	Each–axis machine lock signal	G108	○	○	5.3.1
	MLKO	Software operator’s panel signal(MLK)	F075#4	○	○	12.1.15
	MMDI	Manual data input select check signal	F003#3	○	○	2.6
	MMEM	Automatic operation select check signal	F003#5	○	○	2.6
	MMI1 to MMI8	Mirror image check signal	F108	○	○	1.2.6

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
M	MMLK	All–axis machine lock check signal	F004#1	○	○	5.3.1
	MORA1A	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	F046#6	○	○	9.2, 9.15
	MORA1B		F050#6	○	○	
	MORA1C		F169#6	○	○	
	MORA2A	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	F046#7	○	○	9.2, 9.15
	MORA2B		F050#7	○	○	
	MORA2C		F169#7	○	○	
	MORCMA	Command for spindle orientaion with a magnetic sensor (serial spindle)	G073#0	○	○	9.2, 9.15
	MORCMB		G077#0	○	○	
	MORCMC		G207#0	○	○	
	MP1,MP2	Manual handle feed amount selection signal (incremental feed signal)	G019#4,#5	○	○	3.2
	MP1O	Software operator's panel signal(MP1)	F076#0	○	○	12.1.15
	MP2O	Software operator's panel signal(MP2)	F076#1	○	○	12.1.15
	MPOFA	Motor power stop signal (serial spindle)	G073#2	○	○	9.2, 9.15
	MPOFB		G077#2	○	○	
	MPOFC		G207#2	○	○	
	MRDYA	Machine ready signal (serial spindle)	G070#7	○	○	9.2, 9.15
	MRDYB		G074#7	○	○	
	MRDYC		G204#7	○	○	
	MREF	Manual reference position return selection check signal	F004#5	○	○	4.1
	MRMT	DNC operation select check signal	F003#4	○	○	5.11
	MSBK	Single block check signal	F004#3	○	○	5.3.3
	MSDFON	Motor speed detection function enable signal	G016#0	○	○	2.11
	MTCHIN	TEACH IN select check signal	F003#7	○	○	2.6
	MV1 to MV8	Axis moving signal	F102	○	○	1.2.5
	MVD1 to MVD8	Axis moving direction signal	F106	○	○	1.2.5
N	NOWT	No–wait signal	G063#1	●	●	8.5
	NOZAGC	Perpendicular/angular axis control disable signal	G063#5	○	○	1.11
	NPOS1 to NPOS8	Position display neglect signal	G198	○	○	12.1.10
	NRROA	Short–distant movement command while changing the orientation stop position signal (serial spindle)	G072#2	○	○	9.2, 9.15
	NRROB		G076#2	○	○	
	NRROC		G206#2	○	○	

Group	Symbol	Signal name	Address	T series	M series	Reference item
O	OFN0 to OFN5,OFN6	Tool offset number select signal	G039#0 to #5,G040#0	○	-	14.4.2
	OP	Automatic operation signal	F000#7	○	○	5.1
	ORARA	Orientation completion signal (serial spindle)	F045#7	○	○	9.2, 9.15
	ORARB		F049#7	○	○	
	ORARC		F168#7	○	○	
	ORCMA	Orientation command signal (serial spindle)	G070#6	○	○	9.2, 9.15
	ORCMB		G074#6	○	○	
	ORCMC		G204#6	○	○	
	OUT0 to OUT7	Software operator's panel general–purpose switch signal	F072	○	○	12.1.15
	OVC	Override cancel signal	G006#4	○	○	7.1.7.4
	OVCE	Override cancellation signal (PMC axis control)	G150#5	○	○	15.1
	OVLS1 to OVLS7	Superimposed control axis selection signals	G190#0 to #6	●	-	1.9
	OVRIDA	Analog override command signal (serial spindle)	G072#4	○	○	9.2, 9.15
	OVRIDB		G076#4	○	○	
	OVRIDC		G206#4	○	○	
P	PC1DEA	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	F047#0	○	○	9.2, 9.15
	PC1DEB		F051#0	○	○	
	PC1DEC		F170#0	○	○	
	PC2SLC	2nd position coder selection signal	G028#7	○	-	9.10
	PDT1	Conversation mode selection signal	G062#4	○	-	
	PDT2	Restart operation notification signal	G062#5	○	-	
	PECK2	Small–diameter peck drilling in progress signal	F066#5	-	○	11.17
	PK1 to PK8	Parking signals	G122	○	-	1.8
	PK1 to PK7	Parking signals	G122#0 to #6	●	-	1.9
	PKESS1	First spindle synchronous control signal	G122#6 (G031#6)	○	○	9.16
	PKESS2	Second spindle synchronous control signal	G122#7 (G031#7)	○	○	9.16
	PN1,PN2,PN4,PN8, PN16	Workpiece number search signal	G009#0 to 4	○	○	15.3
	PORA2A	Signal for approximate spindle orientation with a position coder (serial spindle)	F046#5	○	○	9.2, 9.15
	PORA2B		F050#5	○	○	
	PORA2C		F169#5	○	○	
	PRC	Position record signal	G040#6	○	-	14.4.1
	PRGDPL	program screen display mode signal	F053#1	○	○	15.5
	PRTSF	Target parts count reached signal	F062#7	○	○	12.1.11
	PSAR	Spindle polygon speed arrival signal	F063#2	○	-	6.10.2

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
P	PSE1	Master axis not arrival signal	F063#0	○	-	6.10.2
	PSE2	Polygon synchronous axis not arrival signal	F063#1	○	-	6.10.2
	PSW01 to PSW10	Position switch signal	F070#0 to F071#1	○	○	1.2.9
	PSYN	Polygon synchronization under way signal	F063#7	○	-	6.10.1
R	R01I to R12I	Spindle motor speed command signal	G032#0 to G033#3	○	○	15.4
	R01I2 to R12I2		G034#0 to G035#3	○	○	
	R01I3 to R12I3		G036#0 to G037#3	○	○	
	R01O to R12O	S12–bit code signal	F036#0 to F037#3	○	○	9.3
	RCFNA	Output switch completion signal (serial spindle)	F046#3	○	○	9.2, 9.15
	RCFNB		F050#3	○	○	
	RCFNC		F169#3	○	○	
	RCHA	Power line status check signal (serial spindle)	G071#7	○	○	9.2, 9.15
	RCHB		G075#7	○	○	
	RCHC		G205#7	○	○	
	RCHHGA	High–output MCC status signal while a magnetic sensor (serial spindle)	G072#7	○	○	9.2, 9.15
	RCHHGB		G076#7	○	○	
	RCHHGC		G206#7	○	○	
	RCHPA	Output switch signal (serial spindle)	F046#2	○	○	9.2, 9.15
	RCHPB		F050#2	○	○	
	RCHPC		F169#2	○	○	
	RCYO	Retry complete signal	F063#5	–	○	5.13
	RGHTH	Tool axis perpendicular direction handle feed mode signal	G023#6	-	○	3.4
	RGSPM	Spindle rotation direction signal	F065#1	-	○	9.11
	RGSPP		F065#0	-	○	
	RGTAP	Rigid tapping signal	G061#0	○	○	9.11
	RGTSP1,RGTSP2	Rigid tapping spindle selection signal	G061#4,#5	○	-	9.11
	RLSOT3	Stroke check 3 release signal	G007#4	○	○	2.3.3
	RMTDI0 to RMTDI7	Input signal for remote buffer	G052	○	○	13.2
	RMTDO0 to RMTDO7	Output signal for remote buffer	F069	○	○	13.2
	ROTAA	Rotation direction command while changing the orientation stop position signal (serial spindle)	G072#1	○	○	9.2, 9.15
	ROTAB		G076#1	○	○	
	ROTAC		G206#1	○	○	
	ROV1,ROV2	Rapid traverse override signal	G014#0,#1	○	○	7.1.7.1

Group	Symbol	Signal name	Address	T series	M series	Reference item
R	ROV1E,ROV2E	Rapid traverse override signal(PMC axis control)	G150#0,#1	○	○	15.1
	ROV1O	Software operator's panel signal(ROV1)	F076#4	○	○	12.1.15
	ROV2O	Software operator's panel signal(ROV2)	F076#5	○	○	12.1.15
	RPALM	Read/punch alarm signal	F053#3	○	○	13.8, 13.5
	RPBSY	Read/punch in–progress signal	F053#2	○	○	13.8, 13.5
	RPDO	Rapid traversing signal	F002#1	○	○	2.8
	RRW	Reset&rewind signal	G008#6	○	○	5.2
	RSLA	Output switch request signal (serial spindle)	G071#6	○	○	9.2, 9.15
	RSLB		G075#6	○	○	
	RSLC		G205#6	○	○	
	RST	Reset signal	F001#1	○	○	5.2
	RT	Manual rapid traverse selection signal	G019#7	○	○	3.1
	RTAP	Rigid tapping in–progress signal	F076#3	○	○	9.11
	RTE	Manual rapid traverse selection signal (PMC axis control)	G150#6	○	○	15.1
	RTO	Software operator's panel signal(RT)	F077#6	○	○	12.1.15
	RTNCY	Retry start signal	G064#0	–	○	5.13
	RTNMVS	Retry point signal	F066#3	–	○	5.13
	RTRCT	Retract signal	G066#4	–	○	1.13
	RTRCTF	Retract completion signal	F065#4	–	○	1.13
	RVS	Retrace signal	G007#0	-	○	11.15
	RVSL	Retrace–in–progress signal	F082#2	-	○	11.15
	RWD	Rewinding signal	F000#0	○	○	5.2
S	S00 to S31	Spindle speed code signal	F022 to F025	○	○	8.1
	SA	Servo ready signal	F000#6	○	○	2.2
	SAR	Spindle speed arrival signal	G029#4	○	○	9.3
	SARA	Speed arrival signal (serial spindle)	F045#3	○	○	9.2, 9.15
	SARB		F049#3	○	○	
	SARC		F168#3	○	○	
	SBK	Single block signal	G046#1	○	○	5.3.3
	SBKO	Software operator's panel signal(SBK)	F075#3	○	○	12.1.15
	SCLP	Spindle clamp signal	F038#0	○	-	9.8
	SDTA	Speed detection signal (serial spindle)	F045#2	○	○	9.2, 9.15
	SDTB		F049#2	○	○	
	SDTC		F168#2	○	○	

Group	Symbol	Signal name	Address	T series	M series	Refer-ence item
S	SF	Spindle speed strobe signal	F007#2	○	○	8.1
	SFIN	Spindle function completion signal	G005#2	○	○	8.4
	SFRA	CW command signal (serial spindle)	G070#5	○	○	9.2, 9.15
	SFRB		G074#5	○	○	
	SFRC		G204#5	○	○	
	SGN	Spindle motor command polarity select signal	G033#5	○	○	15.4
	SGN2		G035#5	○	○	
	SGN3		G037#5	○	○	
	SHA00 to SHA11	Spindle orientation external stop position command signal	G078#0 to G079#3	○	○	9.13
	SHB00 to SHB11		G080#0 to G081#3	○	○	
	SHC00 to SHC11	Spindle orientation stop position external command signal	G208#0 to G209#3	○	○	9.15
	SIND	Spindle motor speed command select signal	G033#7	○	○	15.4
	SIND2		G035#7	○	○	
	SIND3		G037#7	○	○	
	SKIP	Skip signal	X004#7	○	○	14.3.1
		Overload torque signal	X004#7	-	○	11.17
	SKIP2 to SKIP6, SKIP7,SKIP8	Skip signal	X004#2 to #6, #0,#1	○	○	14.3.3
	SKIPP	Skip signal	G006#6	○	-	14.3.1
	SLCSEQ	Retry point selection signal	G064#1		○	5.13
	SLPCA,SLPCB	Spindle return select signal	G064#2,#3	●	-	9.4
	SLSPA,SLSPB	Spindle command select signal	G063#2,#3	●	-	9.4
	SLVA	Slave operation command signal (serial spindle)	G073#1	○	○	9.2, 9.15
	SLVB		G077#1	○	○	
	SLVC		G207#1	○	○	
	SLVSA	Slave operation status signal (serial spindle)	F046#4	○	○	9.2, 9.15
	SLVSB		F050#4	○	○	
	SLVSC		F169#4	○	○	
	SMZ	Error detect signal	G053#6	○	-	7.2.6.3
	SOCNA	Soft start/stop cancel signal (serial spindle)	G071#4	○	○	9.2, 9.15
	SOCNB		G075#4	○	○	
	SOCNC		G205#4	○	○	
	SOR	Spindle orientation signal	G029#5	○	○	9.3
	SOV0 to SOV7	Spindle speed override signal	G030	○	○	9.3
	SPAL	Spindle fluctuation detection alarm signal	F035#0	○	○	9.6
	SPL	Feed hold lamp signal	F000#4	○	○	5.1

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SPO	Software operator's panel signal(*SP)	F075#7	○	○	12.1.15
	SPPHS	Spindle phase synchronous control signal	G038#3	○	○	9.12
	SPSLA	Spindle select signal (serial spindle)	G071#2	○	○	9.2, 9.15
	SPSLB		G075#2	○	○	
	SPSLC		G205#2	○	○	
	SPSTP	Spindle stop complete signal	G028#6	○	-	9.8
	SPSYC	Spindle synchronous control signal	G038#2	○	○	9.12
	SRLNI0 to SRLNI3	Group number specification signals	G091#0 to #3	○	○	13.8
	SRLNO0 to SRLNO3	Group number output signals	F178#0 to #3	○	○	13.8
	SRN	Program restart signal	G006#0	○	○	5.7
	SRNMV	Program restart under way signal	F002#4	○	○	5.7
	SRVA	CCW command signal (serial spindle)	G070#4	○	○	9.2, 9.15
	SRVB		G074#4	○	○	
	SRVC		G204#4	○	○	
	SSIN	Spindle motor command polarity select signal	G033#6	○	○	15.4
	SSIN2		G035#6	○	○	
	SSIN3		G037#6	○	○	
	SSTA	Speed zero signal (serial spindle)	F045#1	○	○	9.2, 9.15
	SSTB		F049#1	○	○	
	SSTC		F168#1	○	○	
	ST	Cycle start lamp signal	G007#2	○	○	5.1
	STL	Cycle start signal	F000#5	○	○	5.1
	STLK	Start lock signal	G007#1	○	-	2.5
	STRD	Input and run simultaneous mode select signal	G058#5	-	○	13.6
	STWD	Output and run simultaneous mode select signal	G058#6	-	○	13.6
	SUCLP	Spindle unclamp signal	F038#1	○	-	9.8
	SVF1 to SVF8	Servo off signal	G126	○	○	1.2.8
	SWS1	Spindle selection signal	G027#0	○	-	9.10
	SWS2		G027#1	○	-	
	SWS3		G027#2	○	-	
	SYCAL	Spindle synchronous control alarm signal	F044#4	○	○	9.12
	SYCAL	Phase error monitor signal	F044#4	○	○	9.16
	SYN1O to SYN8O	Synchronous control under way signals	F118	○	-	1.8

Group	Symbol	Signal name	Address	T series	M series	Reference item
S	SYN1O to SYN7O	Synchronous/composite/superimposed control under way signals	F118#0 to #6	●	-	1.9
	SYNC1 to SYNC8	Simple synchronous axis select signal	G138	○	○	1.6
	SYNC to SYNC8	Synchronous control axis selection signals	G138	○	-	1.8
	SYNC to SYNC7	Synchronous control axis selection signals	G138#0 to #6	●	-	1.9
	SYNCJ1 to SYNCJ8	Simple synchronous manual feed axis select signal	G140	-	○	1.6
	SYNMOD	EGB mode signal	F065#6		○	1.13
T	T00 to T31	Tool function code signal	F026 to F029	○	○	8.1
	TAP	Tapping signal	F001#5	○	○	11.7
	TF	Tool function strobe signal	F007#3	○	○	8.1
	TFIN	Tool function completion signal	G005#3	○	○	8.4
	THRD	Thread cutting signal	F002#3	○	○	6.4.1
	TIALM	Tool post interference alarm signal	F064#7	●	-	2.3.5
	TICHK	Tool post interference check signal	F064#6	●	-	2.3.5
	TL01 to TL64	Tool group number select signal	G047#0 to #6	○	-	10.3
	TL01 to TL256		G047#0 to G048#0	-	○	
	TLCH	Tool change signal	F064#0	○	○	10.3
	TLCHI	Individual tool change signal	F064#2	-	○	10.3
	TLMA	Torque limit signal (serial spindle)	F045#6	○	○	9.2, 9.15
	TLMB		F049#6	○	○	
	TLMC		F168#6	○	○	
	TLMHA	Torque limit command HIGH signal (serial spindle)	G070#1	○	○	9.2, 9.15
	TLMHB		G074#1	○	○	
	TLMHC		G204#1	○	○	
	TLMLA	Torque limit command LOW signal (serial spindle)	G070#0	○	○	9.2, 9.15
	TLMLB		G074#0	○	○	
	TLMLC		G204#0	○	○	
	TLNW	New tool select signal	F064#1	○	○	10.3
	TLRST	Tool change reset signal	G048#7	○	○	10.3
	TLRSTI	Individual tool change reset signal	G048#6	-	○	10.3
	TLSKP	Tool skip signal	G048#5	○	○	10.3
	TMRON	General–purpose integrating meter start signal	G053#0	○	○	12.1.11
	TRACT	Tool retraction mode signal	F092#3	○	○	5.8
	TRESC	Tool retraction signal	G059#0	○	○	5.8
	TRQL1 to TRQL8	Torque limit reached signal	F114	○	-	14.3.4
	TRRTN	Tool return signal	G059#1	○	○	5.8
	TRSPS	Tool return completion signal	F092#5	○	○	5.8

Group	Symbol	Signal name	Address	T series	M series	Reference item
U	UI000 to UI015	Input signal for custom macro	G054,G055	○	○	11.6.1
	UINT	Interrupt signal for custom macro	G053#3	○	○	11.6.2
	UO000 to UO015	Output signal for custom macro	F054,F055	○	○	11.6.1
	UO100 to UO131		F056 to F059	○	○	
W	WATO	Waiting signal	F063#6	●	●	8.5
	WOQSM	Workpiece coordinate system shift value write mode select signal	G039#6	○	-	14.4.2
	WOSET	Workpiece coordinate system shift value write signal	G040#7	○	-	14.4.2
X	XAE	Measuring position reached signal	X004#0	○	○	14.2
Y	YAE		X004#1	-	○	
Z	ZAE		X004#1	○	-	
	ZAE		X004#2	-	○	
	ZP1 to ZP8	Reference position return end signal	F094	○	○	4.1
	ZP21 to ZP28	2nd reference position return end signal	F096	○	○	4.5
	ZP31 to ZP38	3rd reference position return end signal	F098	○	○	4.5
	ZP41 to ZP48	4th reference position return end signal	F100	○	○	4.5
	ZPX	Spindle orientation completion signal	F094	○	-	9.8
	ZRF1 to ZRF8	Reference position establishment signal	F120	○	○	4.1
	ZRN	Manual reference position return selection signal	G043#7	○	○	4.1
	ZRNO	Software operator's panel signal(ZRN)	F073#4	○	○	12.1.15

A.2.3 List of Signals (In Order of Addresses)

○ : Available
● : Available only with 2–path control
– : Unavailable

Address	Signal name	Symbol	T series	M series	Reference Item
X004#0	Measuring position reached signal	XAE	○	○	14.2
X004#1		YAE	-	○	
X004#1		ZAE	○	-	
X004#2		ZAE	-	○	
X004#2,#4	Manual feed interlock signal for each axis	+MIT1,+MIT2	○	-	2.5
X004#2,#4	Tool offset write signal	+MIT1,+MIT2	○	-	14.4.2
X004#2 to #6,#0,#1	Skip signal	SKIP2 to SKIP6, SKIP7,SKIP8	○	○	14.3.3
X004#3,#5	Manual feed interlock signal for each axis	-MIT1,-MIT2	○	-	14.4.2
X004#3,#5	Tool offset write signal	-MIT1,-MIT2	○	-	14.4.2
X004#6	Skip signal (PMC axis control)	ESKIP	○	○	15.1
X004#7	Skip signal	SKIP	○	○	14.3.1
X004#7	Overload torque signal	SKIP	-	○	11.17
X008#4	Emergency stop signal	*ESP	○	○	2.1
X009	Reference position return deceleration signal	*DEC1 to *DEC8	○	○	4.1
Y(n+0)	Servo motor speed detection signals	DSV1 to DSV8	○	○	2.11
Y(n+1)#0 to #2	Spindle motor speed detection signals	DSP1, DSP2, DSP3	○	○	
G000,G001	Data signal for external data input	ED0 to ED15	○	○	15.2
G002#0 to #6	Address signal for external data input	EA0 to EA6	○	○	15.2
G002#7	Read signal for external data input	ESTB	○	○	15.2
G004#3	End signal	FIN	○	○	8.1
G004#4	2nd M function completion signal	MFIN2	○	○	8.4
G004#5	3rd M function completion signal	MFIN3	○	○	8.4
G005#0	Auxiliary function completion signal	MFIN	○	○	8.4
G005#1	External operation function completion signal	EFIN	-	○	8.4
G005#2	Spindle function completion signal	SFIN	○	○	8.4
G005#3	Tool function completion signal	TFIN	○	○	8.4
G005#4	2nd auxiliary function completion signal	BFIN	○	-	8.4
G005#6	Auxiliary function lock signal	AFL	○	○	8.2
G005#7	2nd auxiliary function completion signal	BFIN	-	○	8.4
G006#0	Program restart signal	SRN	○	○	5.7
G006#2	Manual absolute signal	*ABSM	○	○	5.4
G006#4	Override cancel signal	OVC	○	○	7.1.7.4
G006#6	Skip signal	SKIPP	○	-	14.3.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G007#0	Retrace signal	RVS	-	○	11.15
G007#1	Start lock signal	STLK	○	-	2.5
G007#2	Cycle start signal	ST	○	○	5.1
G007#4	Stroke check 3 release signal	RLSOT3	○	○	2.3.3
G007#5	Follow–up signal	*FLWU	○	○	1.2.7
G007#6	Stored stroke limit select signal	EXLM	○	○	2.3.2
G007#7	Stroke limit release signal	RLSOT	-	○	2.3.2
G008#0	Interlock signal	*IT	○	○	2.5
G008#4	Emergency stop signal	*ESP	○	○	2.1
G008#5	Feed hold signal	*SP	○	○	5.1
G008#6	Reset & rewind signal	RRW	○	○	5.2
G008#7	External reset signal	ERS	○	○	5.2
G009#0 to 4	Workpiece number search signal	PN1,PN2,PN4, PN8,PN16	○	○	15.3
G010,G011	Manual feedrate override signal	*JV0 to *JV15	○	○	3.1
G012	Feedrate override signal	*FV0 to *FV7	○	○	7.1.7.2
G013	2nd feedrate override signal	*AFV0 to *AFV7	○	○	7.1.7.3
G014#0,#1	Rapid traverse override signal	ROV1,ROV2	○	○	7.1.7.1
G016#0	Motor speed detection function enable signal	MSDFON	○	○	2.11
G016#7	F1–digit feed select signal	F1D	-	○	7.1.5
G018#0 to #3	Manual handle feed axis selection signal	HS1A to HS1D	○	○	3.2
G018#4 to #7		HS2A to HS2D	○	○	
G019#0 to #3		HS3A to HS3D	-	○	
G019#4,#5	Manual handle feed amount selection signal (incremental feed signal)	MP1,MP2	○	○	3.2
G019#7	Manual rapid traverse selection signal	RT	○	○	3.1
G023#6	Tool axis perpendicular direction handle feed mode signal	RGHTH	-	○	3.4
G023#7	Tool axis direction handle feed mode signal	ALNGH	-	○	3.4.1
G027#0	Spindle selection signal	SWS1	○	-	9.10
G027#1		SWS2	○	-	
G027#2		SWS3	○	-	
G027#3	Stop signal in each spindle	*SSTP1	○	-	9.10
G027#4		*SSTP2	○	-	
G027#5		*SSTP3	○	-	
G027#7	Cs contour control switch signal	CON	○	○	9.9
G028#1,#2	Gear selection signal (input)	GR1,GR2	○	○	9.3
G028#4	Spindle unclamp completion signal	*SUCPF	○	-	9.8
G028#5	Spindle clamp completion signal	*SCPF	○	-	9.8
G028#6	Spindle stop complete signal	SPSTP	○	-	9.8
G028#7	2nd position coder selection signal	PC2SLC	○	-	9.10

Address	Signal name	Symbol	T series	M series	Reference Item
G029#0	Gear select signal (input)	GR21	○	-	9.10
G029#2		GR31	○	-	
G029#4	Spindle speed arrival signal	SAR	○	○	9.3
G029#5	Spindle orientation signal	SOR	○	○	9.3
G029#6	Spindle stop signal	*SSTP	○	○	9.3
G030	Spindle speed override signal	SOV0 to SOV7	○	○	9.3
G032#0 to G033#3	Spindle motor speed command signal	R01I to R12I	○	○	15.4
G033#5	Spindle motor command polarity select signal	SGN	○	○	15.4
G033#6		SSIN	○	○	
G033#7	Spindle motor speed command select signal	SIND	○	○	15.4
G034#0 to G035#3	Spindle motor speed command signal	R01I2 to R12I2	○	○	15.4
G035#5	Spindle motor command polarity select signal	SGN2	○	○	15.4
G035#6	Spindle motor command polarity select signal	SSIN2	○	○	15.4
G035#7	Spindle motor speed command select signal	SIND2	○	○	15.4
G036#0 to G037#3	Spindle motor speed command signal	R01I3 to R12I3	○	○	15.4
G037#5	Spindle motor command polarity select signal	SGN3	○	○	15.4
G037#6	Spindle motor command polarity select signal	SSIN3	○	○	15.4
G037#7	Spindle motor speed command select signal	SIND3	○	○	15.4
G038#0	Polygon spindle stop signal	*PLSST	○	-	6.10.2
G038#2	Spindle synchronous control signal	SPSYC	○	○	9.12
G038#3	Spindle phase synchronous control signal	SPPHS	○	○	9.12
G038#6	B–axis unclamp completion signal	*BEUCP	-	○	11.11
G038#7	B–axis clamp completion signal	*BECLP	-	○	11.11
G039#0 to #5,G040#0	Tool offset number select signal	OFN0 to OFN5, OFN6	○	-	14.4.2
G039#6	Workpiece coordinate system shift value write mode select signal	WOQSM	○	-	14.4.2
G039#7	Tool offset value write mode select signal	GOQSM	○	-	14.4.2
G040#6	Position record signal	PRC	○	-	14.4.1
G040#7	Workpiece coordinate system shift value write signal	WOSET	○	-	14.4.2
G041#0 to #3	Manual handle interrupt axis selection signal	HS1IA to HS1ID	○	○	3.3
G041#4 to #7		HS2IA to HS2ID	○	○	
G042#0 to #3		HS3IA to HS3ID	-	○	
G042#7	Direct operation select signal	DMMC	○	○	15.6
G043#0 to #2	Mode selection signal	MD1,MD2,MD4	○	○	2.6
G043#5	DNC operation select signal	DNCI	○	○	5.11
G043#7	Manual reference position return selection signal	ZRN	○	○	4.1
G044#0,G045	Optional block skip signal	BDT1, BDT2 to BDT9	○	○	5.5
G044#1	All–axis machine lock signal	MLK	○	○	5.3.1

Address	Signal name	Symbol	T series	M series	Reference Item
G046#1	Single block signal	SBK	○	○	5.3.3
G046#3 to #6	Memory protect signal	KEY1 to KEY4	○	○	12.2.3
G046#7	Dry run signal	DRN	○	○	5.3.2
G047#0 to #6	Tool group number select signal	TL01 to TL64	○	-	10.3
G047#0 to G048#0		TL01 to TL256	-	○	
G048#5	Tool skip signal	TLSKP	○	○	10.3
G048#6	Individual tool change reset signal	TLRSTI	-	○	10.3
G048#7	Tool change reset signal	TLRST	○	○	10.3
G049#0 to G050#1	Tool life count override signal	*TLV0 to *TLV9	-	○	10.3
G051#0 to #3	Chopping feedrate override signals	*CHP8 to *CHP0	–	○	1.12
G051#6	Chopping start signal	CHPST	–	○	1.12
G051#7	Chopping hold signal	*CHLD	–	○	1.12
G052	Input signal for remote buffer	RMTDI0 to RMTDI7	○	○	13.2
G053#0	General–purpose integrating meter start signal	TMRON	○	○	12.1.11
G053#3	Interrupt signal for custom macro	UINT	○	○	11.6.2
G053#6	Error detect signal	SMZ	○	-	7.2.6.3
G053#7	Chamferring signal	CDZ	○	-	11.9
G054,G055	Input signal for custom macro	UI000 to UI015	○	○	11.6.1
G058#0	External start signal program input	MINP	○	○	13.7
G058#1	External read start signal	EXRD	○	○	13.8, 13.5
G058#2	External read/punch stop signal	EXSTP	○	○	13.8, 13.5
G058#3	External punch start signal	EXWT	○	○	13.8, 13.5
G058#5	Input and run simultaneous mode select signal	STRD	-	○	13.6
G058#6	Output and run simultaneous mode select signal	STWD	-	○	13.6
G059#0	Tool retraction signal	TRESC	○	○	5.8
G059#1	Tool return signal	TRRTN	○	○	5.8
G060#7	Tail stock barrier select signal	*TSB	○	-	2.3.4
G061#0	Rigid tapping signal	RGTAP	○	○	9.11
G061#4,#5	Rigid tap spindle select signal	RGTSP1, RGTSP2	○	-	9.11
G062#1	Automatic erase CRT screen display cancel signal	*CRTOF	○	○	12.1.19
G063#0	Path selection signal (Tool post selection signal)	HEAD	●	●	2.7
G063#1	No–wait signal	NOWT	●	●	8.5
G063#2,#3	Spindle command select signal	SLSPA,SLSPB	●	-	9.4
G063#5	Perpendicular/angular axis control disable signal	NOZAGC	○	○	1.11
G064#0	Retry start signal	RTNCY	–	○	5.13
G064#1	Retry point selection signal	SLCSEQ	–	○	5.13
G064#2,#3	Spindle feedback select signal	SLPCA,SLPCB	●	-	9.4
G064#6	Simple spindle synchronous control signal	ESRSYC	○	○	9.16
G066#0	All–axis VRDY OFF alarm ignore signal	IGNVRY	○	○	2.9

Address	Signal name	Symbol	T series	M series	Refer- ence Item
G066#1	External key input mode selection signal	ENBKY	○	○	15.5
G066#4	Retract signal	RTRCT	–	○	1.13
G066#7	Key code read signal	EKSET	○	○	15.5
G070#0	Torque limit command LOW signal (serial spindle)	TLMLA	○	○	9.2
G070#1	Torque limit command HIGH signal (serial spindle)	TLMHA	○	○	9.2
G070#3,#2	Clutch/gear signal (serial spindle)	CTH1A,CTH2A	○	○	9.2
G070#4	CCW command signal (serial spindle)	SRVA	○	○	9.2
G070#5	CW command signal (serial spindle)	SFRA	○	○	9.2
G070#6	Orientation command signal (serial spindle)	ORCMA	○	○	9.2
G070#7	Machine ready completion signal (serial spindle)	MRDYA	○	○	9.2
G071#0	Alarm reset signal (serial spindle)	ARSTA	○	○	9.2
G071#1	Emergency stop signal (serial spindle)	*ESPA	○	○	9.2
G071#2	Spindle select signal (serial spindle)	SPSLA	○	○	9.2
G071#3	Power line switch completion signal (serial spindle)	MCFNA	○	○	9.2
G071#4	Soft start stop cancel signal (serial spindle)	SOCNA	○	○	9.2
G071#5	Signal for controlling velocity integration (serial spindle)	INTGA	○	○	9.2
G071#6	Output switch request signal (serial spindle)	RSLA	○	○	9.2
G071#7	Power line status check signal (serial spindle)	RCHA	○	○	9.2
G072#0	Orientation stop position change signal (serial spindle)	INDXA	○	○	9.2
G072#1	Rotation direction command while changing the orientation stop position signal (serial spindle)	ROTAA	○	○	9.2
G072#2	Short–distant movement command while changing the orientation stop position signal (serial spindle)	NRROA	○	○	9.2
G072#3	Differential mode command signal (serial spindle)	DEFMDA	○	○	9.2
G072#4	Analog override command signal (serial spindle)	OVRIDA	○	○	9.2
G072#5	Incremental command external setting type orientation signal (serial spindle)	INCMDA	○	○	9.2
G072#6	Main spindle MCC status signal while changing spindles signal (serial spindle)	MFNHGA	○	○	9.2
G072#7	High–output MCC status signal while a magnetic sensor (serial spindle)	RCHHGA	○	○	9.2
G073#0	Command for spindle orientaion with a magnetic sensor (serial spindle)	MORCMA	○	○	9.2
G073#1	Slave operation command signal (serial spindle)	SLVA	○	○	9.2
G073#2	Motor power stop signal (serial spindle)	MPOFA	○	○	9.2
G074#0	Torque limit command LOW signal (serial spindle)	TLMLB	○	○	9.2
G074#1	Torque limit command HIGH signal (serial spindle)	TLMHB	○	○	9.2
G074#3,#2	Clutch/gear signal (serial spindle)	CTH1B,CTH2B	○	○	9.2
G074#4	CCW command signal (serial spindle)	SRVB	○	○	9.2
G074#5	CW command signal (serial spindle)	SFRB	○	○	9.2
G074#6	Orientation command signal (serial spindle)	ORCMB	○	○	9.2

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G074#7	Machine ready completion signal (serial spindle)	MRDYB	○	○	9.2
G075#0	Alarm reset signal (serial spindle)	ARSTB	○	○	9.2
G075#1	Emergency stop signal (serial spindle)	*ESPB	○	○	9.2
G075#2	Spindle select signal (serial spindle)	SPSLB	○	○	9.2
G075#3	Power line switch completion signal (serial spindle)	MCFNB	○	○	9.2
G075#4	Soft start stop cancel signal (serial spindle)	SOCNB	○	○	9.2
G075#5	Signal for controlling velocity integration (serial spindle)	INTGB	○	○	9.2
G075#6	Output switch request signal (serial spindle)	RSLB	○	○	9.2
G075#7	Power line status check signal (serial spindle)	RCHB	○	○	9.2
G076#0	Orientation stop position change signal (serial spindle)	INDXB	○	○	9.2
G076#1	Rotation direction command while changing the orientation stop position signal (serial spindle)	ROTAB	○	○	9.2
G076#2	Short–distant movement command while changing the orientation stop position signal (serial spindle)	NRROB	○	○	9.2
G076#3	Differential mode command signal (serial spindle)	DEFMDB	○	○	9.2
G076#4	Analog override command signal (serial spindle)	OVRIDB	○	○	9.2
G076#5	Incremental command external setting type orientation signal (serial spindle)	INCMDB	○	○	9.2
G076#6	Main spindle MCC status signal while changing spindles signal (serial spindle)	MFNHGB	○	○	9.2
G076#7	High–output MCC status signal while a magnetic sensor (serial spindle)	RCHHGB	○	○	9.2
G077#0	Command for spindle orientaion with a magnetic sensor (serial spindle)	MORCMB	○	○	9.2
G077#1	Slave operation command signal (serial spindle)	SLVB	○	○	9.2
G077#2	Motor power stop signal (serial spindle)	MPOFB	○	○	9.2
G078#0 to G079#3	Spindle orientation external stop position command signal	SHA00 to SHA11	○	○	9.13
G080#0 to G081#3		SHB00 to SHB11	○	○	
G086#0 to #3	Feed axis and direction selection signals	+Jg, –Jg, +Ja, –Ja	○	○	3.5
G091#0 to #3	Group number specification signals	SRLNI0 to SRLNI3	○	○	13.8
G092#0	I/O Link confirmation signal	IOLACK	○	○	13.8
G092#1	I/O Link specification signal	IOLS	○	○	13.8
G092#2	Power Mate read/write inprogress signal	BGION	○	○	13.8
G092#3	Power Mate read/write alarm signal	BGIALM	○	○	13.8
G092#4	Power Mate background busy signal	BGEN	○	○	13.8
G096#0 to #6	1% step rapid traverse override signal	*HROV0 to *HROV6	○	○	7.1.7.1
G096#7	1% step rapid traverse override select signal	HROV	○	○	7.1.7.1
G098	Key code signal	EKC0 to EKC7	○	○	15.5
G100	Feed axis and direction selection signal	+J1 to +J8	○	○	3.1
G102		-J1 to -J8	○	○	
G106	Mirror image signal	MI1 to MI8	○	○	1.2.6

Address	Signal name	Symbol	T series	M series	Reference Item
G108	Each–axis machine lock signal	MLK1 to MLK8	○	○	5.3.1
G110	Stroke limit external setting signal	+LM1 to +LM8	-	○	2.3.2
G112		-LM1 to -LM8	-	○	
G114	Overtravel signal	*+L1 to *+L8	○	○	2.3.1
G116		*-L1 to *-L8	○	○	
G118	External deceleration signal	*+ED1 to *+ED8	○	○	7.1.9
G120		*-ED1 to *-ED8	○	○	
G122	Parking signals	PK1 to PK8	○	-	1.8
G122#0 to #6		PK1 to PK7	●	–	1.9
G122#6 (G031#6)	First spindle synchronous control signal	PKESS1	○	○	9.16
G122#7 (G031#7)	Second spindle synchronous control signal	PKESS2	○	○	9.16
G124	Controlled axis detach signal	DTCH1 to DTCH8	○	○	1.2.4
G126	Servo off signal	SVF1 to SVF8	○	○	1.2.8
G128#0 to #6	Composite control axis selection signals	MIX1 to MIX7	●	–	1.9
G130	Interlock signal for each axis	*IT1 to *IT8	○	○	2.5
G132#0 to #3	Interlock signal for each axis and direction	+MIT1 to +MIT4	-	○	2.5
G134#0 to #3		-MIT1 to -MIT4	-	○	
G136	Control axis select signal (PMC axis control)	EAX1 to EAX8	○	○	15.1
G138	Simple synchronous axis select signal	SYNC1 to SYNC8	○	○	1.6
G140	Simple synchronous manual feed axis select signal	SYNCJ1 to SYNCJ8	-	○	1.6
G138	Synchronous control axis selection signals	SYNC to SYNC8	○	-	1.8
G138#0 to #6		SYNC to SYNC7	●	-	1.9
G142#0	Auxiliary function completion signal (PMC axis control)	EFINA	○	○	15.1
G142#2	Buffering disable signal (PMC axis control)	EMBUFA	○	○	15.1
G142#3	Block stop signal (PMC axis control)	ESBKA	○	○	15.1
G142#4	Servo off signal (PMC axis control)	ESOFA	○	○	15.1
G142#5	Axis control temporary stop signal (PMC axis control)	ESTPA	○	○	15.1
G142#6	Reset signal (PMC axis control)	ECLRA	○	○	15.1
G142#7	Axis control command read signal (PMC axis control)	EBUFA	○	○	15.1
G143#0 to #6	Axis control command signal (PMC axis control)	EC0A to EC6A	○	○	15.1
G143#7	Block stop disable signal (PMC axis control)	EMSBKA	○	○	15.1
G144,G145	Axis control feedrate signal (PMC axis control)	EIF0A to EIF15A	○	○	15.1
G146 to G149	Axis control data signal (PMC axis control)	EID0A to EID31A	○	○	15.1
G150#0,#1	Rapid traverse override signal (PMC axis control)	ROV1E,ROV2E	○	○	15.1
G150#5	Override cancel signal (PMC axis control)	OVCE	○	○	15.1
G150#6	Manual rapid traverse selection signal (PMC axis control)	RTE	○	○	15.1
G150#7	Dry run signal (PMC axis control)	DRNE	○	○	15.1
G151	Feedrate override signal (PMC axis control)	*FV0E to *FV7E	○	○	15.1
G154#0	Auxiliary function completion signal (PMC axis control)	EFINB	○	○	15.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G154#2	Buffering disable signal (PMC axis control)	EMBUFB	○	○	15.1
G154#3	Block stop signal (PMC axis control)	ESBKB	○	○	15.1
G154#4	Servo off signal (PMC axis control)	ESOFB	○	○	15.1
G154#5	Axis control temporary stop signal (PMC axis control)	ESTPB	○	○	15.1
G154#6	Reset signal (PMC axis control)	ECLRB	○	○	15.1
G154#7	Axis control command read signal (PMC axis control)	EBUFB	○	○	15.1
G155#0 to #6	Axis control command signal (PMC axis control)	EC0B to EC6B	○	○	15.1
G155#7	Block stop disable signal (PMC axis control)	EMSBKB	○	○	15.1
G156,G157	Axis control feedrate signal (PMC axis control)	EIF0B to EIF15B	○	○	15.1
G158 to G161	Axis control data signal (PMC axis control)	EID0B to EID31B	○	○	15.1
G166#0	Auxiliary function completion signal (PMC axis control)	EFINC	○	○	15.1
G166#2	Buffering disable signal (PMC axis control)	EMBUFC	○	○	15.1
G166#3	Block stop signal (PMC axis control)	ESBKC	○	○	15.1
G166#4	Servo off signal (PMC axis control)	ESOFC	○	○	15.1
G166#5	Axis control temporary stop signal (PMC axis control)	ESTPC	○	○	15.1
G166#6	Reset signal (PMC axis control)	ECLRC	○	○	15.1
G166#7	Axis control command read signal (PMC axis control)	EBUFC	○	○	15.1
G167#0 to #6	Axis control command signal (PMC axis control)	EC0C to EC6C	○	○	15.1
G167#7	Block stop disable signal (PMC axis control)	EMSBKC	○	○	15.1
G168,G169	Axis control feedrate signal (PMC axis control)	EIF0C to EIF15C	○	○	15.1
G170 to G173	Axis control data signal (PMC axis control)	EID0C to EID31C	○	○	15.1
G178#0	Auxiliary function completion signal (PMC axis control)	EFIND	○	○	15.1
G178#2	Buffering disable signal (PMC axis control)	EMBUFD	○	○	15.1
G178#3	Block stop signal (PMC axis control)	ESBKD	○	○	15.1
G178#4	Servo off signal (PMC axis control)	ESOFD	○	○	15.1
G178#5	Axis control temporary stop signal (PMC axis control)	ESTPD	○	○	15.1
G178#6	Reset signal (PMC axis control)	ECLRD	○	○	15.1
G178#7	Axis control command read signal (PMC axis control)	EBUFD	○	○	15.1
G179#0 to #6	Axis control command signal (PMC axis control)	EC0D to EC6D	○	○	15.1
G179#7	Block stop disable signal (PMC axis control)	EMSBKD	○	○	15.1
G180,G181	Axis control feedrate signal (PMC axis control)	EIF0D to EIF15D	○	○	15.1
G182 to G185	Axis control data signal (PMC axis control)	EID0D to EID31D	○	○	15.1
G190#0 to #6	Superimposed control axis selection signals	OVLS1 to OVLS7	●	-	1.9
G192	Each–axis VRDY OFF alarm ignore signal	IGVRY1 to IGVRY8	○	○	2.9
G198	Position display neglect signal	NPOS1 to NPOS8	○	○	12.1.10
G204#0	Torque limit command LOW signal (serial spindle)	TLMLC	○	○	9.15
G204#1	Torque limit command HIGH signal (serial spindle)	TLMHC	○	○	9.15
G204#3,#2	Clutch/gear signal (serial spindle)	CTH1C,CTH2C	○	○	9.15
G204#4	CCW command signal (serial spindle)	SRVC	○	○	9.15

Address	Signal name	Symbol	T series	M series	Refer-ence Item
G204#5	CW command signal (serial spindle)	SFRC	○	○	9.15
G204#6	Orientation command signal (serial spindle)	ORCMC	○	○	9.15
G204#7	Machine ready signal (serial spindle)	MRDYC	○	○	9.15
G205#0	Alarm reset signal (serial spindle)	ARSTC	○	○	9.15
G205#1	Emergency stop signal (serial spindle)	*ESPC	○	○	9.15
G205#2	Spindle select signal (serial spindle)	SPSLC	○	○	9.15
G205#3	Power line switch completion signal (serial spindle)	MCFNC	○	○	9.15
G205#4	Soft start/stop cancel signal (serial spindle)	SOCNC	○	○	9.15
G205#5	Signal for controlling velocity integration (serial spindle)	INTGC	○	○	9.15
G205#6	Output switch request signal (serial spindle)	RSLC	○	○	9.15
G205#7	Power line status check signal (serial spindle)	RCHC	○	○	9.15
G206#0	Orientation stop position change signal (serial spindle)	INDXC	○	○	9.15
G206#1	Rotation direction command while changing the orientation stop position signal (serial spindle)	ROTAC	○	○	9.15
G206#2	Short–distant movement command while changing the orientation stop position signal (serial spindle)	NRROC	○	○	9.15
G206#3	Differential mode command signal (serial spindle)	DEFMDC	○	○	9.15
G206#4	Analog override command signal (serial spindle)	OVRIDC	○	○	9.15
G206#5	Incremental command external setting type orientation signal (serial spindle)	INCMDC	○	○	9.15
G206#6	Main spindle MCC status signal while changing spindles signal (serial spindle)	MFNHGC	○	○	9.15
G206#7	High–output MCC status signal while a magnetic sensor (serial spindle)	RCHHGC	○	○	9.15
G207#0	Command for spindle orientaion with a magnetic sensor (serial spindle)	MORCMC	○	○	9.15
G207#1	Slave operation command signal (serial spindle)	SLVC	○	○	9.15
G207#2	Motor power stop signal (serial spindle)	MPOFC	○	○	9.15
G208#0 to G209#3	Spindle orientation stop position external command signal	SHC00 to SHC11	○	○	9.15
F000#0	Rewinding signal	RWD	○	○	5.2
F000#4	Feed hold lamp signal	SPL	○	○	5.1
F000#5	Cycle start lamp signal	STL	○	○	5.1
F000#6	Servo ready completion signal	SA	○	○	2.2
F000#7	Automatic operation signal	OP	○	○	5.1
F001#0	Alarm signal	AL	○	○	2.4
F001#1	Resetting signal	RST	○	○	5.2
F001#2	Battery alarm signal	BAL	○	○	2.4
F001#3	Distribution end signal	DEN	○	○	8.1
F001#4	Spindle enable signal	ENB	○	○	9.3
F001#5	Tapping signal	TAP	○	○	11.7
F001#7	CNC signal	MA	○	○	2.2

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F002#0	Inch input signal	INCH	○	○	11.4
F002#1	Rapid traversing signal	RPDO	○	○	2.8
F002#2	Constant surface speed signal	CSS	○	○	9.5
F002#3	Thread cutting signal	THRD	○	○	6.4.1
F002#4	Program restart under way signal	SRNMV	○	○	5.7
F002#6	Cutting feed signal	CUT	○	○	2.8
F002#7	Dry run check signal	MDRN	○	○	5.3.2
F003#0	Incremental feed select check signal	MINC	○	○	2.6
F003#1	Manual handle feed select check signal	MH	○	○	2.6
F003#2	Jog feed select check signal	MJ	○	○	2.6
F003#3	Manual data input select check signal	MMDI	○	○	2.6
F003#4	DNC operation selection confirm signal	MRMT	○	○	5.11
F003#5	Automatic operation select check signal	MMEM	○	○	2.6
F003#6	Memory edit select check signal	MEDT	○	○	2.6
F003#7	TEACH IN select check signal	MTCHIN	○	○	2.6
F004#0,F005	Optional block skip check signal	MBDT1,MBDT2 to MBDT9	○	○	5.5
F004#1	All–axis machine lock check signal	MMLK	○	○	5.3.1
F004#2	Manual absolute check signal	MABSM	○	○	5.4
F004#3	Single block check signal	MSBK	○	○	5.3.3
F004#4	Auxiliary function lock check signal	MAFL	○	○	8.2
F004#5	Manual reference position return selection check signal	MREF	○	○	4.1
F007#0	Miscellaneous function strobe signal	MF	○	○	8.1
F007#1	Externaloperation signal for high–speed interface	EFD	-	○	8.4
F007#2	Spindle–speed function strobe signal	SF	○	○	8.1
F007#3	Tool function strobe signal	TF	○	○	8.1
F007#4	2nd auxiliary function strobe signal	BF	○	-	8.1
F007#7		BF	-	○	
F008#0	External operation signal	EF	-	○	11.8
F008#4	2nd M function strobe signal	MF2	○	○	8.3
F008#5	3rd M function strobe signal	MF3	○	○	8.3
F009#4	Decode M signal	DM30	○	○	8.1
F009#5		DM02	○	○	
F009#6		DM01	○	○	
F009#7		DM00	○	○	
F010 to F013	Miscellaneous function code signal	M00 to M31	○	○	8.1
F014 to F015	2nd M function code signal	M200 to M215	○	○	8.3
F016 to F017	3rd M function code signal	M300 to M315	○	○	8.3
F022 to F025	Spindle speed code signal	S00 to S31	○	○	8.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F026 to F029	Tool function code signal	T00 to T31	○	○	8.1
F030 to F033	2nd miscellaneous function code signal	B00 to B31	○	○	8.1
F034#0 to #2	Gear selection signal (output)	GR1O,GR2O, GR3O	-	○	9.3
F035#0	Spindle fluctuation detection alarm signal	SPAL	○	○	9.6
F036#0 to F037#3S	12–bit code signal	R01O to R12O	○	○	9.3
F038#0	Spindle clamp signal	SCLP	○	-	9.8
F038#1	Spindle unclamp signal	SUCLP	○	-	9.8
F038#2	Spindle enable signal	ENB2	○	-	9.10
F038#3		ENB3	○	-	
F039#2	Chopping–in–progress signal	CHPMD	–	○	1.12
F039#3	Chopping cycle signal	CHPCYL	–	○	1.12
F040,F041	Actual spindle speed signal	AR0 to AR15	○	-	9.7
F044#1	Cs contour control switch completion signal	FSCSL	○	○	9.9
F044#2	Spindle synchronous speed control completion signal	FSPSY	○	○	9.12
F044#3	Spindle phase synchronous control completion signal	FSPPH	○	○	9.12
F044#4	Spindle synchronous control alarm signal	SYCAL	○	○	9.12
F044#4	Phase error monitor signal	SYCAL	○	○	9.16
F045#0	Alarm signal (serial spindle)	ALMA	○	○	9.2
F045#1	Speed zero signal (serial spindle)	SSTA	○	○	9.2
F045#2	Speed detection signal (serial spindle)	SDTA	○	○	9.2
F045#3	Speed arrival signal (serial spindle)	SARA	○	○	9.2
F045#4	Load detection signal 1 (serial spindle)	LDT1A	○	○	9.2
F045#5	Load detection signal 2 (serial spindle)	LDT2A	○	○	9.2
F045#6	Torque limit signal (serial spindle)	TLMA	○	○	9.2
F045#7	Orientation completion signal (serial spindle)	ORARA	○	○	9.2
F046#0	Power line switch signal (serial spindle)	CHPA	○	○	9.2
F046#1	Spindle switch completion signal (serial spindle)	CFINA	○	○	9.2
F046#2	Output switch signal (serial spindle)	RCHPA	○	○	9.2
F046#3	Output switch completion signal (serial spindle)	RCFNA	○	○	9.2
F046#4	Slave operation status signal (serial spindle)	SLVSA	○	○	9.2
F046#5	Signal for approximate spindle orientation with a position coder (serial spindle)	PORA2A	○	○	9.2
F046#6	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	MORA1A	○	○	9.2
F046#7	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	MORA2A	○	○	9.2
F047#0	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	PC1DEA	○	○	9.2
F047#1	Incremental method orientation signal (serial spindle)	INCSTA	○	○	9.25
F049#0	Alarm signal (serial spindle)	ALMB	○	○	9.2

Address	Signal name	Symbol	T series	M series	Refer- ence Item
F049#1	Speed zero signal (serial spindle)	SSTB	○	○	9.2
F049#2	Speed detection signal (serial spindle)	SDTB	○	○	9.2
F049#3	Speed arrival signal (serial spindle)	SARB	○	○	9.2
F049#4	Load detection signal 1 (serial spindle)	LDT1B	○	○	9.2
F049#5	Load detection signal 2 (serial spindle)	LDT2B	○	○	9.2
F049#6	Torque limit signal (serial spindle)	TLMB	○	○	9.2
F049#7	Orientation completion signal (serial spindle)	ORARB	○	○	9.2
F050#0	Power line switch signal (serial spindle)	CHPB	○	○	9.2
F050#1	Spindle switch completion signal (serial spindle)	CFINB	○	○	9.2
F050#2	Output switch signal (serial spindle)	RCHPB	○	○	9.2
F050#3	Output switch completion signal (serial spindle)	RCFNB	○	○	9.2
F050#4	Slave operation status signal (serial spindle)	SLVSB	○	○	9.2
F050#5	Signal for approximate spindle orientation with a position coder (serial spindle)	PORA2B	○	○	9.2
F050#6	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	MORA1B	○	○	9.2
F050#7	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	MORA2B	○	○	9.2
F051#0	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	PC1DEB	○	○	9.2
F051#1	Incremental method orientation signal (serial spindle)	INCSTB	○	○	9.25
F053#0	Key input disable signal	INHKY	○	○	15.5
F053#1	Program screen display mode signal	PRGDPL	○	○	15.5
F053#2	Read/punch in–progress signal	RPBSY	○	○	13.8, 13.5
F053#3	Read/punch alarm signal	RPALM	○	○	13.8, 13.5
F053#4	Background busy signal	BGEACT	○	○	13.8, 13.5
F053#7	Key code read completion signal	EKENB	○	○	15.5
F054,F055	Output signal for custom macro	UO000 to UO015	○	○	11.6.1
F056 to F059		UO100 to UO131	○	○	
F060#0	Read completion signal for external data input	EREND	○	○	15.2
F060#1	Search completion signal for external data input	ESEND	○	○	15.2
F061#0	B–axis unclamp signal	BUCLP	-	○	11.11
F061#1	B–axis clamp signal	BCLP	-	○	11.11
F062#7	Target part count reached signal	PRTSF	○	○	12.1.11
F063#0	Master axis not arrival signal	PSE1	○	-	6.10.2
F063#1	Polygon synchronous axis not arrival signal	PSE2	○	-	6.10.2
F063#2	Spindle polygon speed arrival signal	PSAR	○	-	6.10.2
F063#5	Retry complete signal	RCYO	–	○	5.13
F063#6	Waiting signal	WATO	●	●	8.5
F063#7	Polygon synchronization under way signal	PSYN	○	-	6.10.1

Address	Signal name	Symbol	T series	M series	Reference Item
F064#0	Tool change signal	TLCH	○	○	10.3
F064#1	New tool select signal	TLNW	○	○	10.3
F064#2	Individual tool change signal	TLCHI	-	○	10.3
F064#5	Spindle command signal	COSP	●	-	9.4
F064#6	Tool post interference check signal	TICHK	●	-	2.3.5
F064#7	tool post interference alarm signal	TIALM	●	-	2.3.5
F065#0	Spindle rotation direction signal	RGSPP	-	○	9.11
F065#1		RGSPM	-	○	
F065#4	Retract completion signal	RTRCTF	–	○	1.13
F065#6	EGB mode signal	SYNMOD	–	○	1.13
F066#0	Lock–ahead control mode signal	G08MD	-	○	7.1.13
F066#3	Retry point signal	RTNMVS	–	○	5.13
F066#5	Small–diameter peck drilling in progress signal	PECK2	-	○	11.17
F066#6	HPCC mode signal	MHPCC	-	○	7.1.14
F066#7	HPCC operation signal	EXHPCC	-	○	7.1.14
F069	Output signal for remote buffer	RMTDO0 to RMTDO7	○	○	13.2
F070#0 to F071#1	Position switch signal	PSW01 to PSW10	○	○	1.2.9
F072	Software operator's panelgeneral–purpose switch signal	OUT0 to OUT7	○	○	12.1.15
F073#0	Software operator's panel signal (MD1)	MD1O	○	○	12.1.15
F073#1	Software operator's panel signal (MD2)	MD2O	○	○	12.1.15
F073#2	Software operator's panel signal (MD4)	MD4O	○	○	12.1.15
F073#4	Software operator's panel signal (ZRN)	ZRNO	○	○	12.1.15
F075#2	Software operator's panel signal (BDT)	BDTO	○	○	12.1.15
F075#3	Software operator's panel signal (SBK)	SBKO	○	○	12.1.15
F075#4	Software operator's panel signal (MLK)	MLKO	○	○	12.1.15
F075#5	Software operator's panel signal (DRN)	DRNO	○	○	12.1.15
F075#6	Software operator's panel signal (KEY1 to KEY4)	KEYO	○	○	12.1.15
F075#7	Software operator's panel signal (*SP)	SPO	○	○	12.1.15
F076#0	Software operator's panel signal (MP1)	MP1O	○	○	12.1.15
F076#1	Software operator's panel signal (MP2)	MP2O	○	○	12.1.15
F076#3	Rigid tapping mode signal	RTAP	○	○	9.11
F076#4	Software operator's panel signal (ROV1)	ROV1O	○	○	12.1.15
F076#5	Software operator's panel signal (ROV2)	ROV2O	○	○	12.1.15
F077#0	Software operator's panel signal (HS1A)	HS1AO	○	○	12.1.15
F077#1	Software operator's panel signal (HS1B)	HS1BO	○	○	12.1.15
F077#2	Software operator's panel signal (HS1C)	HS1CO	○	○	12.1.15
F077#3	Software operator's panel signal (HS1D)	HS1DO	○	○	12.1.15
F077#6	Software operator's panel signal (RT)	RTO	○	○	12.1.15

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F078	Software operator's panel signal (*FV0 to *FV7)	*FV0O to *FV7O	○	○	12.1.15
F079,F080	Software operator's panel signal (*JV0 to *JV15)	*JV0O to *JV15O	○	○	12.1.15
F081#0,#2, #4,#6	Software operator's panel signal (+J1 to +J4)	+J1O to +J4O	○	○	12.1.15
F081#1,#3, #5,#7	Software operator's panel signal (-J1 to -J4)	-J1O to -J4O	○	○	12.1.15
F082#2	Retrace–in–progress signal	RVSL	-	○	11.15
F090#0	Servo axis abnormal load detected signal	ABTQSV	○	○	2.10
F090#1	First–spindle abnormal load detected signal	ABTSP1	○	○	2.10
F090#2	Second–spindle abnormal load detected signal	ABTSP2	○	○	2.10
F090#3	Abnormal load detection for the third spindle	ABTSP3	○	○	9.15
F092#3	Tool retraction mode signal	TRACT	○	○	5.8
F092#5	Tool return completion signal	TRSPS	○	○	5.8
F094	Reference position return end signal	ZP1 to ZP8	○	○	4.1
	Spindle orientation completion signal	ZPX	○	-	9.8
F096	2nd reference position return end signal	ZP21 to ZP28	○	○	4.5
F098	3rd reference position return end signal	ZP31 to ZP38	○	○	4.5
F100	4th reference position return end signal	ZP41 to ZP48	○	○	4.5
F102	Axis moving signal	MV1 to MV8	○	○	1.2.5
F104	In–position signal	INP1 to INP8	○	○	7.2.6.1
F106	Axis moving direction signal	MVD1 to MVD8	○	○	1.2.5
F108	Mirror image check signal	MMI1 to MMI8	○	○	1.2.6
F110	Controlled axis detach status signal	MDTCH1 to MDTCH8	○	○	1.2.4
F112	Distribution completion signal (PMC axis control)	EADEN1 to EADEN8	○	○	15.1
F114	Torque limit reached signal	TRQL1 to TRQL8	○	-	14.3.4
F116	Floating reference position return end signal	FRP1 to FRP8	○	○	4.6
F118	Synchronous control under way signals	SYN1O to SYN8O	○	-	1.8
F118#0 to #6	Synchronous/composite/superimposed control under way signals	SYN1O to SYN7O	●	-	1.9
F120	Reference position establishment signal	ZRF1 to ZRF8	○	○	4.1
F122	High–speed skip status signal	HDO0 to HDO7	○	○	14.3.2
F129#5	Override 0% signal (PMC axis control)	EOV0	○	○	15.1
F129#7	Control axis selection status signal (PMC axis control)	*EAXSL	○	○	15.1
F130#0	In–position signal (PMC axis control)	EINPA	○	○	15.1
F130#1	Following zero checking signal (PMC axis control)	ECKZA	○	○	15.1
F130#2	Alarm signal (PMC axis control)	EIALA	○	○	15.1
F130#3	Auxiliary function executing signal (PMC axis control)	EDENA	○	○	15.1
F130#4	Axis moving signal (PMC axis control)	EGENA	○	○	15.1
F130#5	Positive–direction overtravel signal (PMC axis control)	EOTPA	○	○	15.1
F130#6	Negative–direction overtravel signal (PMC axis control)	EOTNA	○	○	15.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F130#7	Axis control command read completion signal (PMC axis control)	EBSYA	○	○	15.1
F131#0	Auxiliary function strobe signal (PMC axis control)	EMFA	○	○	15.1
F131#1	Bufferful signal (PMC axis control)	EABUFA	○	○	15.1
F132,F142	Auxiliary function code signal (PMC axis control)	EM11A to EM48A	○	○	15.1
F133#0	In–position signal (PMC axis control)	EINPB	○	○	15.1
F133#1	Following zero checking signal (PMC axis control)	ECKZB	○	○	15.1
F133#2	Alarm signal (PMC axis control)	EIALB	○	○	15.1
F133#3	Auxiliary function executing signal (PMC axis control)	EDENB	○	○	15.1
F133#4	Axis moving signal (PMC axis control)	EGENB	○	○	15.1
F133#5	Positive–direction overtravel signal (PMC axis control)	EOTPB	○	○	15.1
F133#6	Negative–direction overtravel signal (PMC axis control)	EOTNB	○	○	15.1
F133#7	Axis control command read completion signal (PMC axis control)	EBSYB	○	○	15.1
F134#0	Auxiliary function strobe signal (PMC axis control)	EMFB	○	○	15.1
F134#1	Bufferful signal (PMC axis control)	EABUFB	○	○	15.1
F135,F145	Auxiliary function code signal (PMC axis control)	EM11B to EM48B	○	○	15.1
F136#0	In–position signal (PMC axis control)	EINPC	○	○	15.1
F136#1	Following zero checking signal (PMC axis control)	ECKZC	○	○	15.1
F136#2	Alarm signal (PMC axis control)	EIALC	○	○	15.1
F136#3	Auxiliary function executing signal (PMC axis control)	EDENC	○	○	15.1
F136#4	Axis moving signal (PMC axis control)	EGENC	○	○	15.1
F136#5	Positive–direction overtravel signal (PMC axis control)	EOTPC	○	○	15.1
F136#6	Negative–direction overtravel signal (PMC axis control)	EOTNC	○	○	15.1
F136#7	Axis control command read completion signal (PMC axis control)	EBSYC	○	○	15.1
F137#0	Auxiliary function strobe signal (PMC axis control)	EMFC	○	○	15.1
F137#1	Buffer full signal (PMC axis control)	EABUFC	○	○	15.1
F138,F148	Auxiliary function code signal (PMC axis control)	EM11C to EM48C	○	○	15.1
F139#0	In–position signal (PMC axis control)	EINPD	○	○	15.1
F139#1	Following zero checking signal (PMC axis control)	ECKZD	○	○	15.1
F139#2	Alarm signal (PMC axis control)	EIALD	○	○	15.1
F139#3	Auxiliary function executing signal (PMC axis control)	EDEND	○	○	15.1
F139#4	Axis moving signal (PMC axis control)	EGEND	○	○	15.1
F139#5	Positive–direction overtravel signal (PMC axis control)	EOTPD	○	○	15.1
F139#6	Negative–direction overtravel signal (PMC axis control)	EOTND	○	○	15.1
F139#7	Axis control command read completion signal (PMC axis control)	EBSYD	○	○	15.1
F140#0	Auxiliary function strobe signal (PMC axis control)	EMFD	○	○	15.1
F140#1	Buffer full signal (PMC axis control)	EABUFD	○	○	15.1

Address	Signal name	Symbol	T series	M series	Refer-ence Item
F141,F151	Auxiliary function code signal (PMC axis control)	EM11D to EM48D	○	○	15.1
F168#0	Alarm signal (serial spindle)	ALMC	○	○	9.15
F168#1	Speed zero signal (serial spindle)	SSTC	○	○	9.15
F168#2	Speed detection signal (serial spindle)	SDTC	○	○	9.15
F168#3	Speed arrival signal (serial spindle)	SARC	○	○	9.15
F168#4	Load detection signal 1 (serial spindle)	LDT1C	○	○	9.15
F168#5	Load detection signal 2 (serial spindle)	LDT2C	○	○	9.15
F168#6	Torque limit signal (serial spindle)	TLMC	○	○	9.15
F168#7	Orientation completion signal (serial spindle)	ORARC	○	○	9.15
F269#0	Power line switch signal (serial spindle)	CHPC	○	○	9.15
F169#1	Spindle switch completion signal (serial spindle)	CFINC	○	○	9.15
F169#2	Output switch signal (serial spindle)	RCHPC	○	○	9.15
F169#3	Output switch completion signal (serial spindle)	RCFNC	○	○	9.15
F169#4	Slave operation status signal (serial spindle)	SLVSC	○	○	9.15
F169#5	Signal for approximate spindle orientation with a position coder (serial spindle)	PORA2C	○	○	9.15
F169#6	Signal for completion of spindle orientation with a magnetic sensor (serial spindle)	MORA1C	○	○	9.15
F169#7	SIgnal for approximate spindle orientation with a magnetic sensor (serial spindle)	MORA2C	○	○	9.15
F170#0	Signal indicating the status of the detected one–rotation position coder signal (serial spindle)	PC1DEC	○	○	9.15
F170#1	Incremental method orientation signal (serial spindle)	INCSTC	○	○	9.15
F177#0	Slave I/O Link selection signal	IOLNK	○	○	13.8
F177#1	Slave external read start signal	ERDIO	○	○	13.8
F177#2	Slave read/write stop signal	ESTPIO	○	○	13.8
F177#3	Slave external write start signal	EWTIO	○	○	13.8
F177#4	Slave program selection signal	EPRG	○	○	13.8
F177#5	Slave macro variable selection signal	EVAR	○	○	13.8
F177#6	Slave parameter selection signal	EPARM	○	○	13.8
F177#7	Slave diagnosis selection signal	EDGN	○	○	13.8
F178#0 to #3	Group number output signals	SRLNO0 to SRLNO3	○	○	13.8
F180	Torque limit reach signals for butt–type reference position setting	CLRCH1 to CLRCH8	○	○	4.7
F182	Controlling signal (PMC axis control)	EACNT1 to EACNT8	○	○	15.1

Index

≪Numbers≫

≪A≫

≪B≫

≪C≫

≪D≫

≪E≫

≪F≫

≪G≫

≪H≫

≪I≫

≪J≫

≪L≫

≪M≫

≪T≫

≪U≫

≪V≫

≪W≫

Revision Record

FANUC Series 16*i*/18*i*/21*i*/160*i*/180*i*/210*i*–MODEL A CONNECTION MANUAL (FUNCTION) (B–63003EN–1)

Edition	Date	Contents	Edition	Date	Contents
01	Apr., ’97				