Changes for the Better

MITSUBISHI ELECTRIC

MITSUBISHI CNC

EZMotion-NC E60/E68 Series

SPECIFICATIONS MANUAL

IB-1500171(ENG)-B

Introduction

This manual describes the specifications of EZMotion-NC E60/E68 series.
To safely use this CNC unit, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual

At the beginning of each item, a table indicating its specification according to the model.

○ : Standard
△ : Additional hardware is required separately.
□ : Selection
☆ : Special option

⚠ CAUTION

⚠ The items that are not described in this manual must be interpreted as "not possible".
⚠ This manual is written on the assumption that all option functions are added.
⚠ Some functions may differ or some functions may not be usable depending on the NC system (software) version.

General precautions

(1) When the contents of this manual is updated, the version (A, B, …) on the cover will be incremented.

Precautions for Safety

Always read the specifications issued by the machine maker, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
Understand this numerical controller, safety items and cautions before using the unit.
This manual ranks the safety precautions into "DANGER", "WARNING" and "CAUTION".

DANGER	When there is a great risk that the user could be subject to fatalities or serious injuries if handling is mistaken.
WARNING	When the user could be subject to fatalities or serious injuries if handling is mistaken.
CAUTION	When the user could be subject to injuries or when physical damage could occur if handling is mistaken.

Note that even items ranked as " **CAUTION**", may lead to major results depending on the situation. In any case, important information that must always be observed is described.

DANGER

Not applicable in this manual.

WARNING

Not applicable in this manual.

CAUTION

1. Items related to product and manual

- The items that are not described in this manual must be interpreted as "not possible".
- This manual is written on the assumption that all option functions are added.
- Some functions may differ or some functions may not be usable depending on the NC system (software) version.

2. Items related to start up and maintenance

- Follow the power specifications (input voltage range, frequency range, momentary power failure time range) described in this manual.
- Follow the environment conditions (ambient temperature, humidity, vibration, atmosphere) described in this manual.
- Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)
- If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

CONTENTS

EZMotion-NC E60/E68 Series Specifications List

1. Control Axes

The NC axis, spindle, PLC axis are generically called the control axis.
The NC axis is an axis that can be manually operated, or automatically operated with the machining program.
The PLC axis is an axis that can be controlled from the PLC ladder.

1.1 Control Axes

1.1.1 Number of Basic Control Axes (NC axes)

	E60	E68
M system	○3	○3
L system	○2	○2

1.1.2 Max. Number of Control Axes (NC axes + Spindles + PLC axes + Auxiliary axes)

	E60	E68
M system	5	8
L system	5	8

A number of axes that are within the maximum number of control axes, and that does not exceed the maximum number given for the NC axis, spindle, PLC axis and auxiliary axis can be used.

Connection specifications of NC axis, PLC axis, spindle and auxiliary axis
There are two channels with which the servo and spindle are connected.
Maximum 5 axes (for E60) or 6 axes (for E68) can be connected with the channel 1.

NC axis, PLC axis, spindle : They can be connected with the channel 1. Connection Nos. for the 1st to 5th axis (for E60) or 1st to 6th axis (for E68) are assigned to each channel. Connect them from the first axis in order. More than one axis must be connected with the channel 1.

Auxiliary axis (J2CT) : They can be connected with the channel 2.

Max. number of axes (NC axes + spindles + PLC axes)

	E60	E68
M system	5	6
L system	5	6

Max. number of servo axes (NC axes + PLC axes)

	E60	E68
M system	4	6
L system	4	6

Max. number of NC axes (in total for all the part systems)

	E60	E68
M system	3	4
L system	3	4

Max. number of spindles
Includes analog spindles.

	E60	E68
M system	1	2
L system	1	2

Max. number of PLC axes

	E60	E68
M system	1	2
L system	1	2

Max. number of auxiliary axes (MR-J2-CT)

	E60	E68
M system	1	4
L system	1	4

1.1.3 Number of Simultaneous Contouring Control Axes

	E60	E68
M system	3	4
L system	3	4

1.1.4 Max. Number of NC Axes in a Part System

	E60	E68
M system	3	4
L system	3	4

1.2 Control Part System

1.2.1 Standard Number of Part Systems

	E60	E68
M system	1	1
L system	1	1

1.2.2 Max. Number of Part Systems

	E60	E68
M system	○1	○1
L system	○1	○1

For actual use, the machine maker specification will apply.

1.3 Control Axes and Operation Modes

1.3.1 Tape (RS-232C Input) Mode

	E60	E68
M system	○	○
L system	○	○

In this mode, operation is performed using the machining program data from the RS-232C interface built in the NC unit. A paper tape reader must be provided if machining programs on paper tape are to be run.

1.3.2 Memory

	E60	E68
M system	○	○
L system	○	○

The machining programs stored in the memory of the NC unit are run.

1.3.3 MDI

	E60	E68
M system	○	○
L system	○	○

The MDI data stored in the memory of the NC unit is executed. Once executed, the MDI data is set to the "setting incomplete" status, and the data will not be executed unless the "setting completed" status is established by performing screen operations.

1.3.5 IC Card Mode

1.3.5.2 Front IC Card Operation

	E60	E68
M system	–	○
L system	–	–

The IC card operation function enables a machining program registered in the IC card to be called and operated. With IC card, machining program can be edited on PC.
During IC card operation, a machining program in the NC memory can be called as the sub-program by M98 command. Also, a machining program in the IC card can be called from the main program in the memory as the sub-program by M198 command and operated.
It is recommended to use the SanDisk CF card.

(Note) When inserting/removing a commercially available CF card, preferably, turn the MITSUBISHI device's power OFF to avoid any troubles. When inserting/removing a card while the power is ON, make sure to have sufficient time (approx. ten seconds or more) in between.

2. Input Command

2.1 Data Increment

Least command increment: 1 μm

	E60	E68
M system	○	○
L system	○	○

Least command increment: 0.1 μm

	E60	E68
M system	–	○
L system	–	○

The data increment handled in the controller include the least setting increment, least command increment and least detection increment. Each type is set with parameters.

(1) The least setting increment indicates the increment handled in the internal processing of the controller. The counter and tool offset data, etc., input from the screen is handled with this increment.

(2) The least command increment indicates the command increment of the movement command in the machining program. This can be set per axis.

(a) For E60

Increment type	**Type**	**Metric unit system**		**Inch unit system**	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least command increment	B	0.001	0.001	0.0001	0.001

(Note) The inch and metric systems cannot be used together.

(b) For E68

Increment type	**Type**	**Metric unit system**		**Inch unit system**	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least command increment	B	0.001	0.001	0.0001	0.001
	C	0.0001	0.0001	0.00001	0.0001

(Note) The inch and metric systems cannot be used together.

(3) The least detection increment indicates the detection increment of the NC axis and PLC axis detectors. The increment is determined by the detector being used.

(4) If the least command increment is 0.1μm/0.01μm, the movement amount, movement range and command speed may be limited compared to the 1μm.

2.2 Unit System

2.2.1 Inch/Metric Changeover

	E60	E68
M system	○	○
L system	○	○

The unit systems of the data handled in the controller include the metric unit system and inch unit system. The type can be designated with the parameters and machining program. The unit system can be set independently for the (1) Program command, (2) Setting data such as offset amount and (3) Parameters.

Unit system	Length data	Meaning
Metric unit system	1.0	1.0 mm
Inch unit system	1.0	1.0 inch

(Note) For the angle data, 1.0 means 1 degree (°) regardless of the unit system.

<table>
<tr><th colspan="2">Data
Parameter</th><th colspan="2">Machining program</th><th>Screen data
(Offset amount, etc.)</th><th colspan="2">Parameter</th></tr>
<tr><td rowspan="4">I_inch</td><td rowspan="2">0</td><td>G20</td><td>Inch unit system</td><td rowspan="2">Metric unit system</td><td rowspan="4" colspan="2">Not affected</td></tr>
<tr><td>G21</td><td>Metric unit system</td></tr>
<tr><td rowspan="2">1</td><td>G20</td><td>Inch unit system</td><td rowspan="2">Inch unit system</td></tr>
<tr><td>G21</td><td>Metric unit system</td></tr>
<tr><td rowspan="2">M_inch</td><td>0</td><td rowspan="2" colspan="2">Not affected</td><td rowspan="2">Not affected</td><td colspan="2">Metric unit system</td></tr>
<tr><td>1</td><td colspan="2">Inch unit system</td></tr>
</table>

(Note 1) The parameter changeover is valid after the power is turned ON again.
(Note 2) Even if parameter "I_inch" is changed, the screen data (offset amount, etc.) will not be automatically converted.
(Note 3) When the power is turned ON or resetting is performed, the status of the G20/G21 modal depends on the "I_G20" parameter setting.

2.3 Program Format

2.3.1 Character Code

	E60	E68
M system	○	○
L system	○	○

The command information used in this CNC system consists of alphanumerics and symbols which are collectively known as characters.
These characters are expressed as combinations of 8-bit data inside the NC unit.
The expressions formed in this way are called codes, and this CNC system uses shift JIS codes.
The characters which are valid in this CNC system are listed below.

Character	Remarks
0 to 9	Always significant
A to Z	Always significant
+	Always significant
–	Always significant
.	Always significant
,	Always significant
/	Always significant
%	Always significant
CR	Always significant
LF/NL	Always significant
(	Always significant
)	Always significant
:	Always significant
#	Always significant
*	Always significant
=	Always significant
[	Always significant
]	Always significant
SP	Always significant
!	Always significant
$	Always significant
BS	An error results during operation (except when the character is part of a comment) .
HT	An error results during operation (except when the character is part of a comment) .
&	An error results during operation (except when the character is part of a comment) .
'(Apostrophe)	An error results during operation (except when the character is part of a comment) .
;	An error results during operation (except when the character is part of a comment) .
<	An error results during operation (except when the character is part of a comment) .
>	An error results during operation (except when the character is part of a comment) .
?	An error results during operation (except when the character is part of a comment) .
@	An error results during operation (except when the character is part of a comment) .
"	An error results during operation (except when the character is part of a comment) .
DEL	Always ignored
NULL	Always ignored

2.3.2 Program Format

2.3.2.1 Format 1 for Lathe

	E60	E68
M system	–	–
L system	○	○

The G-code of L system is selected by parameter.
This specification manual explains the G function with G-code series 3 as standard.

2.3.2.3 Special Format for Lathe

	E60	E68
M system	–	–
L system	○	○

2.3.2.4 Format 1 for Machining Center

	E60	E68
M system	○	○
L system	–	–

2.4 Command Value

2.4.1 Decimal Point Input I, II

	E60	E68
M system	○	○
L system	○	○

There are two types of the decimal point input commands and they can be selected by parameter.

(1) Decimal point input type I (When parameter #1078 Decpt2 is 0.)

When axis coordinates and other data are supplied in machining program commands, the assignment of the program data can be simplified by using the decimal point input. The minimum digit of a command not using a decimal point is the same as the least command unit.
Usable addresses can be applied not only to axis coordinate values but also to speed commands and dwell commands.
The decimal point position serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in a time designation of dwell command.

(2) Decimal point input type II (When parameter #1078 Decpt2 is 1.)

As opposed to type I, when there is no decimal point, the final digit serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in the time designation.
The "." (point) must be added when commands below the decimal point are required.

	Unit interpretation (for metric system)	
	Type I	**Type II**
G00 X100. Y-200.5	X100mm, Y-200.5mm	←
G1 X100 F20.	X100µm, F20mm/min	X100mm, F20mm/min
Y200 F100 (*1)	Y200µm, F100mm/min	Y200mm, F100mm/min
G4 X1.5	Dwell 1.5 s	←
X2	2ms	2s

(*1) The F unit is mm/min for either type (inch system : inch/min).

2.4.2 Absolute / Incremental Command

	E60	E68
M system	○	○
L system	○	○

(1) M system

When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be selected.
The absolute and incremental commands can be both used in one block, and are switched with G90 or G91. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

G90 ... Absolute command (absolute value command)
G91 ... Incremental command (incremental value command)

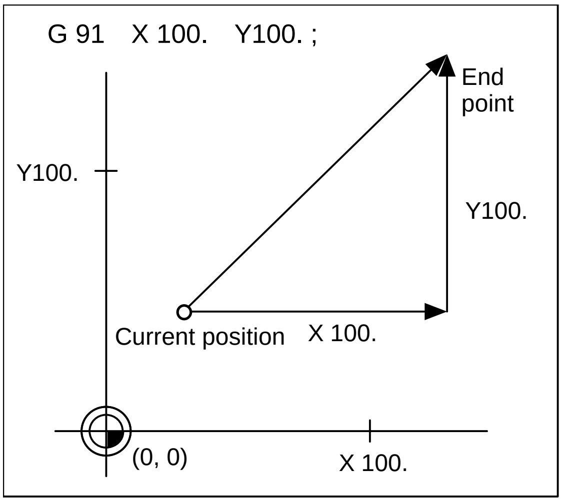
These G codes can be commanded multiple times in one block.

Example

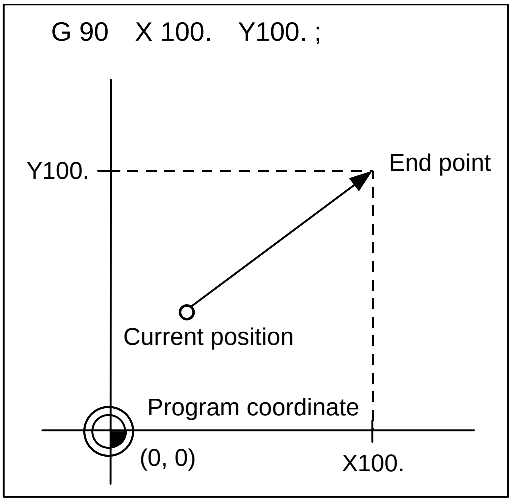
G90 X100.	G91 Y200.	G90 Z300.	;
Absolute value	Incremental value	Absolute value	

(Note 1) As with the memory command, if there is no G90/G91 designation in the MDI command, the previously executed modal will be followed.

(Incremental value command)



(Absolute value command)



(2) L system

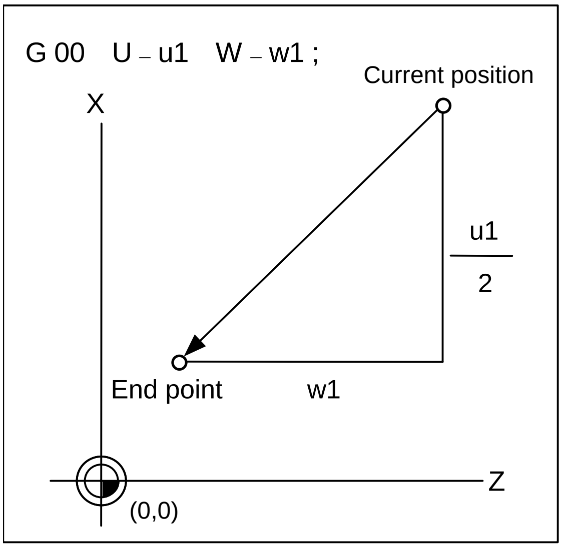
When axis coordinate data is issued in a machining program command, either the incremental command method that commands a relative distance from the current position or the absolute command method that moves to a designated position in a predetermined coordinate system can be selected.
When issuing an incremental value command, the axis address to be commanded as the incremental axis name is registered in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

Absolute command (absolute value command) ... X, Z
Incremental command (incremental value command) ... U, W

Example

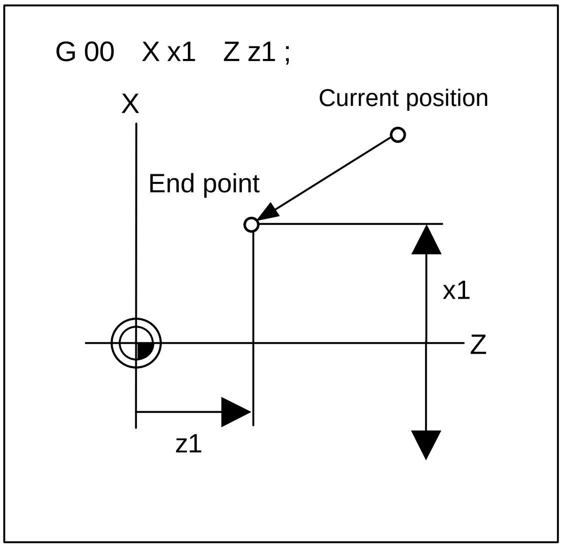
G00	X100.	W200.	;
	Absolute value	Incremental value	

(Incremental value command)



The above drawing shows the case for the diameter command.

(Absolute value command)



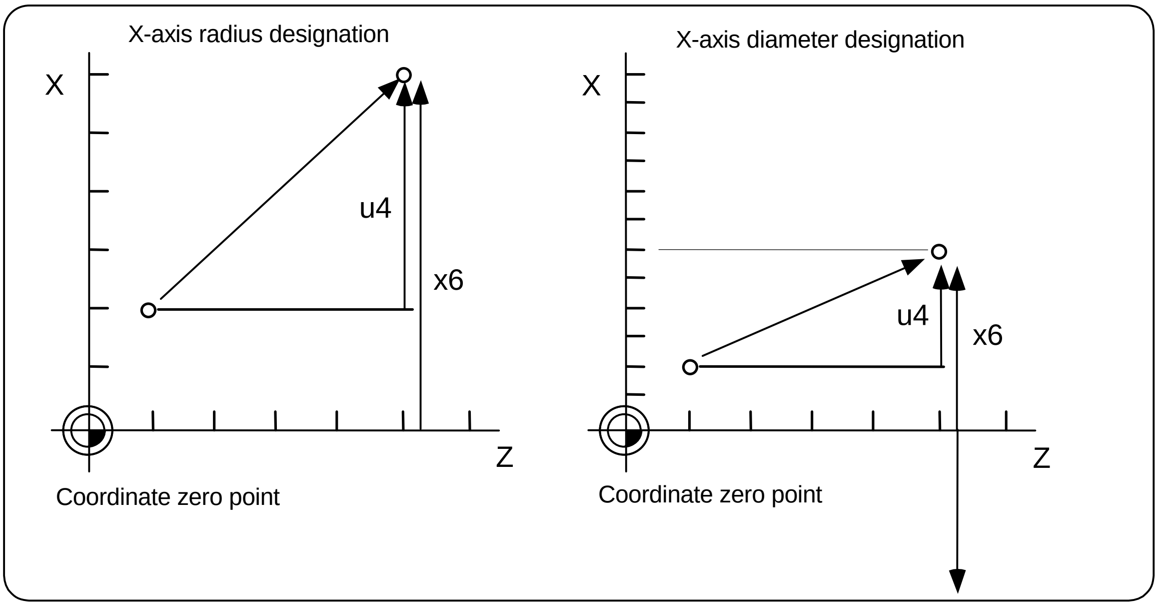
The above drawing shows the case for the diameter command.

(Note) In addition to the above command method using axis addresses, to switch the absolute value command/incremental value command, the command method using G90/91 can also be selected using a parameter.

2.4.3 Diameter/Radius Designation

	E60	E68
M system	–	–
L system	○	○

For command value, the radius designation or diameter designation can be changed over with parameters.
When the diameter designation is selected, the scale of the length of the selected axis is doubled. (For instance, an actual length of 1 mm will be treated as 2 mm.)
This function is used when programming the work dimensions on a lathe as diameters. Changing over from the diameter designation to the radius designation or vice versa can be set separately for each axis.



The difference in the diameter designation and radius designation is shown below.

Absolute value command		**Incremental value command**	
Radius designation	**Diameter designation**	**Radius designation**	**Diameter designation**
Actual movement amount = x1	Actual movement amount = 2 x1	Actual movement amount = u1	Actual movement amount = 2 u1

2.5 Command Value and Setting Value Range

2.5.1 Command Value and Setting Value Range

	E60	E68
M system	○	○
L system	○	○

<Brief summary of format details>

		M system			
		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44 α+44	X+53 Y+53 Z+53 α+53	X+53 Y+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Y+44 Z+44 α+44	X+35 Y+35 Z+35 α+35	X+44 Y+44 Z+44 α+44	X+44 Y+44 Z+44 α+44
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35 (Note 5)
Dwell	0.001(°) mm/ 0.0001 inch	X+53 P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution)	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution) (Note 6)
	0.0001 (°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
Tool offset		H3 D3	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←

		L system			
		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44 α+44	X+53 Y+53 Z+53 α+53	X+53 Y+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Z+44 α+44	X+35 Z+35 α+35	X+44 Z+44 α+44	X+44 Z+44 α+44
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44 K+44 R+44	I+35 K+35 R+35	I+44 K+44 R+44	I+35 K+35 R+35 (Note 5)
Dwell	0.001(°) mm/ 0.0001 inch	X+53 P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F35(Feed per revolution)	F44(Feed per minute) F26(Feed per revolution)	F63(Feed per minute) F34(Feed per revolution)	F44(Feed per minute) F26(Feed per revolution) (Note 6)
	0.0001(°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
Tool offset		T1/T2	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←

(Note 1) α indicates the additional axis address, such as A, B or C.

(Note 2) The No. of digits check for a word is carried out with the maximum number of digits of that address.

(Note 3) Numerals can be used without the leading zeros.

(Note 4) The meanings of the details are as follows :
Example 1 : 08 : 8-digit program number
Example 2 : G21 : Dimension G is 2 digits to the left of the decimal point, and 1 digit to the right.
Example 3 : X+53 : Dimension X uses + or - sign and represents 5 digits to the left of the decimal point and 3 digits to the right.
For example, the case for when the X axis is positioned (G00) to the 45.123 mm position in the absolute value (G90) mode is as follows :

G00 X45.123 ;
123 — 3 digits below the decimal point
X45 — 5 digits above the decimal point, so it's +00045, but the leading zeros and the mark (+) have been omitted.
G00 — G0 is possible.

(Note 5) If an arc is commanded using a rotary axis and linear axis while inch commands are being used, the degrees will be converted into 0.1 inches for interpolation.

(Note 6) While inch commands are being used, the rotary axis speed will be in increments of 10 degrees.
Example: With the F1. (per-minute-feed) command, this will become the 10 degrees/minute command.

(Note 7) The decimal places below the decimal point are ignored when a command, such as an S command, with an invalid decimal point has been assigned with a decimal point.

(Note 8) This format is the same for the value input from the memory, MDI or setting and display unit.

(Note 9) Command the program No. in an independent block. Command the program NO. in the head block of the program.

(Note 10) Least input setting increment "C" (0.0001(°)mm/0.00001inch) and dwell's U address are specifications for E68 system. These cannot be used with E60.

<List of Command Value and Setting Value Ranges>

	Linear axis		Rotary axis
	Input unit: mm	**Input unit: inch**	**Degree (°)**
Minimum input setting unit	0.001 0.0001	0.0001 0.00001	0.001 0.0001
Maximum stroke (Value on machine coordinate system)	±99999.999 mm ±9999.9999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 °
Maximum command value	±99999.999 mm ±9999.9999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 °
Rapid traverse rate (Including during dry run)	1 to 1000000 mm/min 1 to 100000 mm/min	1 to 39370 inch/min 1 to 3937 inch/min	1 to 1000000 °/min 1 to 100000 °/min
M system cutting feed rate (Including during dry run)	0.01 to 1000000 mm/min 0.001 to 100000 mm/min	0.001 to 39370 inch/min 0.0001 to 3937 inch/min	0.01 to 1000000 °/min 0.001 to 100000 °/min
L system cutting feed rate (Including during dry run)	0.001 to 1000000 mm/min 0.0001 to 100000 mm/min	0.0001 to 39370.0787 inch/min 0.00001 to 3937.00787 inch/min	0.001 to 1000000 °/min 0.0001 to 100000 °/min
M system synchronous feed	0.001 to 999.999 mm/rev 0.0001 to 99.9999 mm/rev	0.0001 to 999.9999 inch/rev 0.00001 to 99.99999 inch/rev	0.01 to 999.99 °/rev 0.001 to 99.999 °/rev
L system synchronous feed	0.0001 to 999.9999 mm/rev 0.00001 to 99. 99999 mm/rev	0.000001 to 99.999999 inch/rev 0.0000001 to 9.9999999 inch/rev	0.0001 to 999.9999 °/rev 0.00001 to 99.99999 °/rev
2nd to 4th reference point offset (value on machine coordinate system)	±99999.999 mm ±9999.9999 mm	±9999.9999 inch ±999.99999 inch	±99999.999 ° ±9999.9999 °
Tool offset amount (shape)	±999.999 mm ±99.9999 mm	±99.9999 inch ±9.99999 inch	
Tool offset amount (wear)	±99.999 mm ±9.9999 mm	±9.9999 inch ±0.99999 inch	
Incremental feed amount	0.001 mm/pulse 0.0001 mm/pulse	0.0001 inch/pulse 0.00001 inch/pulse	0.001 °/pulse 0.0001 °/pulse
Handle feed amount	0.001 mm/pulse 0.0001 mm/pulse	0.0001 inch/pulse 0.00001 inch/pulse	0.001 °/pulse 0.0001 °/pulse
Soft limit range (value on machine coordinate system)	–99999.999 to +99999.999 mm –9999.9999 to +9999. 9999 mm	–9999.9999 to +9999.9999 inch –999.99999 to +999.99999 inch	1 to 359.999 ° 1 to 359.9999 °
Dwell time	0 to 99999.999 s	0 to 99999.999 s	
Backlash compensation amount	0 to ±9999 pulse	0 to ±9999 pulse	0 to ±9999 pulse
Pitch error compensation	0 to ±9999 pulse	0 to ±9999 pulse	0 to ±9999 pulse
M system thread lead (F)	0.001 to 999.999 mm/rev 0.0001 to 99.9999 mm/rev	0.0001 to 39.3700 inch/rev 0.00001 to 3.93700 inch/rev	
M system thread lead (Precise E)	0.00001 to 999.99999 mm/rev 0.000001 to 99.999999 mm/rev	0.000001 to 39.370078 inch/rev 0.000001 to 3.937007 inch/rev	
L system thread lead (F)	0.0001 to 999.9999 mm/rev 0.00001 to 99.99999 mm/rev	0.000001 to 99.999999 inch/rev 0.0000001 to 9.9999999 inch/rev	
L system thread lead (Precise E)	0.00001 to 999.99999 mm/rev 0.000001 to 99.999999 mm/rev	0.000010 to 9.9999999 inch/rev 0.0000010 to 0.99999999 inch/rev	

(Note 1) The second line in the table applies when the least setting increment is 0.001, 0.0001 from the first line.

(Note 2) Least input setting increment "C" (0.0001(°)mm/0.00001inch) is a specification for E68 system. This cannot be used with E60.

3. Positioning / Interpolation

3.1 Positioning

3.1.1 Positioning

	E60	E68
M system	○	○
L system	○	○

This function carries out positioning at high speed using rapid traverse with the movement command value given in the program.

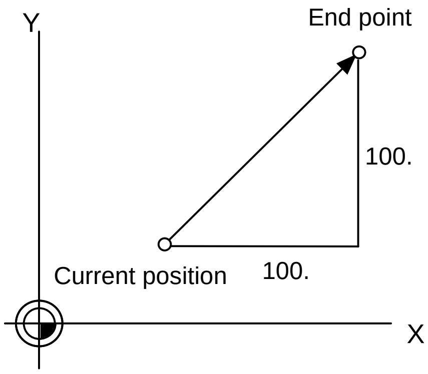
G00 Xx1 Yy1 Zz1 ··· (NC axis) ;
(x1, y1, z1: numerical values denoting the position data)

The above command positions the tool by rapid traverse. The tool path takes the shortest distance to the end point in the form of a straight line.
For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned.

(1) The rapid traverse feed rate for each axis can be set independently with parameters.
(2) The number of axes that can be driven simultaneously depends on the specifications (number of simultaneously controlled axes). The axes can be used in any combination within this range.
(3) The feed rate is controlled within the range that it does not exceed the rapid traverse rate of each axis and so that the shortest time is taken. (Linear type)
Parameter(#1086 G00 non-interpolation) setting enables movement at the rapid traverse rates of the respective axes independently for each axis. In this case, the tool path does not take the form of a straight line to the end point. (Non-Linear type)

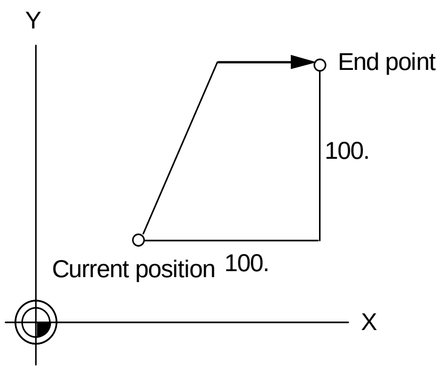
(Example) Linear type (Moves lineary to the end point.)

G 00 G 91 X 100. Y 100. ;



(Example) Non-linear type (Each axis moves with each parameter speed.)

G 00 G 91 X 100. Y 100. ;



(4) The tool is always accelerated at the start of the program command block and decelerated at the end of the block.

3.1.2 Unidirectional Positioning

	E60	E68
M system	○	○
L system	–	–

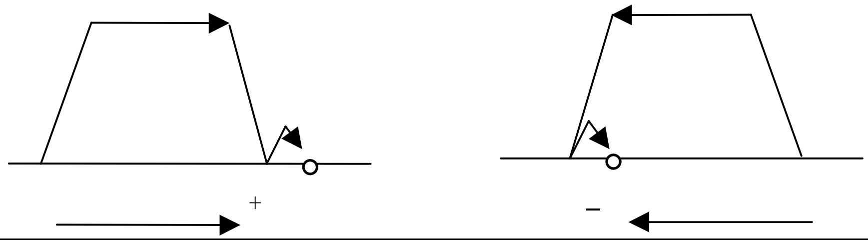
The G60 command always moves the tool to the final position in a direction determined with parameters. The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ··· (NC axis) ;

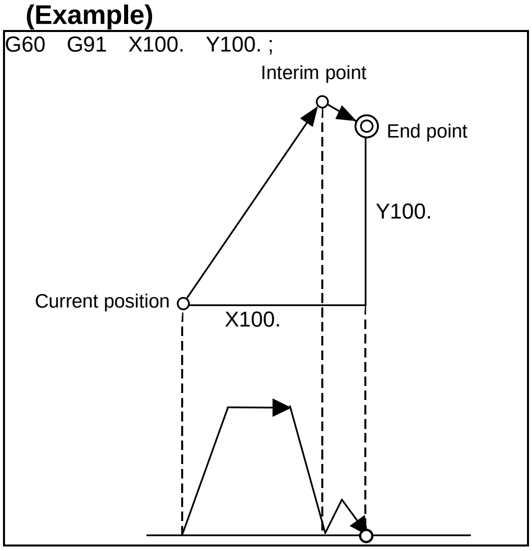
(x1, y1, z1: numerical values denoting the position data)

With the above command, the tool is first moved to a position distanced from the end point position by an amount equivalent to the creep distance (parameter setting) and then moved to its final position.
For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate". Since the actual rapid traverse feed rate depends on the machine, refer to the specifications of the machine concerned.

Positioning to the final point is shown below (when this positioning is in the "+" direction.)



(Example)



1. The rapid traverse rate for each axis is the value set with parameters as the G00 speed.
2. The vector speed to the interim point is the value produced by combining the distance and respective speeds.
3. The creep distance of the distance between the interim and end points can be set independently for each axis by "parameters".

(Note 1) The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel.

(Note 2) The creep distance is moved with rapid traverse.

(Note 3) G60 is valid even for positioning in drilling in the fixed cycle

(Note 4) When the mirror image function is on, the tool will be moved in the reverse direction by mirror image as far as the interim position, but operation over the creep distance with the final advance will not be affected by the mirror image.

3.2 Linear / Circular Interpolation

3.2.1 Linear Interpolation

	E60	E68
M system	○	○
L system	○	○

Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code.

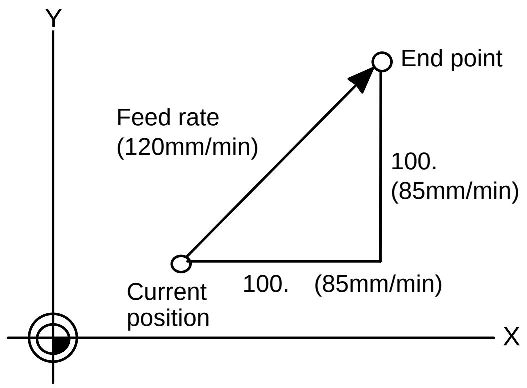
G01 Xx1 Yy1 Zz1 ··· (NC axis) Ff1 ;

(x1, y1, z1 : numerical values denoting the position data;
f1 : numerical value denoting the feed rate data)

Linear interpolation is executed by the above command at the f1 feed rate. The tool path takes the shortest distance to the end point in the form of a straight line.
For details on the f1 command values for NC, refer to the section entitled "Cutting Feed Rate". Since the actual cutting feed rate depends on the machine, refer to the specifications of the machine concerned.

(Example)

G01 G91 X100. Y100. F120 ;



1. The cutting feed rate command moves the tool in the vector direction.
2. The component speeds of each axis are determined by the proportion of respective command values to the actual movement distance with linear interpolation.

(1) The number of axes that can be driven simultaneously depends on the specifications (number of simultaneously controlled axes). The axes can be used in any combination within this range.
(2) The feed rate is controlled so that it does not exceed the cutting feed rate clamp of each axis.
(3) When a rotary axis has been commanded in the same block, it is treated as a linear axis in degree(°) units (1° = 1mm), and linear interpolation is performed.

3.2.2 Circular Interpolation (Center / Radius Designation)

	E60	E68
M system	○	○
L system	○	○

(1) Circular interpolation with I, J, K commands

This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command value supplied in the program.

G02(G03) Xx1 Yy1 (NC axis) Ii1 Jj1 Ff1 ;

G02, G03 : Arc rotation direction
Xx1, Yy1 : End point coordinate values
Ii1, Jj1 : Arc center values
Ff1 : Feed rate

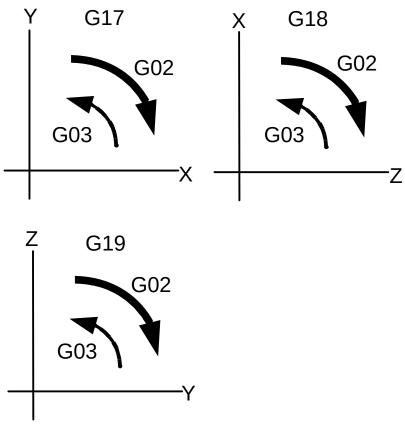
The above commands move the tool along the circular arc at the f1 feed rate. The tool moves along a circular path, whose center is the position from the start point designated by distance "i1" in the X-axis direction and distance "j1" in the Y-axis direction, toward the end point.

The direction of the arc rotation is specified by G02 or G03.

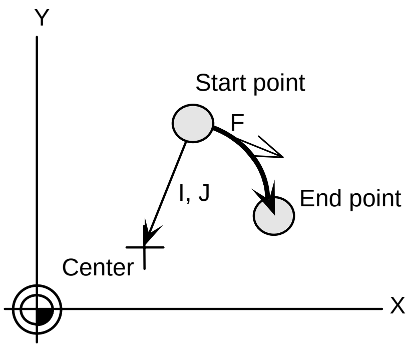
G02: Clockwise (CW)
G03: Counterclockwise (CCW)

The plane is selected by G17, G18 or G19.

G17: XY plane
G18: ZX plane
G19: YZ plane



(Example) See below for examples of circular commands.



(a) The axes that can be commanded simultaneously are the two axes for the selected plane.
(b) The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.
(c) Circular interpolation can be commanded within a range extending from 0° to 360°.
(d) The max. value of the radius can be set up to six digits above the decimal point.

(Note 1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur.

(Note 2) The axes configuring a plane can be designated by parameters. Refer to the section entitled "Plane Selection".

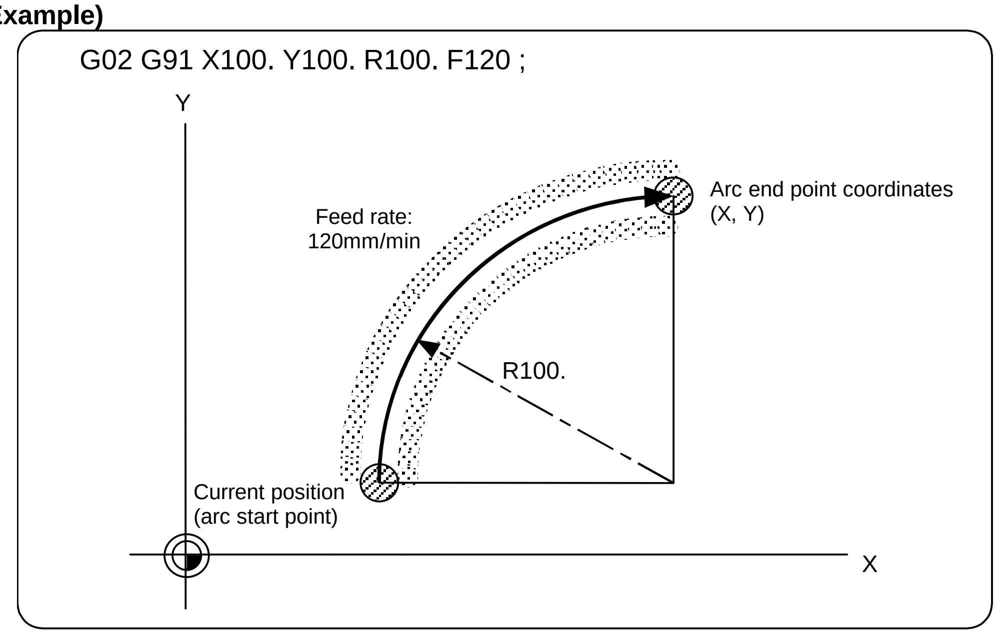
(2) R-specified circular interpolation

Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

G02(G03) Xx1 Yy1 (NC axis) Rr1 Ff1 ;	
G02, G03	: Arc rotation direction
Xx1, Yy1	: End point coordinate values
Rr1	: Arc radius
Ff1	: Feed rate

G02 or G03 is used to designate the direction of the arc rotation.
The arc plane is designated by G17, G18 or G19.
The arc center is on the bisector which orthogonally intersects the segment connecting the start and end points, and the point of intersection with the circle, whose radius has been designated with the start point serving as the center, is the center coordinate of the arc command.
When the sign of the value of R in the command program is positive, the command will be for an arc of 180° or less; when it is negative, it will be for an arc exceeding 180°.

(Example)



(a) The axes that can be commanded simultaneously are the two axes for the selected plane.
(b) The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

(Note 1) The arc plane is always based on the G17, G18 or G19 command. If a command is issued with two addresses which do not match the plane, an alarm will occur.

3.2.3 Helical Interpolation

	E60	E68
M system	○	○
L system	–	○

With this function, any two of three axes intersecting orthogonally are made to perform circular interpolation while the third axis performs linear interpolation in synchronization with the arc rotation. This simultaneous 3-axis control can be exercised to machine large-diameter screws or 3-dimensional cams.

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Pp1 Ff1 ;

G17	: Arc plane
G02, G03	: Arc rotation direction
Xx1, Yy1	: End point coordinate values for arc
Zz1	: End point coordinate of linear axis
Ii1, Jj1	: Arc center coordinates
Pp1	: Pitch No.
Ff1	: Feed rate

(1) The arc plane is designated by G17, G18 or G19.
(2) G02 or G03 is used to designate the direction of the arc rotation.
(3) Absolute or incremental values can be assigned for the arc end point coordinates and the end point coordinates of the linear axis, but incremental values must be assigned for the arc center coordinates.
(4) The linear interpolation axis is the other axis which is not included in the plane selection.
(5) Command the speed in the component direction that represents all the axes combined for the feed rate.

Pitch l1 is obtained by the formula below.

$$l1 = z1/((2\pi \cdot p1 + \theta)/2\pi)$$

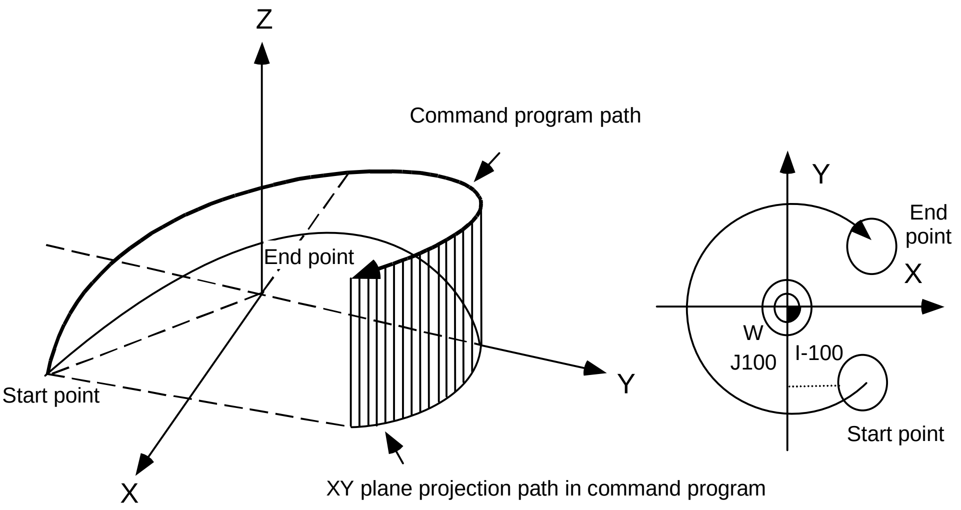
$$\theta = \theta e - \theta s = \arctan(ye/xe) - \arctan(ys/xs)$$

Where xs, ys are the start point coordinates ($0 \le \theta < 2\pi$)
xe, ye are the end point coordinates

The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination within this range. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

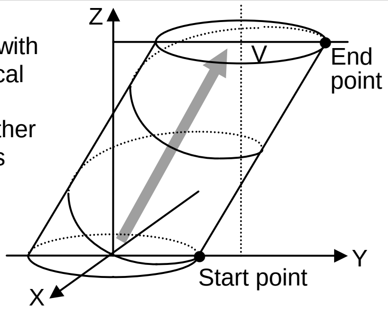
(Example)

G91 G17 G02 X0. Y200. Z100. I–100. J100. F120 ;



(Note 1) Helical sharps are machined by assigning linear commands for one axis which is not a circular interpolation axis using an orthogonal coordinate system. It is also possible to assign these commands to two or more axes which are not circular interpolation axes.

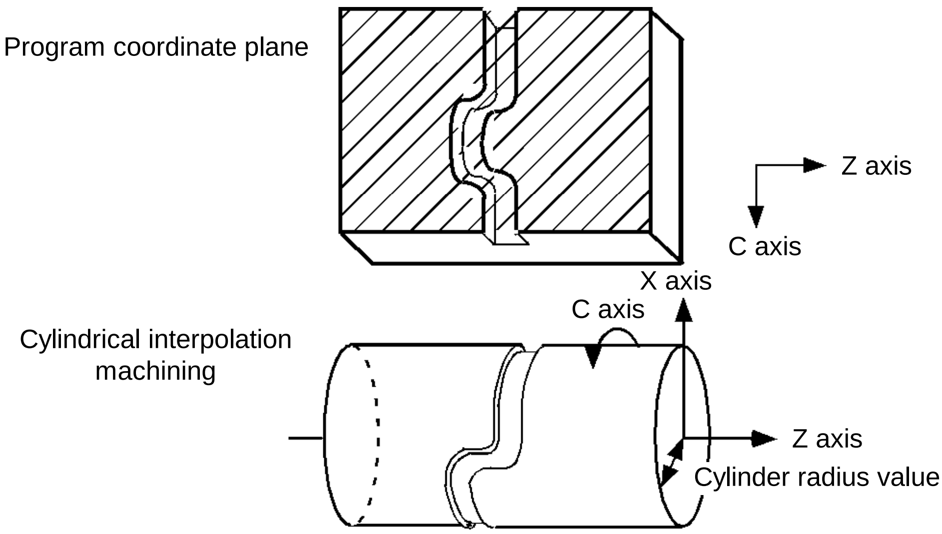
When a simultaneous 4-axis command is used with the V axis as the axis parallel to the Y axis, helical interpolation will result for a cylinder which is inclined as shown in the figure on the right. In other words, linear interpolation of the Z and V axes is carried out in synchronization with the circular interpolation on the XY plane.



3.2.5 Cylindrical Interpolation

	E60	E68
M system	–	○
L system	–	○

This function transfers the shape that is on the cylinder's side surface (shape yielded by the cylindrical coordinate system) onto a plane, and when the transferred shape is designated in the program in the form of plane coordinates, the shape is converted into a movement along the linear and rotary axes of the original cylinder coordinates, and the contours are controlled by means of the CNC unit during machining.
Since the programming can be performed for the shapes produced by transferring the side surfaces of the cylinders, this function is useful when it comes to machining cylindrical cams and other such parts.



(1) Cylindrical interpolation mode start
(G07.1 name of rotary axis cylinder radius value;)

Cylindrical interpolation is performed between the rotary axis designated in the G07.1 block and any other linear axis.

(a) Linear interpolation or circular interpolation can be designated in the cylindrical interpolation mode. However, assign the G19 command (plane selection command) immediately before the G07.1 block.
(b) The coordinates can be designated either in absolute values or incremental values.
(c) Tool radius compensation can be applied to the program commands. Cylindrical interpolation is performed for the path after tool radius compensation.
(d) For the feed rate, designate a tangential rate over the cylinder transfer surface using the F command.
The F rate is in either mm/min or inch/mm units.

(2) Cylindrical interpolation mode cancel
(G07.1 name of rotary axis 0;)
If "C" is the name of the rotary axis, the cylindrical interpolation cancel mode is established with the command below.
G07.1 C0 ;

3.2.6 Polar Coordinate Interpolation

	E60	E68
M system	–	–
L system	–	O

This function converts the commands programmed by the orthogonal coordinate axes into rectilinear axis movements (tool movements) and rotary axis movements (workpiece rotation) to control the contours. It is useful for cutting linear cutouts on the outside diameter of the workpiece, grinding cam shafts, etc.

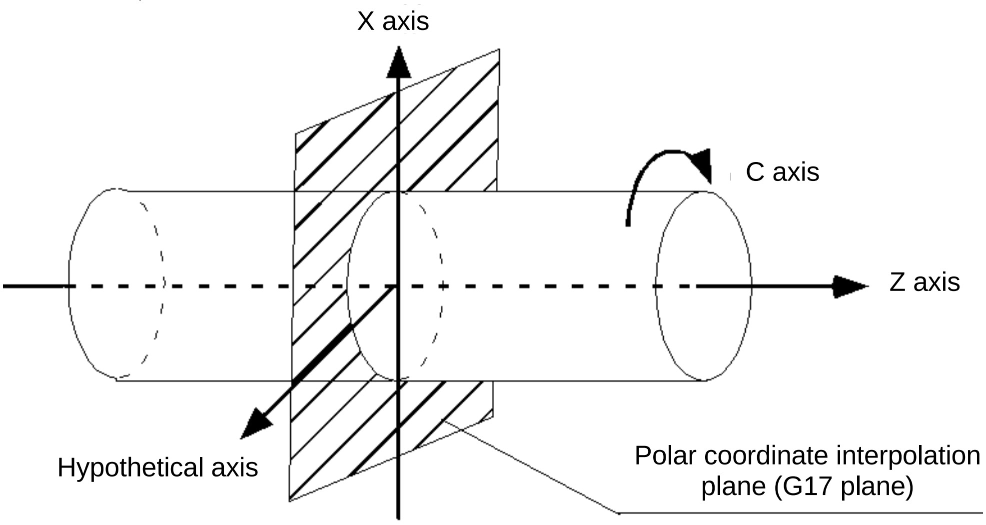


Fig. 1 Diagram explaining polar coordinate interpolation axes

(1) Polar coordinate interpolation mode
(G12.1)
The polar coordinate interpolation mode is established by designating the G12.1 command.
The plane (hereafter referred to as the "polar coordinate interpolation plane"), for which the linear axis will be first orthogonal axis and the hypothetical axis at right angles to the linear axis will be the second axis, is selected.
Polar coordinate interpolation is performed on this plane.

(a) Linear interpolation and circular interpolation can be designated in the polar coordinate interpolation mode.
(b) Either absolute values or incremental values can be designated.
(c) Tool radius compensation can be applied to the program commands. Polar coordinate interpolation is performed for the path after tool radius compensation.
(d) For the feed rate, designate a tangential rate on the polar coordinate interpolation plane (orthogonal coordinate system) using the F command.
The F rate is in either mm/min or inch/mm units.

(2) Polar coordinate interpolation cancel mode
(G13.1)
The polar coordinate interpolation cancel mode is established by designating the G13.1 command.

4. Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

	E60	E68
M system	1000	1000
L system	1000	1000

The rapid traverse rate can be set independently for each axis.
G00, G27, G28, G29, G30 and G60 are the effective commands for the rapid traverse rate.
Override can be applied to the rapid traverse rate using the external signal supplied.

- Rapid Traverse Rate setting range

(1) For E60

Least input increment	**B**
Metric input	1 to 1000000 (mm/min, °/min)
Inch input	1 to 39370 (inch/min)

Least input increment B: 0.001 mm (0.0001 inch)

(2) For E68

Least input increment	**B**	**C**
Metric input	1 to 1000000 (mm/min, °/min)	1 to 100000 (mm/min, °/min)
Inch input	1 to 39370 (inch/min)	1 to 3937 (inch/min)

Least input increment B: 0.001 mm (0.0001 inch)
Least input increment C: 0.0001 mm (0.00001 inch)

4.1.2 Cutting Feed Rate (m/min)

	E60	E68
M system	1000	1000
L system	1000	1000

This function specifies the feed rate of the cutting commands, and a feed amount per spindle rotation or feed amount per minute is commanded.
Once commanded, it is stored in the memory as a modal value. The feed rate modal value is cleared to zero only when the power is turned ON.
The maximum cutting feed rate is clamped by the cutting feed rate clamp parameter (whose setting range is the same as that for the cutting feed rate).

- Cutting Feed Rate setting range

(1) For E60

Least input increment	**B**
Metric input	1 to 1000000 (mm/min, °/min)
Inch input	1 to 39370 (inch/min)

Least input increment B: 0.001 mm (0.0001 inch)

(2) For E68

Least input increment	**B**	**C**
Metric input	1 to 1000000 (mm/min, °/min)	1 to 100000 (mm/min, °/min)
Inch input	1 to 39370 (inch/min)	1 to 3937 (inch/min)

Least input increment B: 0.001 mm (0.0001 inch)
Least input increment C: 0.0001 mm (0.00001 inch)

- Effective G-code commands for the cutting feed
  G01, G02, G03, G2.1 G3.1 G33, etc. As to others, refer to the interpolation specifications.

4.1.3 Manual Feed Rate (m/min)

	E60	E68
M system	1000	1000
L system	1000	1000

The manual feed rates are designated as the feed rate in the jog mode or incremental feed mode for manual operation and the feed rate during dry run ON for automatic operation. The manual feed rates are set with external signals.
The manual feed rate signals from the PLC includes two methods, the code method and numerical value method.
Which method to be applied is determined with a signal common to the entire system.
The signals used by these methods are common to all axes.

- Setting range under the code method
  Metric input 0.00 to 14000.00 mm/min (31 steps)
  Inch input 0.000 to 551.000 inch/min (31 steps)

- Setting range under the numerical value method
  Metric input 0 to 1000000.00 mm/min in 0.01 mm/min increments
  Inch input 0 to 39370 inch/min in 0.001 inch/min increments

  Multiplication factor PCF1 and PCF2 are available with the numerical value setting method.

4.2 Feed Rate Input Methods

4.2.1 Feed per Minute

	E60	E68
M system	○	○
L system	○	○

[M system]

By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feed rate per minute (mm/min, inch/min).

(1) For E60

Metric input (mm)

Least input increment		(B) 0.001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.01 to 1000000.000

Inch input (inch)

Least input increment		(B) 0.0001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.001 to 100000.0000

(2) For E68

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.01 to 1000000.000	0.001 to 100000.000

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.001 to 100000.0000	0.001 to 10000.0000

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min). To assign commands under 1 mm/min (or 1 inch/min), ensure that commands are assigned with a decimal point.
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter (I_Sync) to OFF.

[L system]

By issuing the G94 command, the commands from that block are issued directly by the numerical value following F as the feed rate per minute (mm/min, inch/min).

(1) For E60

Metric input (mm)

Least input increment		(B) 0.001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.001 to 1000000.000

Inch input (inch)

Least input increment		(B) 0.0001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.0001 to 39370.0787

(2) For E68

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/min)	**without decimal point** **with decimal point**	F1 = 1 mm/min F1. = 1 mm/min	F1 = 1 mm/min F1. = 1 mm/min
Command range (mm/min)		0.001 to 1000000.000	0.0001 to 100000.0000

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/min)	**without decimal point** **with decimal point**	F1 = 1 inch/min F1. = 1 inch/min	F1 = 1 inch/min F1. = 1 inch/min
Command range (inch/min)		0.0001 to 39370.0787	0.00001 to 3937.00787

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min). To assign commands under 1 mm/min (or 1 inch/min), ensure that commands are assigned with a decimal point.
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter (I_Sync) to OFF.

4.2.2 Feed per Revolution

	E60	E68
M system	○	○
L system	○	○

By issuing the G95 command, the commands from that block are issued directly by the numerical value following F as the feed rate per spindle revolution (mm/revolution or inch/revolution).
The least command increment and command range of the feed rate designation F are as follows.

[M system]

(1) For E60

Metric input (mm)

Least input increment		(B) 0.001 mm
F command increment (mm/rev)	**without decimal point** **with decimal point**	F1 = 0.01 F1. = 1
Command range (mm/rev)		0.001 to 999.999

Inch input (inch)

Least input increment		(B) 0.0001 inch
F command increment (inch/rev)	**without decimal point** **with decimal point**	F1 = 0.001 F1. = 1
Command range (inch/rev)		0.0001 to 999.9999

(2) For E68

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/rev)	**without decimal point** **with decimal point**	F1 = 0.01 F1. = 1	F1 = 0.01 F1. = 1
Command range (mm/rev)		0.001 to 999.999	0.0001 to 99.9999

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/rev)	**without decimal point** **with decimal point**	F1 = 0.001 F1. = 1	F1 = 0.001 F1. = 1
Command range (inch/rev)		0.0001 to 999.9999	0.00001 to 99.99999

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min).
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter (I_Sync) to OFF.

[L system]

(1) For E60

Metric input (mm)

Least input increment		(B) 0.001 mm
F command increment (mm/rev)	**without decimal point with decimal point**	F1 = 0.0001 F1. = 1
Command range (mm/rev)		0.0001 to 999.999

Inch input (inch)

Least input increment		(B) 0.0001 inch
F command increment (inch/rev)	**without decimal point with decimal point**	F1 = 0.000001 F1. = 1
Command range (inch/rev)		0.000001 to 99.999999

(2) For E68

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/rev)	**without decimal point with decimal point**	F1 = 0.0001 F1. = 1	F1 = 0.0001 F1. = 1
Command range (mm/rev)		0.0001 to 999.9999	0.00001 to 99.99999

Inch input (inch)

Least input increment		(B) 0.0001 inch	(C) 0.00001 inch
F command increment (inch/rev)	**without decimal point with decimal point**	F1 = 0.000001 F1. = 1	F1 = 0.000001 F1. = 1
Command range (inch/rev)		0.000001 to 99.999999	0.0000001 to 9.9999999

- When commands without a decimal point have been assigned, it is not possible to assign commands under 1 mm/min (or 1 inch/min).
- The initial status after power-ON can be set to asynchronous feed (per-minute-feed) by setting the "Initial synchronous feed" parameter (I_Sync) to OFF.

4.2.4 F1-Digit Feed

	E60	E68
M system	○	○
L system	○	○

When the "F1digt" parameter is ON, the feed rate registered by parameter in advance can be assigned by designating a single digit following address F.
There are six F codes: F0 and F1 to F5. The rapid traverse rate is established when F0 is designated which is the same as the G00 command. When one of the codes F1 to F5 is designated, the cutting feed rate set to support the code serves as the valid rate command. When a command higher than F5 is designated, it serves as a regular F command's direct command.
When an F 1-digit feed command has been designated, the "In F 1-digit" external output signal is output.

4.3 Override

4.3.1 Rapid Traverse Override

	E60	E68
M system	○	○
L system	○	○

(1) Type 1 (code method)

Four levels of override (1%, 25%, 50% and 100%) can be applied to manual or automatic rapid traverse using the external input signal supplied.

(2) Type 2 (value setting method)

Override can be applied in 1% steps from 0% to 100% to manual or automatic rapid traverse using the external input signal supplied.

(Note 1) Type 1 and type 2 can be selected by PLC processing.

(Note 2) A PLC must be built into the unit for type 2.

4.3.2 Cutting Feed Override

	E60	E68
M system	○	○
L system	○	○

(1) Type 1 (code method)

Override can be applied in 10% steps from 0% to 300% to the feed rate command designated in the machining program using the external input signal supplied.
Code method commands are assigned as combinations of Y bit signals from the PLC.

(2) Type 2 (value setting method)

Override can be applied in 1% steps from 0% to 327% to the feed rate command designated in the machining program using the external input signal supplied.

(Note 1) A PLC must be built into the unit for type 2.

4.3.3 2nd Cutting Feed Override

	E60	E68
M system	○	○
L system	○	○

Override can be further applied in 0.01% steps from 0% to 327.67% as a second stage override to the feed rate after the cutting feed override has been applied.

(Note 1) A PLC must be built into the unit for this function.

4.3.4 Override Cancel

	E60	E68
M system	○	○
L system	○	○

By turning on the override cancel external signal, the override is automatically set to 100% for the cutting feed during an automatic operation mode (tape, memory and MDI).

(Note 1) The override cancel signal is not valid for manual operation.

(Note 2) When the cutting feed override or second cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.

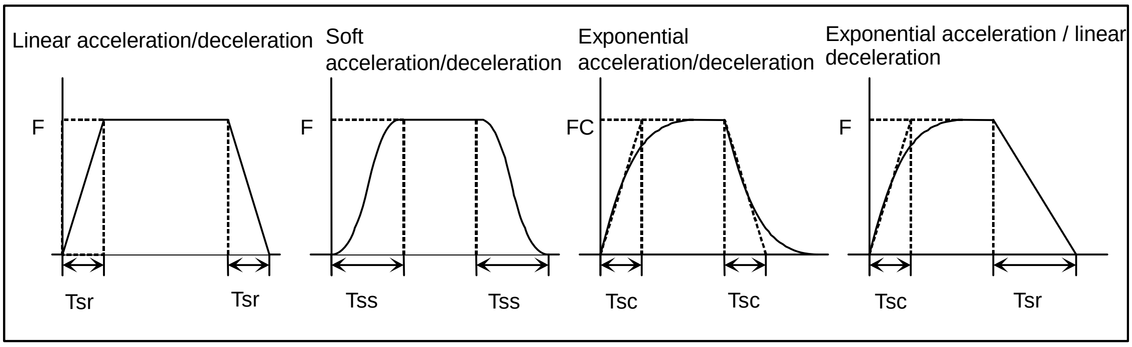
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration / Deceleration

4.4.1 Automatic Acceleration / Deceleration after Interpolation

	E60	E68
M system	○	○
L system	○	○

Acceleration/deceleration is applied to all commands automatically. The acceleration/deceleration patterns are linear acceleration/deceleration, soft acceleration/deceleration, exponent function acceleration/deceleration, exponent function acceleration/linear deceleration and any of which can be selected by using a parameter.
For rapid traverse feed or manual feed, acceleration/deceleration is always made for each block, and the time constant can be set for each axis separately.



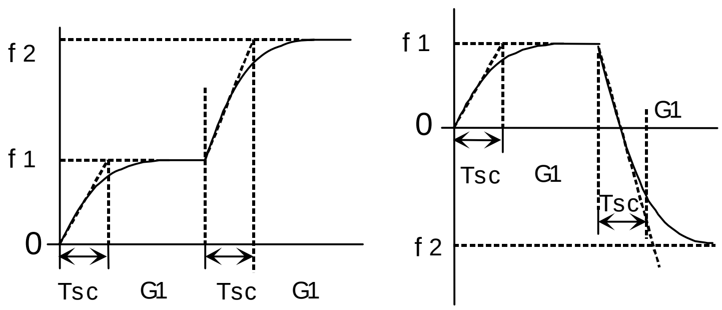
(Note 1) The rapid traverse feed acceleration/deceleration patterns are also effective for the following:

G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG, incremental feed, return to reference position.

(Note 2) Acceleration/deceleration in handle feed mode is usually performed according to the acceleration/deceleration pattern for cutting feed. However, a parameter can be specified to select a pattern with no acceleration/deceleration (step).

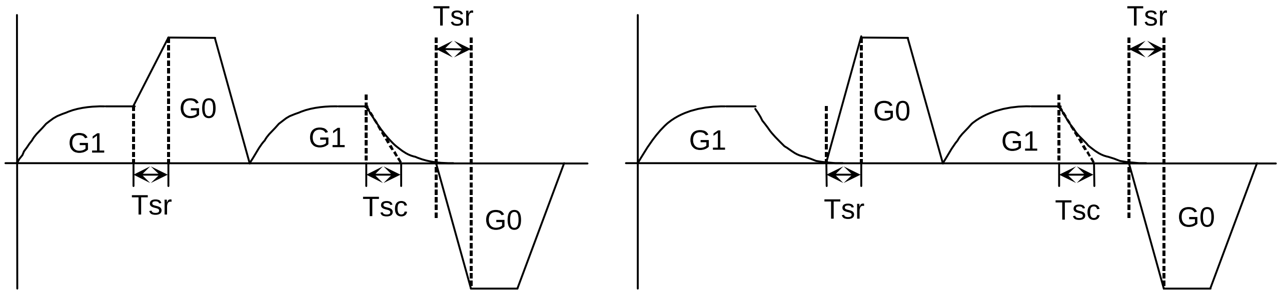
Acceleration / Deceleration during Continuing Blocks

(1) Continuous G1 blocks



The tool does not decelerate between blocks.

(2) Continuous G1-G0 blocks



If the G0 command direction is the same as that for G1, whether G1 is to be decelerated is selected using a parameter.
If no deceleration is set, superposition is performed even when G0 is in the constant inclination acceleration/deceleration state.
If the G0 command direction is the opposite of that for G1, G0 will be executed after G1 has decelerated.
(In the case of two or more simultaneous axes, G0 will also be executed after G1 has decelerated when the G0 command direction is the opposite of that for G1 for even one axis.)

4.4.2 Rapid Traverse Constant Inclination Acceleration / Deceleration

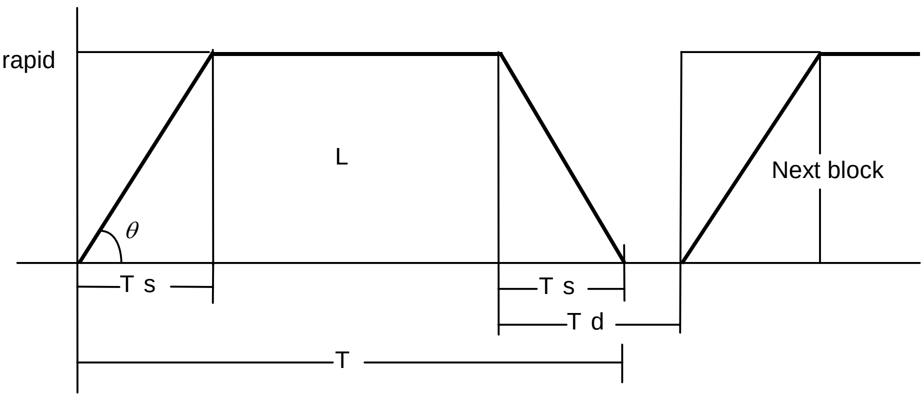
	E60	E68
M system	○	○
L system	○	○

This function performs acceleration and deceleration at a constant inclination during linear acceleration/deceleration in the rapid traverse mode. Compared to the method of acceleration/deceleration after interpolation, the constant inclination acceleration/deceleration method makes for improved cycle time.

Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration and linear deceleration.

The acceleration/deceleration patterns in the case where rapid traverse constant inclination acceleration/deceleration are performed are as follows.

(1) When the interpolation distance is longer than the acceleration and deceleration distance



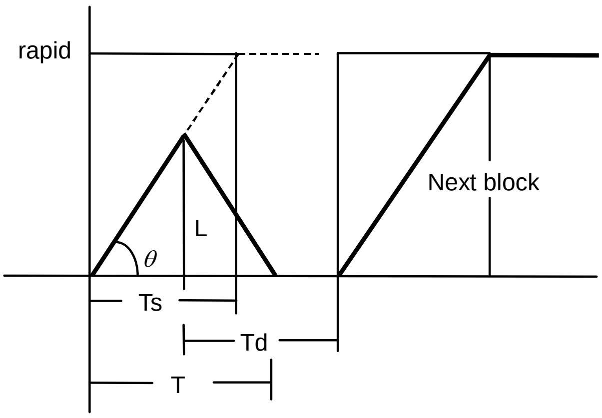
rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

$$T = \frac{L}{rapid} + Ts$$

$$Td = Ts + (0\sim1.7\ \text{ms})$$

$$\theta = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

(2) When the interpolation distance is shorter than the acceleration and deceleration distance



rapid: Rapid traverse rate
Ts: Acceleration/deceleration time constant
Td: Command deceleration check time
θ: Acceleration/deceleration inclination
T: Interpolation time
L: Interpolation distance

$$T = 2\times\sqrt{Ts \times L / rapid}$$

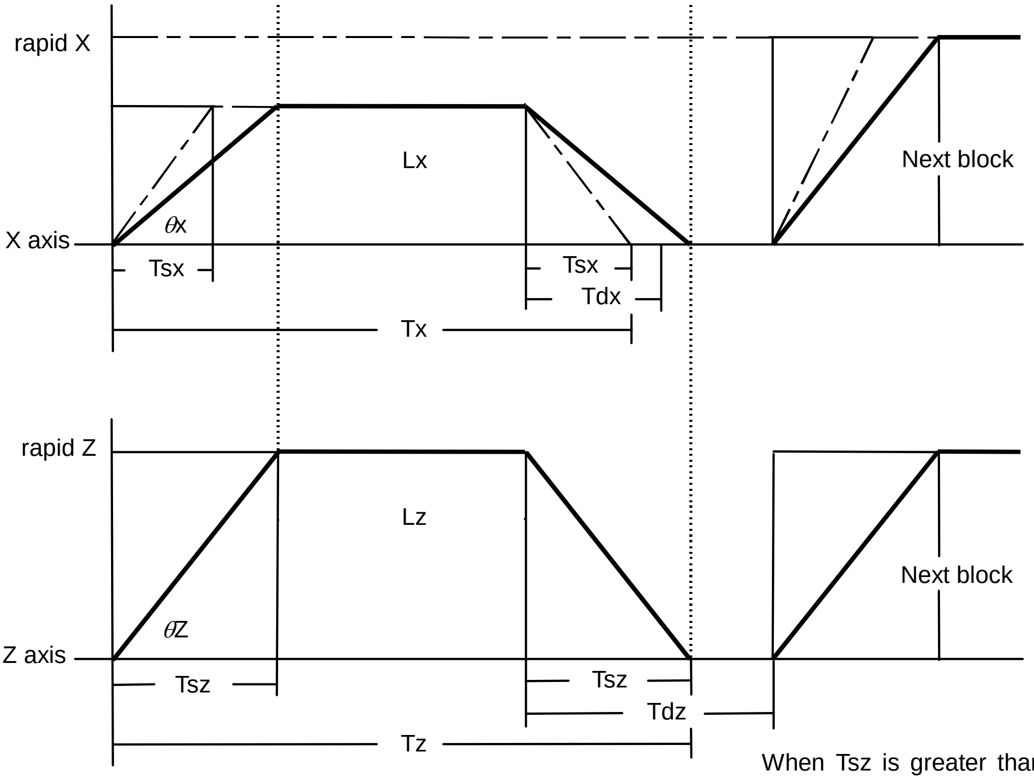
$$Td = \frac{T}{2} + (0 \sim 1.7\ \text{ms})$$

$$\theta = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

The time required to perform a command deceleration check during rapid traverse constant inclination acceleration/deceleration is the longest value among the rapid traverse deceleration check times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration/deceleration time constant, and the interpolation distance, respectively.

(3) 2-axis simultaneous interpolation (When linear interpolation is used, Tsx < Tsz, and Lx ≠ Lz)

When 2-axis simultaneous interpolation (linear interpolations) is performed during rapid traverse constant inclination acceleration and deceleration, the acceleration (deceleration) time is the longest value of the acceleration (deceleration) times determined for each axis by the rapid traverse rate of commands executed simultaneously, the rapid traverse acceleration and deceleration time constant, and the interpolation distance, respectively. Consequently, linear interpolation is performed even when the axes have different acceleration and deceleration time constants.



When Tsz is greater than Tsx, Tdz is also greater than Tdx, and Td = Tdz in this block.

The program format of G0 (rapid traverse command) when rapid traverse constant inclination acceleration/deceleration are executed is the same as when this function is invalid (time constant acceleration/deceleration).
This function is valid only for G0 (rapid traverse).

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

	E60	E68
M system	○	○
L system	○	○

(1) Lead designation

The thread cutting with designated lead are performed based on the synchronization signals from the spindle encoder.

G33 Zz1 Qq1 Ff1/Ee1 ;

G33 : Thread command
Zz1 : Thread length
Qq1 : Shift angle ("q1" is the shift angle at thread cutting start 0 to 360°)
Ff1 : Thread lead
Ee1 : Thread lead (precise lead threads)

The tables below indicate the thread lead ranges.

[M system]

(a) For E60

Metric command			**Inch command**		
Command increment (mm)	**F (mm/rev)**	**E (mm/rev)**	**Command increment (inch)**	**F (inch/rev)**	**E (inch/rev)**
0.001	0.001 to 999.999	0.00001 to 999.99999	0.0001	0.0001 to 39.3700	0.000001 to 39.370078

(b) For E68

Metric command			**Inch command**		
Command increment (mm)	**F (mm/rev)**	**E (mm/rev)**	**Command increment (inch)**	**F (inch/rev)**	**E (inch/rev)**
0.001	0.001 to 999.999	0.00001 to 999.99999	0.0001	0.0001 to 39.3700	0.000001 to 39.370078
0.0001	0.0001 to 99.9999	0.000001 to 99.999999	0.00001	0.00001 to 3.93700	0.000001 to 3.937007

[L system]

(a) For E60

Metric command			Inch command		
Command increment (mm)	**F (mm/rev)**	**E (mm/rev)**	**Command increment (inch)**	**F (inch/rev)**	**E (inch/rev)**
0.001	0.0001 to 999.9999	0.00001 to 999.99999	0.0001	0.00001 to 99.999999	0.000010 to 9.9999999

(b) For E68

Metric command			Inch command		
Command increment (mm)	**F (mm/rev)**	**E (mm/rev)**	**Command increment (inch)**	**F (inch/rev)**	**E (inch/rev)**
0.001	0.0001 to 999.9999	0.00001 to 999.99999	0.0001	0.00001 to 99.999999	0.000010 to 9.9999999
0.0001	0.00001 to 99.99999	0.000001 to 99.999999	0.00001	0.000001 to 9.9999999	0.0000010 to 0.99999999

The axis direction characterized by a large movement serves as the reference for the lead.

(2) Thread number designation

Inch threads are cut by designating the number of threads per inch with the E address.
Whether the E command is a thread number designation or lead designation is selected with the parameters.

G33 Zz1 Qq1 Ee1 ;	
G33	: Thread command
Zz1	: Thread length
Qq1	: Shift angle ("q1" is the shift angle at thread cutting start 0 to 360°)
Ee1	: Thread number per inch

The tables below indicate the thread leads.

[M system]

(a) For E60

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03 to 999.99	0.0001	0.0255 to 9999.9999

(b) For E68

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03 to 999.99	0.0001	0.0255 to 9999.9999
0.0001	255 to 9999.999	0.00001	0.25401 to 999.9999

[L system]

(a) For E60

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03 to 999.99	0.0001	0.0101 to 9999.9999

(b) For E68

Metric command		Inch command	
Command increment (mm)	Thread number command range (thread/inch)	Command increment (inch)	Thread number command range (thread/inch)
0.001	0.03 to 999.99	0.0001	0.0101 to 9999.9999
0.0001	0.255 to 9999.999	0.00001	0.1001 to 999.9999

The number of thread per inch is commanded for both metric and inch systems, and the direction of the axis with a high movement serves as the reference.

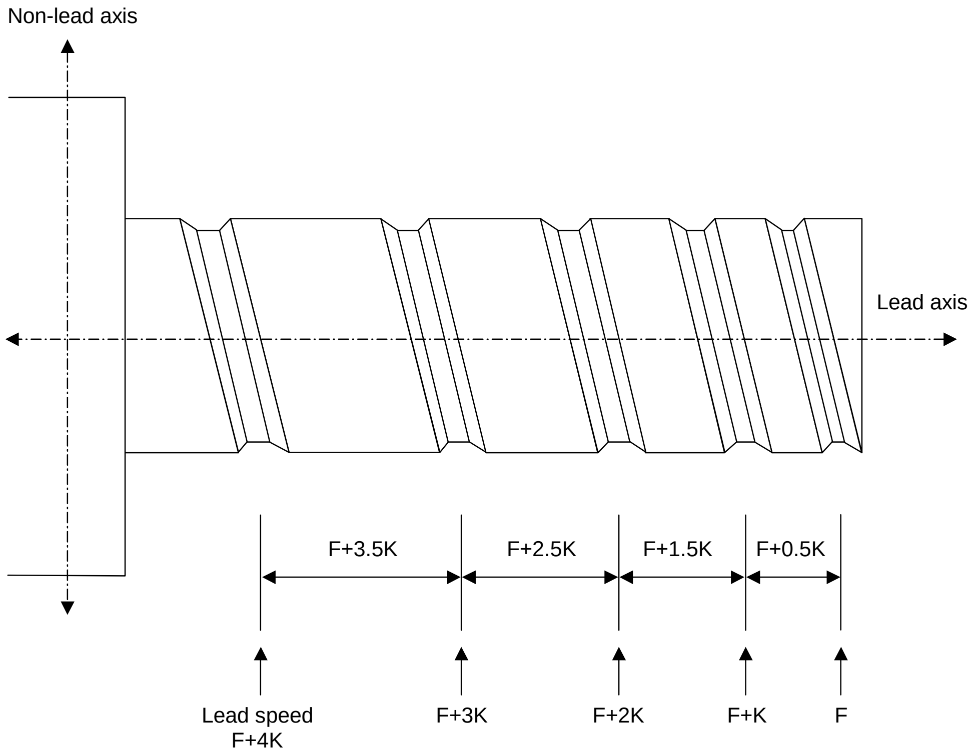
4.5.2 Variable Lead Thread Cutting

	E60	E68
M system	–	–
L system	○	○

By commanding the lead increment/decrement amount per thread rotation, variable lead thread cutting can be done.
The machining program is commanded in the following manner.

G34 X/U__Z/W__F/E__K__;

X/U : Thread end point X coordinate
Z/W : Thread end point Z coordinate
F/E : Thread's basic lead
K : Lead increment/decrement amount per thread rotation



4.5.3 Synchronous Tapping

4.5.3.1 Synchronous Tapping Cycle

	E60	E68
M system	Δ	○
L system	Δ	○

This function performs tapping through the synchronized control of the spindle and servo. This eliminates the need for floating taps and enables tapping to be conducted at a highly precise tap depth.

(1) Tapping pitch assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Pp1 Ff1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,R2 ;

X, Y	: Hole position data, hole drilling coordinate position
Z	: Hole machining data, hole bottom position
R	: Hole machining data, hole R position
P	: Hole machining data, dwell time at hole bottom
F	: Z-axis feed amount (tapping pitch) per spindle rotation
S	: Spindle speed
,S	: Rotation speed of spindle during retract
,I	: In-position width of positioning axis
,J	: In-position width of hole drilling axis
,R	: Synchronization method selection (r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode)

(2) Tapping thread number assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Pp1 Ee1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,R2 ;

X, Y	: Hole position data, hole drilling coordinate position
Z	: Hole machining data, hole bottom position
R	: Hole machining data, hole R position
P	: Hole machining data, dwell time at hole bottom
E	: Tap thread number per 1-inch feed of Z axis
S	: Spindle speed
,S	: Rotation speed of spindle during retract
,I	: In-position width of positioning axis
,J	: In-position width of hole drilling axis
,R	: Synchronization method selection (r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode)

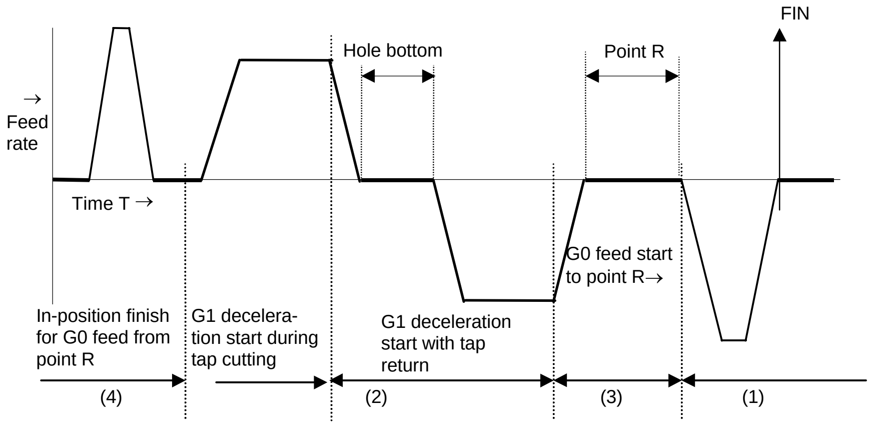
The control state will be as described below when a tapping mode command (G74, G84) is commanded.

1. Cutting override	Fixed to 100%
2. Feed hold invalid	
3. "In tapping mode" signal is output	
4. Deceleration command between blocks invalid	
5. Single block invalid	

The tapping mode will be canceled with the following G commands.

G61 .... Exact stop check mode
G61.1 .... High-accuracy control mode (E68)
G62 .... Automatic corner override
G64 .... Cutting mode

(Note) The synchronous tapping cycle can be used for axes other than the Z axis with the plane selection.
Furthermore, in-position checks can be performed at the hole bottom or point R, etc. using the parameters. The figure below shows the correlation between the in-position width and the movement of the tapping axis of the synchronous tapping in-position check.



(1) Section where in-position check is performed using servo in-position width
(2) Section where in-position check is performed using in-position width for tapping
(3) Section where in-position check is performed using in-position width for cutting feed (G1, G2, G3)
(4) Section where in-position check is performed using in-position width for rapid traverse (G0)

4.5.3.2 Pecking Tapping Cycle

	E60	E68
M system	–	O
L system	–	–

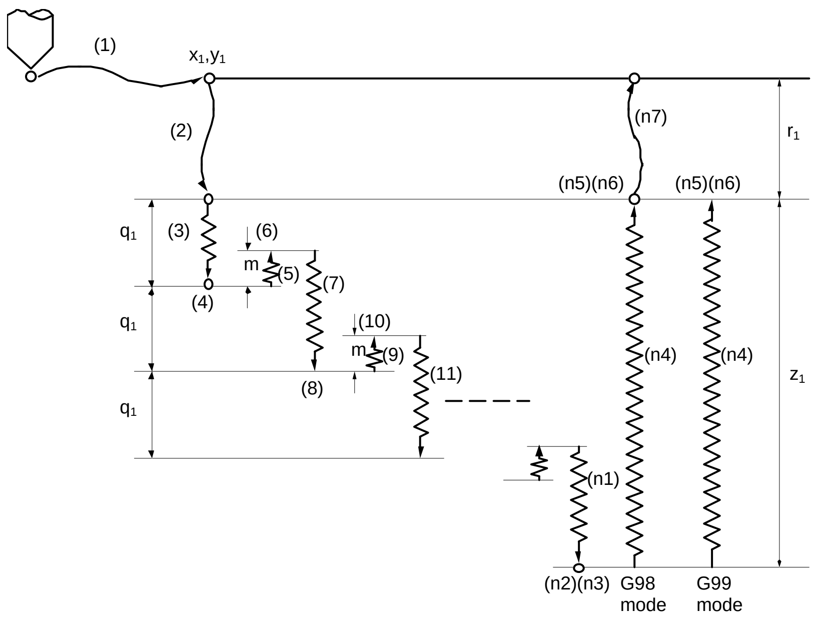
The load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.
The amount retracted from the hole bottom is set to the parameters.
Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter.
When "depth of cut per pass Q" is designated in the block containing the G84 or G74 command in the state where the pecking tapping cycle is selected by parameter, the pecking tapping cycle is executed.
In the following cases, the normal tapping cycle is established.

- When Q is not designated
- When the command value of Q is "0"

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 ;

X,Y	: Hole drilling point position
Z	: Hole bottom position
R	: Point R position
Q	: Depth of cut per pass (designated as an incremental value)
F	: Z-axis feed amount (tapping pitch) per spindle rotation
P	: Dwell time at hole bottom position
S	: Rotation speed of spindle
,S	: Rotation speed of spindle during retract
,I	: In-position width of positioning axis
,J	: In-position width of hole drilling axis
,R	: Synchronization method selection
	(r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode)

(Note) When ",R0" is commanded, F address is regarded as cutting feed rate.



(1)	G0 Xx1 Yy1 ,Ii1
(2)	G0 Zr1
(3)	G1 Zq1 Ff1
(4)	M4 (Spindle reverse rotation)
(5)	G1 Z-m Ff1
(6)	M3 (Spindle forward rotation)
(7)	G1 Z(q1+m) Ff1
(8)	M4 (Spindle reverse rotation)
(9)	G1 Z-m Ff1
(10)	M3 (Spindle forward rotation)
(11)	G1 Z(q1+m) Ff1
:	:
(n1)	G1 Z(z1-q1*n) Ff1
(n2)	G4 Pp1
(n3)	M4 (Spindle reverse rotation)
(n4)	G1 Z-z1 Ff1 Ss2
(n5)	G4 Pp1
(n6)	M3 (Spindle forward rotation)
(n7)	G98 mode G0 Z-r1 ,Ij1 G99 mode No movement

*1. m : Parameter
2. This program is for the G84 command. The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

4.5.3.3 Deep-hole Tapping Cycle

	E60	E68
M system	–	○
L system	–	–

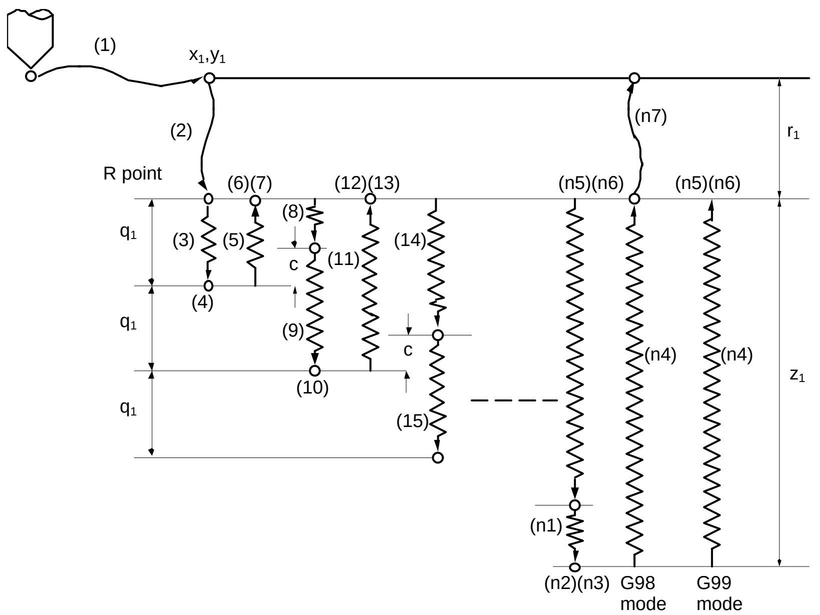
In the deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes. Under the deep-hole tapping cycle, the tool is retracted to the R-point every time.
Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter. When "depth of cut per pass Q" is designated in the block containing the G84 or G74 tapping cycle command in the state where the deep-hole tapping cycle is selected, the deep-hole tapping cycle is executed.
In the following cases, the normal tapping cycle is established.

- When Q is not designated
- When the command value of Q is "0"

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 ;

X,Y	: Hole drilling point position
Z	: Hole bottom position
R	: Point R position
Q	: Depth of cut per pass (designated as an incremental value)
F	: Z-axis feed amount (tapping pitch) per spindle rotation
P	: Dwell time at hole bottom and point R return
S	: Rotation speed of spindle
,S	: Rotation speed of spindle during retract
,I	: In-position width of positioning axis
,J	: In-position width of hole drilling axis
,R	: Synchronization method selection (r2=1 synchronous tapping mode, r2=0 asynchronous tapping mode)

(Note) When ",R0" is commanded, F address is regarded as cutting feed rate.



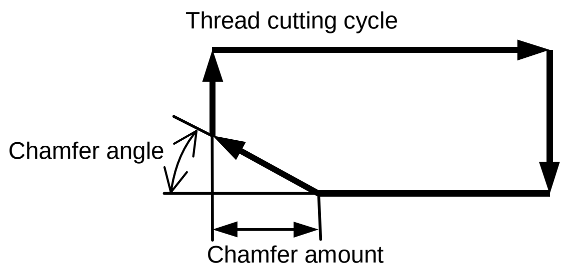
(1)	G0 Xx1 Yy1
(2)	G0 Zr1
(3)	G9 G1 Zq1 Ff1
(4)	M4 (Spindle reverse rotation)
(5)	G9 G1 Z-q1 Ff1
(6)	G4 Pp1
(7)	M3(Spindle forward rotation)
(8)	G1 Z(q1-c) Ff1
(9)	G9 G1 Z(q1+c) Ff1
(10)	M4 (Spindle reverse rotation)
(11)	G9 G1 Z-(2･q1) Ff1
(12)	G4 Pp1
(13)	M3 (Spindle forward rotation)
(14)	G1 Z(2･q1-c) Ff1
(15)	G9 G1 Z(q1+c) Ff1
:	:
(n1)	G9 G1 Z(z1-q1*n+c) Ff1
(n2)	G4 Pp1
(n3)	M4 (Spindle reverse rotation)
(n4)	G9 G1 Z-z1 Ff1
(n5)	G4 Pp1
(n6)	M3 (Spindle forward rotation)
(n7)	G98 mode G0 Z-r1 G99 mode No movement

*1. Clearance amount c : Parameter
2. This program is for the G84 command. The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

4.5.4 Chamfering

	E60	E68
M system	–	–
L system	○	○

Chamfering can be validated during the thread cutting cycle by using external signals.
The chamfer amount and angle are designated with parameters.

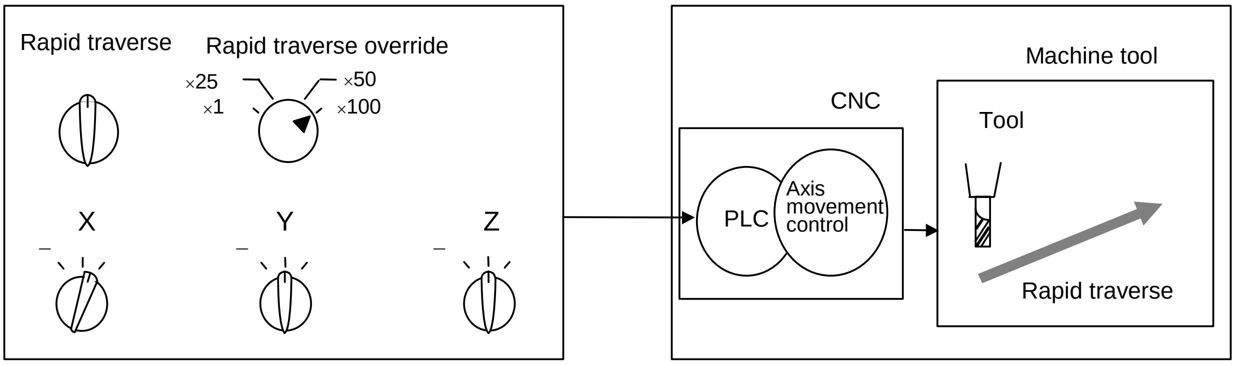


4.6 Manual Feed

4.6.1 Manual Rapid Traverse

	E60	E68
M system	○	○
L system	○	○

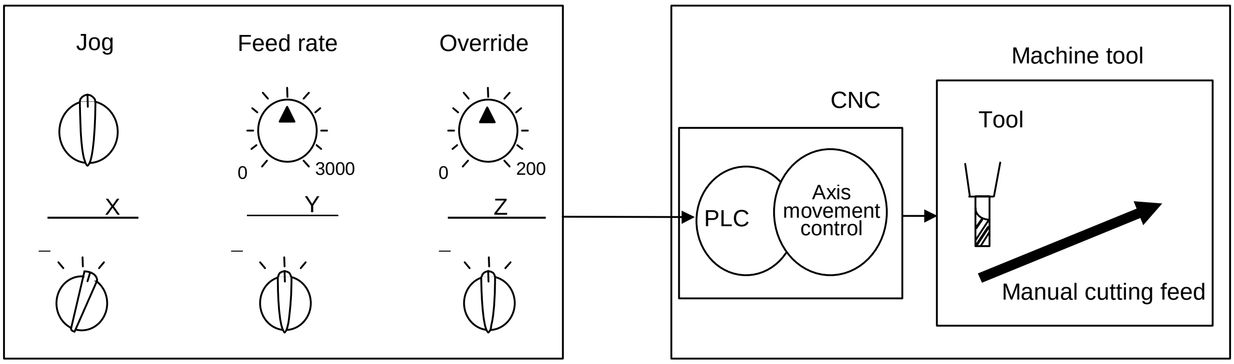
When the manual rapid traverse mode is selected, the tool can be moved at the rapid traverse rate for each axis separately. Override can also be applied to the rapid traverse rate by means of the rapid traverse override function.



4.6.2 Jog Feed

	E60	E68
M system	○	○
L system	○	○

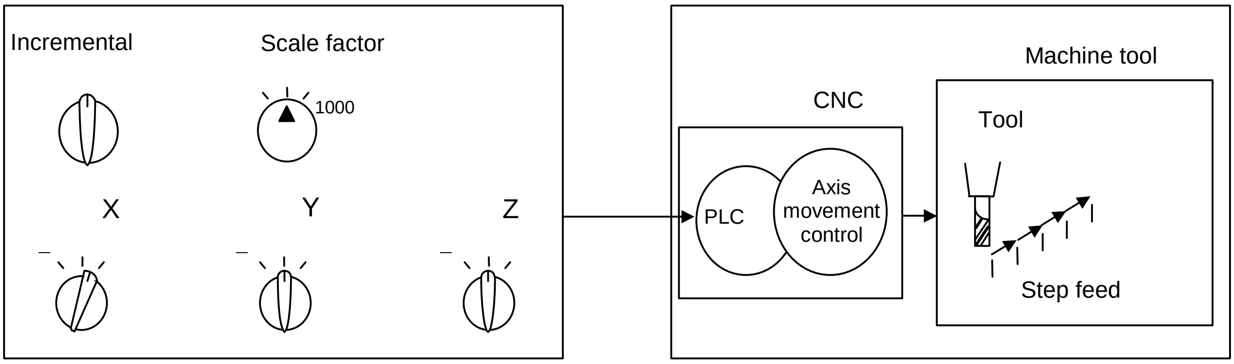
When the jog feed mode is selected, the tool can be moved in the axis direction (+ or –) in which the machine is to be moved at the per-minute feed.



4.6.3 Incremental Feed

	E60	E68
M system	○	○
L system	○	○

When the incremental feed mode is selected, the tool can be operated by an amount equivalent to the designated amount (incremental value) in the axis direction each time the jog switch is pressed. The incremental feed amount is the amount obtained by multiplying the least input increment that was set with the parameter by the incremental feed magnification rate.



4.6.4 Handle Feed

	E60	E68
M system	○(2)	○(2)
L system	○(2)	○(2)

(1-axis)
In the handle feed mode, the machine can be moved in very small amounts by rotating the manual pulse generator. The scale can be selected from X1, X10, X100, X1000 or random.

(Note 1) The actual movement amount and scale may not match if the manual pulse generator is rotated quickly.

(2 axes)
In the handle feed mode, individual axes can be moved in very small amounts either separately or simultaneously by rotating the manual pulse generators installed on each of the axes.

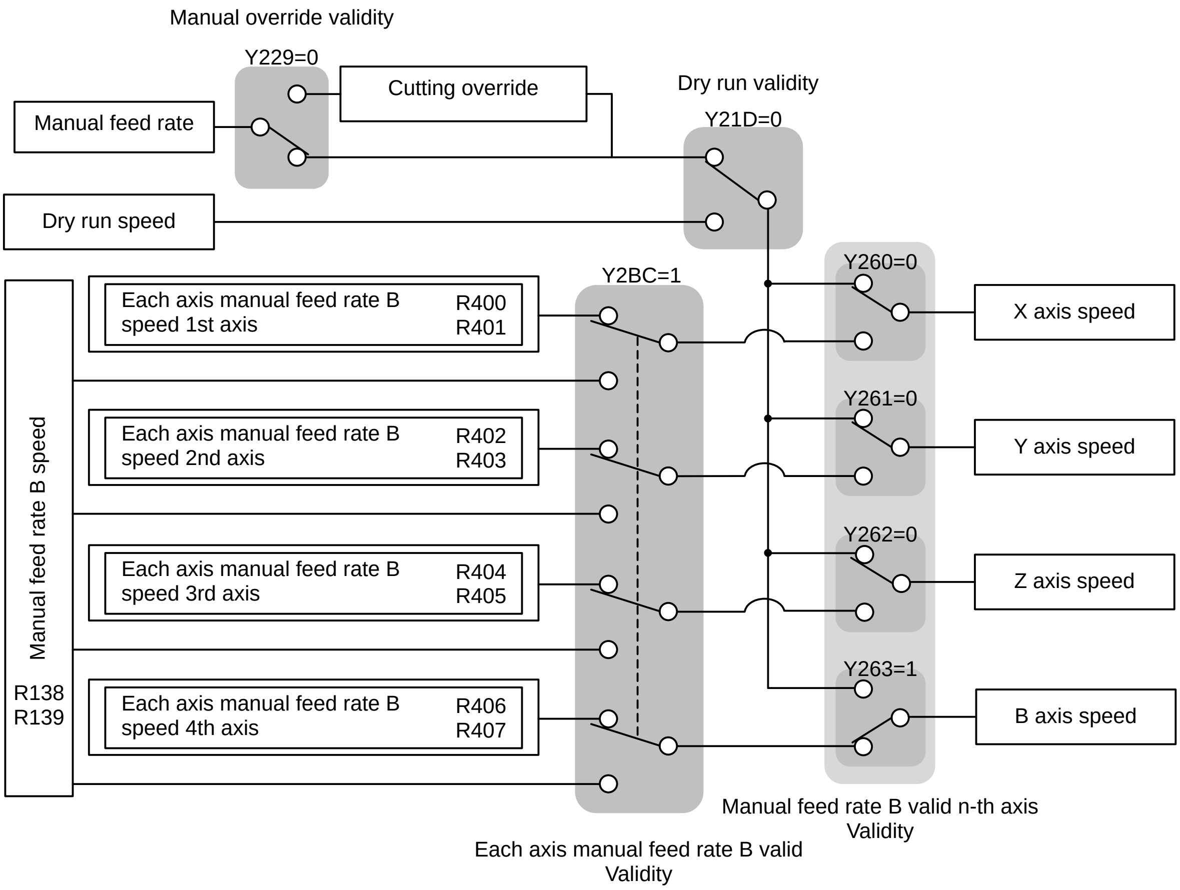
(Note 1) The actual movement amount and scale may not match if the manual pulse generator is rotated quickly.

4.6.5 Manual Feed Rate B

	E60	E68
M system	○	○
L system	○	○

"Manual feed rate B" is a function that sets a random axis feed rate from the user PLC separately from the "manual feed rate". The "manual feed rate B" feed rate setting can be selected from the feed rate common for all axes and the feed rate independent of reach axis. By combining the "manual feed rate B" function with the manual/automatic simultaneous function, a random axis can be moved at the "manual feed rate B" independently of the machining program operation even during automatic operation. Similarly, if the jog mode and other manual operation mode are set simultaneously, a random axis can be moved at a speed independent from the "manual feed rate" even during the manual operation mode.
The "manual feed rate B" function can move an axis at a speed different from the "manual feed rate". This is not affected by dry run, or by manual or cutting override, so a random axis can be moved independently even in operations during automatic operation or override during manual axis movement.

The relation of the "manual feed rate B" and "manual feed rate" is shown below.



4.7 Dwell

4.7.1 Dwell (Time-based Designation)

	E60	E68
M system	○	○
L system	○	○

The G04 command temporarily stops the machine movement and sets the machine standby status for the time designated in the program.

(G94) G04 Xx1/Uu1 ; or (G94) G04 Pp1 ;

G94	: Asynchronous
G04	: Dwell
Xx1, Uu1, Pp1	: Time (For U, E68 L system only.)

(1) When designating the dwell time with X or U, the decimal point command is valid.
(2) When designating the dwell time with P, the availability of the decimal point command can be selected with the parameter. When the decimal point command is invalid in the parameter setting, the command below the decimal point issued with P is ignored.
(3) When the decimal point command is valid or invalid, the dwell time command range is as follows.

Command range when the decimal point command is valid	Command range when the decimal point command is invalid
0.001 to 99999.999 (s)	1 to 99999999 (ms)

5. Program Memory / Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory.

5.1.1 Memory Capacity (Number of Programs Stored)

600 m (400 programs)

	E60	E68
M system	○	○
L system	○	○

5.2 Editing

5.2.1 Program Editing

	E60	E68
M system	○	○
L system	○	○

The following editing functions are possible.

(1) Program erasing

(a) Machining programs can be erased individually or totally.
(b) When all machining programs are to be erased, the programs are classified with their No. into B: 8000 to 8999, C: 9000 to 9999, and A: all others.

(2) Program filing

(a) This function displays a list of the machining programs stored (registered) in the controller memory.
(b) The programs are displayed in ascending order.
(c) Comments can be added to corresponding program numbers.

(3) Program copying

(a) Machining programs stored in the controller memory can be copied, condensed or merged.
(b) The program No. of the machining programs in the memory can be changed.

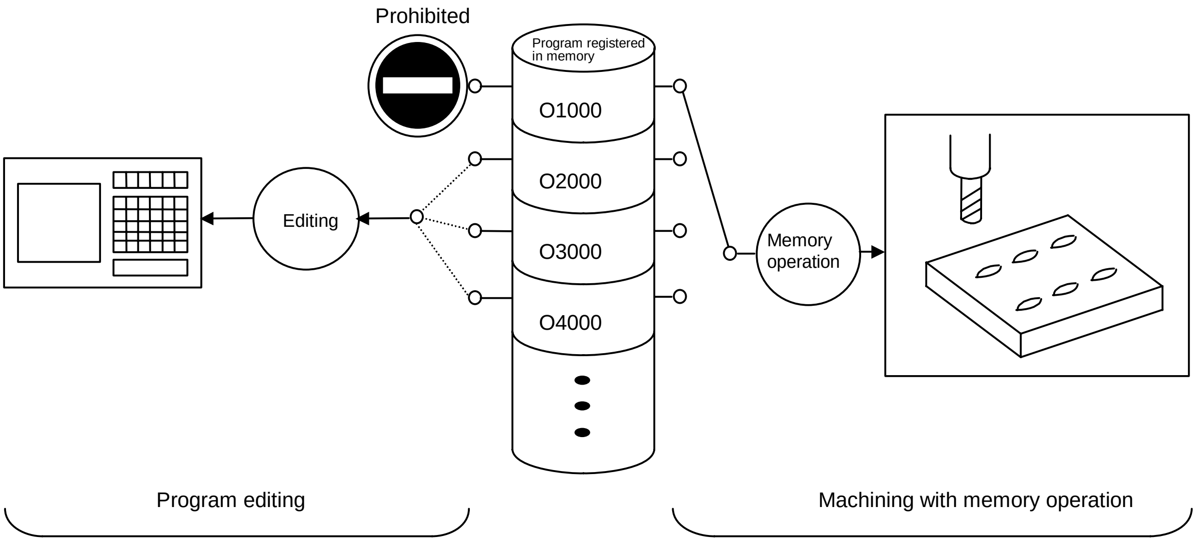
(4) Program editing

(a) Overwriting, inserting and erasing can be done per character.

5.2.2 Background Editing

	E60	E68
M system	○	○
L system	○	○

This function enables one machining program to be created or editing while another program is being run.

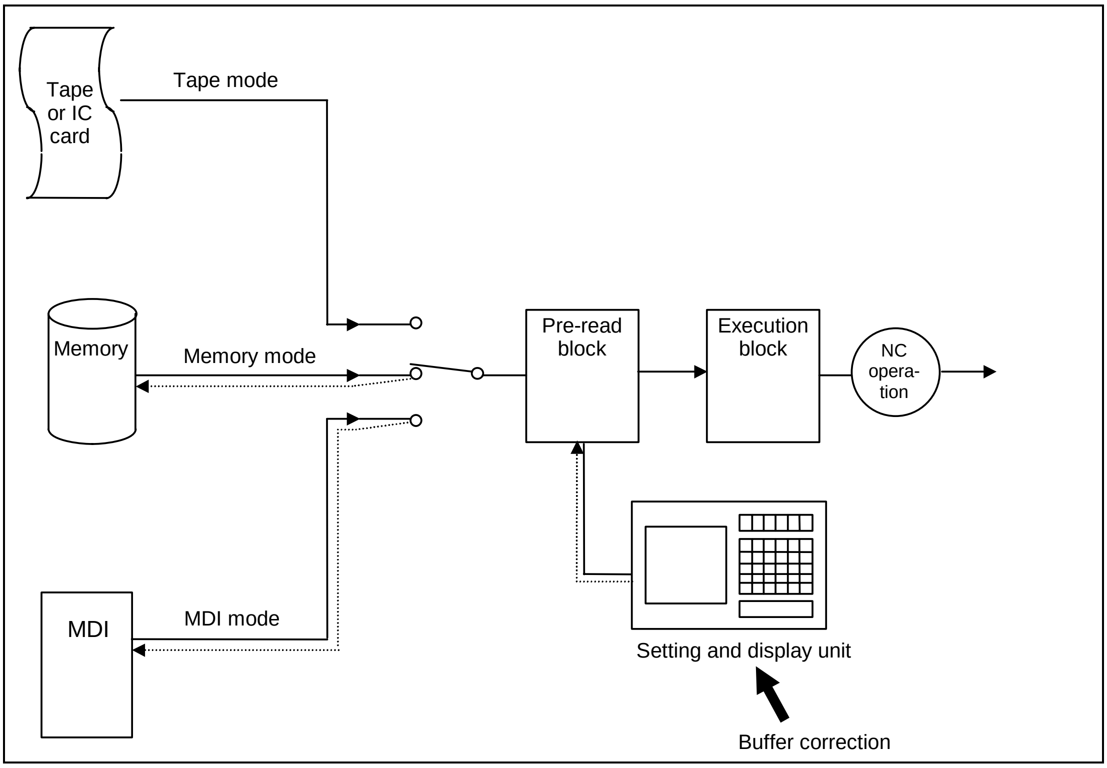


(1) The machining programs being used in memory operation cannot be edited, but can be displayed.
(2) The editing functions such as adding, revising or deleting data can be used at any time for machining programs which are not being used for memory operation.
This makes it possible to prepare and edit the next program for machining, and so the machining preparations can be made more efficiently.
(3) The machining program will not be searched as the operation target even when searched in the edit screen.

5.2.3 Buffer Correction

	E60	E68
M system	○	○
L system	○	○

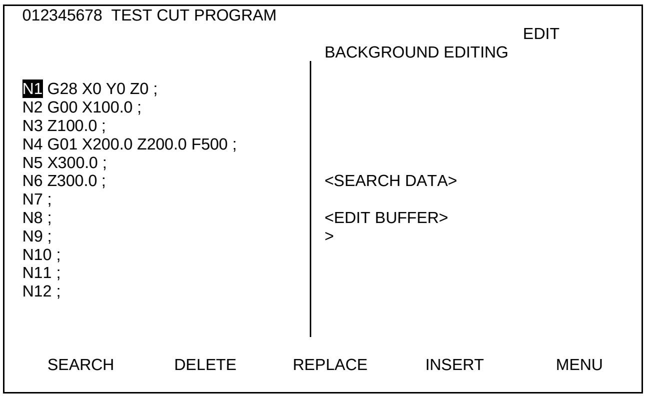
During automatic operation (memory, tape or IC card) or MDI operation, this function initiates single block stop and enables the next command to be corrected or changed.
Only memory operation allows the changes with buffer corrections to be updated in the machining program.
When a program error has occurred, the function enables the block in which the error occurred to be corrected and operation to be resumed without having to perform NC resetting.



5.2.4 Word Editing

	E60	E68
M system	○	○
L system	○	○

In addition to the conventional editing function, this function enables programs to be edited in word increments.
It is also possible to create programs by deleting, replacing and inserting in word increments.



<List of function>

Menu	Function
Delete	The word on which the cursor is positioned can be deleted. (A deleted word can also be un-deleted.)
Replace	The word on which the cursor is positioned can be replaced with editing buffer data. (The same word can also be repeatedly replaced.)
Insert	The editing buffer data can be inserted after the word on which the cursor is positioned. (The same word can also be repeatedly inserted.)
Copy	The word on which the cursor is positioned can be copied into the editing buffer. (The copied word can be used for replacement or insertion.)
Program	A list of the programs is displayed.
Operation search	Program numbers, sequence numbers and block numbers in the foreground can be searched.
Background search	Program numbers, sequence numbers and block numbers for background editing can be searched. New machining programs can be registered as well.
Background exit	Background editing is exited.
Comment	Comments can be set in machining programs.
Word ↓	A downward search for a word is conducted, and the cursor is moved to the word in question. (The same word can be repeatedly searched.)
Word ↑	An upward search for a word is conducted, and the cursor is moved to the word in question. (The same word can be repeatedly searched.)
String ↓	A downward search for a character string is conducted, and the cursor is moved to the words in question. (The same character string can be repeatedly searched.)
String ↑	An upward search for a character string is conducted, and the cursor is moved to the words in question. (The same character string can be repeatedly searched.)
Running program display	The program being run is displayed when the program running display request (PLC) is ON.
Program operation start position setting	The start block can be designated by moving the cursor on the editing screen.

6. Operation and Display

6.1 Structure of Operation / Display Panel

The setting and display unit is configured of the display unit and keyboard unit. When the key switches are pressed, a buzzer sounds allowing the operation to be confirmed visually and audibly.

(1) 7.2-type monochrome LCD display

	E60	E68
M system	□	–
L system	□	–

(2) 9-type monochrome CRT display

	E60	E68
M system	□	–
L system	□	–

(3) 8.4-type color LCD(TFT) display

	E60	E68
M system	□	□
L system	□	□

6.2 Operation Methods and Functions

6.2.1 Memory Switch (PLC switch)

	E60	E68
M system	○	○
L system	○	○

The toggle switches (PLC switches) can be defined on the screen.
The screen can be operated by turning the switches ON/OFF, and the status can be read from the PLC ladder. This screen has been prepared in advance, so the switch names (display on screen) can be defined with the PLC ladder.

6.3 Display Methods and Contents

6.3.1 Status Display

	E60	E68
M system	○	○
L system	○	○

The status of the program now being executed is indicated.

(1) Display of G, S, T, M commands and 2nd miscellaneous command modal values
(2) Feed rate display
(3) Tool offset number and offset amount display
(4) Real speed display (*)

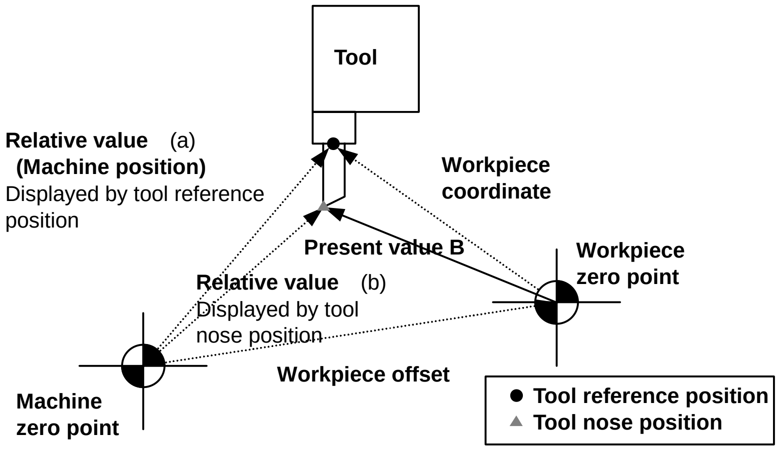
(*) The feed rate of each axis is converted from the final speed output to the drive amplifier, and is displayed. However, during follow up, the speed is converted and displayed with the signals from the detector installed on the servomotor.

6.3.2 Position Display

	E60	E68
M system	○	○
L system	○	○

Position data such as present positions for tools, coordinate positions and workpiece coordinate positions can be displayed.

(1) Present position counter
Each axis' present value including tool length offset amount, tool radius compensation amount and workpiece coordinate offset amount is indicated.
Whether the tool reference position (figure below (a)) or the present position of the tool nose position (figure below (b)) that considers offset, such as tool length offset amount or tool diameter compensation amount, in the tool reference position is applied to the display of the relative value can be selected with the parameter.



(2) Workpiece coordinate counter
The workpiece coordinate system modal number from G54 to G59 and the workpiece coordinate value in the workpiece coordinate system are indicated.
(3) Remaining command counter
The remaining distance of the movement command during the execution (incremental distance from the present value to the end point of the block) is indicated during the automatic start and automatic stop.
(4) Machine position counter
Each axis's coordinate value in the basic machine coordinate system whose zero point is the characteristic position determined depending on the machine is indicated.
(5) Present value B
Each axis' value not including tool length offset amount, tool radius compensation amount and workpiece coordinate offset amount is indicated.
Whether the counter value on the Position screen is expressed with the Present or with the present value B can be selected using parameter.
The present value B can be selected for the counter value indicated on the coordinate value screen using parameter.
(6) Manual interrupt amount counter
The amount moved with the manual mode while the manual absolute switch was OFF is indicated.
On the coordinate value screen, in addition to the manual interrupt amount, the MST display, next command counter and present value B can be selected for the indicated counter using parameter.

6.3.3 Program Running Status display

	E60	E68
M system	○	○
L system	○	○

Program now being executed is displayed.

6.3.4 Setting and Display

	E60	E68
M system	○	○
L system	○	○

The parameters used in controller operations can be set and displayed.

6.3.5 MDI Data Setting and Display

	E60	E68
M system	○	○
L system	○	○

The MDI data having a multiple number of blocks can be set and displayed. As with the editing of machining programs, the MDI programs can be revised using the delete, change and add functions. Operation can be repeated using the programs which have been set.

6.3.7 Clock

	E60	E68
M system	○	○
L system	○	○

The clock is built-in, and the date and time are displayed.
Once the time is set, it can be seen as a clock on the screen.
The clock time can be read/written (read/set) from PLC using the DDB function.

6.3.8 Hardware / Software Configuration Display

	E60	E68
M system	○	○
L system	○	○

This function displays the configuration of the installed hardware and software.

6.3.9 Integrated Time Display

	E60	E68
M system	○	○
L system	○	○

The integrating run time count during each signal of power-ON, automatic operation, automatic start and external integrating run time is ON can be set and displayed. The maximum time displayed is 9999 hours 59 minutes 59 seconds.

Power-ON: Total of all the integrated run times, each starting when the power of the NC control unit is turned ON and ending when it is turned OFF.

Automatic operation: Total of the integrated run times for all machining periods, each starting when the auto start button is pressed in the memory mode and ending when the reset status is established (usually when the M02 / M30 command is designated or the reset button is pressed). (This differs according to PLC machining.)

Automatic start: Total of the integrated run times for all automatic start operations, each starting when the auto start button is pressed in the memory or MDI mode and ending when the feed hold stop or block stop is established or the reset button is pressed.

External integration: Based on the PLC sequence, this is the integrated run time of the signal set by the PLC, and it comes in two types, external integration 1 and external integration 2.

6.3.10 Standard Language

	E60	E68
M system	○13 languages	○13 languages
L system	○13 languages	○13 languages

6.3.11 Additional Languages

Japanese

	E60	E68
M system	○	○
L system	○	○

English

	E60	E68
M system	○	○
L system	○	○

German

	E60	E68
M system	○*	○*
L system	○*	○*

Italian

	E60	E68
M system	○*	○*
L system	○*	○*

French

	E60	E68
M system	○*	○*
L system	○*	○*

Spanish

	E60	E68
M system	○*	○*
L system	○*	○*

Chinese

(1) Chinese (Traditional Chinese characters)

	E60	E68
M system	○*	○*
L system	○*	○*

(2) Chinese (Simplified Chinese characters)

	E60	E68
M system	○	○
L system	○	○

Korean

	E60	E68
M system	○*	○*
L system	○*	○*

Portuguese

	E60	E68
M system	○*	○*
L system	○*	○*

Hungarian

	E60	E68
M system	○*	○*
L system	○*	○*

Dutch

	E60	E68
M system	○*	○*
L system	○*	○*

Swedish

	E60	E68
M system	○*	○*
L system	○*	○*

* : Display only, Manual for each language will be provided from next version.

6.3.12 Screen Saver, Backlight OFF

	E60	E68
M system	○	○
L system	○	○

The screen saver and backlight OFF functions turn off the displays when there is no need to view the screen.

<Screen saver>

This function protects the screen display unit by blanking the screen after the time set in the parameter has elapsed.
Inputting any key causes the screen to reappear.

<Backlight OFF>

This function turns off the backlight in order to extend the service life of the LCD screen's backlight.

6.3.13 Screen Deletion

	E60	E68
M system	○	○
L system	○	○

When there is no need to use a screen for extended periods, the entire screen can be cleared to prevent deterioration of the display unit by the following procedures.

7. Input / Output Functions and Devices

7.1 Input / Output Data

Certain kinds of data handled by the NC system can be input and output between the NC system's memory and external devices.

Machining program input / output (including user macros and fixed cycle macros)

	E60	E68
M system	○	○
L system	○	○

Tool offset data input / output

	E60	E68
M system	○	○
L system	○	○

Common variable input / output

	E60	E68
M system	○	○
L system	○	○

Parameter input / output

	E60	E68
M system	○	○
L system	○	○

History data output

	E60	E68
M system	○	○
L system	○	○

Remote program input

	E60	E68
M system	–	○
L system	–	○

System configuration data output

	E60	E68
M system	○	○
L system	○	○

With this function, NC hardware/software configuration and version information can be output to outside the NC.

7.2 Input / Output I/F

7.2.1 RS-232C I/F

	E60	E68
M system	○	○
L system	○	○

Port 2 of the RS-232C interface can be used. (Port 1 is used for the maintenance.)

Port	Port 2
Transmission speed	to 19.2kbps
Bandshake method	DC code method, RTS/CTS method

It can be used for tape operation, data input/output and printing, etc.
(The application is designated with parameters.)

7.2.2 IC Card I/F

7.2.2.2 I/F for Front IC Card

	E60	E68
M system	–	○
L system	–	○

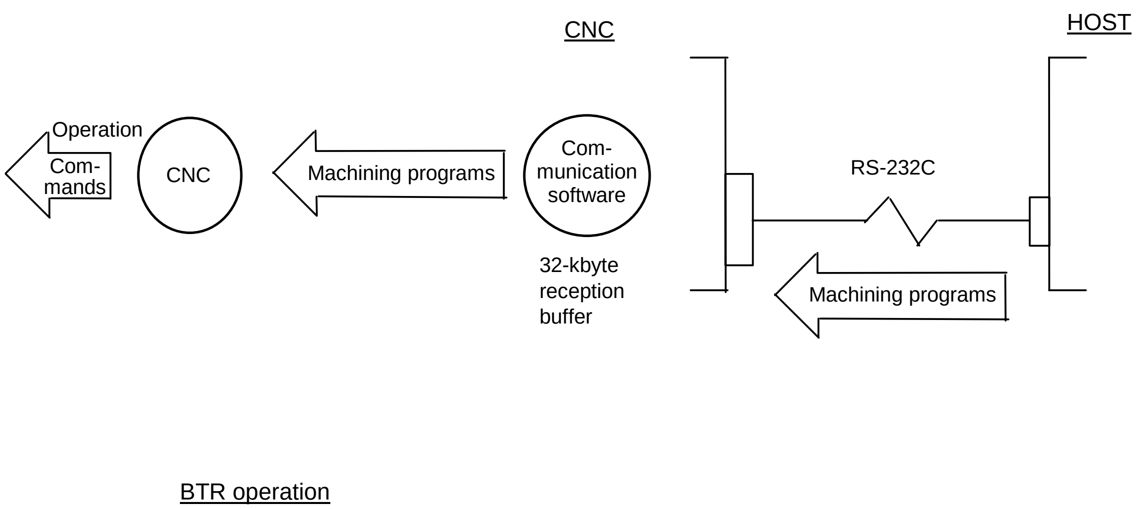
I/F card to use IC card can be attached in front of the display unit and used.
A SanDisk CF card is recommended.

7.3 Computer Link

7.3.1 Computer Link B

	E60	E68
M system	–	○
L system	–	○

This function sends DC1 to the host computer (hereafter abbreviated to "HOST") at the CNC cycle start, and it enables operation to be performed while the machining programs are received from the HOST.
The computer link has a 32-kbyte reception buffer so that operation will be less susceptible to the effects of the data transfer status at the HOST end. This means that when the machine is connected to a HOST capable of transferring data at a high speed (of 38,400 bps), it is possible to perform high-speed machining based on fine-segment data.
The high-speed machining mode option is required for high-speed fine-segment machining.



8. Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

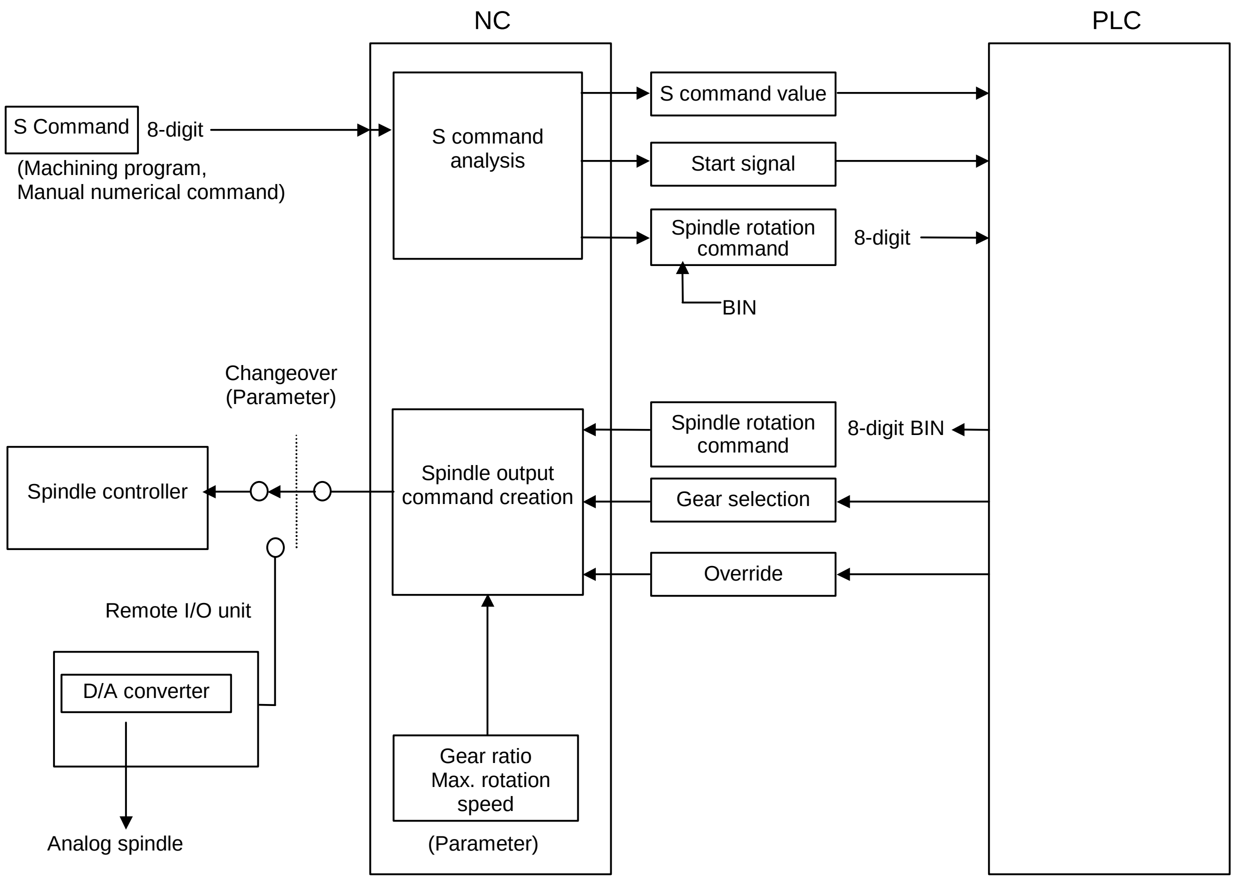
8.1.1 Command / Output

8.1.1.1 Spindle Functions

	E60	E68
M system	○	○
L system	○	○

The spindle rotation speed is determined in consideration of the override and gear ratio for the S command commanded in automatic operation or with manual numerical commands, and the spindle is rotated. The following diagram shows an outline of the spindle control.

When a 8-digit number following address S (S00000000 to S±99999999) is commanded, a signed 32-bit binary data or 8-digit BCD data and start signal will be output to the PLC.
Only one set of S commands can be commanded in one block.
Processing and complete sequences must be incorporated on the PLC side for all S commands.



(1) The override can be designated as 50% to 120% in 10% increments or 0 to 200% in 1% increments (with built-in PLC specifications).
The override is not changed while the spindle stop input is ON, during the tapping mode, or during the thread cutting mode.
(2) The number of gear steps can be commanded up to four steps.
(3) The max. spindle rotation speed can be set for each gear.

8.1.1.2 Spindle Serial I/F

	E60	E68
M system	Δ	○
L system	Δ	○

Digital spindle
This interface is used to connect the spindle driver (SP, SPJ2) with the AC spindle motor.

8.1.1.3 Spindle Analog I/F

	E60	E68
M system	○	□
L system	○	□

Spindle control can be executed using an analog spindle instead of the digital spindle.
In this case, the remote I/O unit DX12x/DX14x and the base I/O unit HR341/HR351 are required.
The analog output voltage is calculated from the present rotation speed regarding the voltage at the max. rotation speed as the maximum analog voltage.
The specifications of the analog voltage output are as follows.

(1) Output voltage ... 0 to ±10V (±5%)
(2) Resolution ... 1/4095 (–12 multiplier of 2)
(3) Load conditions ... 10 kΩ
(4) Output impedance ... 220 Ω

8.1.1.4 Coil Change

	E60	E68
M system	○	○
L system	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.
This is a system under which commands are assigned from the PLC.

8.1.1.5 Automatic Coil Change

	E60	E68
M system	○	○
L system	○	○

Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.
This is a system under which the NC unit switches the coils automatically in accordance with the motor speed.

8.1.2 Speed Control

8.1.2.1 Constant Surface Speed Control

	E60	E68
M system	○	○
L system	○	○

With radial direction cutting, this function enables the spindle speed to be changed in accordance with changes in the radial direction coordinate values and the workpiece to be cut with the cutting point always kept at a constant speed (constant surface speed).

G code	Function
G96	Constant surface speed
G97	Constant surface speed cancel

The surface speed is commanded with an S code. For the metric designation, the speed is commanded with an m/min unit, and for the inch designation, the speed is commanded with a feet/min unit.
In the constant surface speed cancel mode, the S code is a spindle rotation speed command.
The axis for which constant surface speed is controlled is generally the X axis. However, this can be changed with the parameter settings or with address P in the G96 block.

8.1.2.2 Spindle Override

	E60	E68
M system	○	○
L system	○	○

This function applies override to the rotation speed of a spindle or mill spindle assigned by the machining program command during automatic operation or by manual operation. There are two types of override.

(1) Type 1 (code method)
Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 10% increments from 50% to 120%.

(2) Type 2 (number setting method)
Using an external signal, override can be applied to the commanded rotation speed of a spindle or mill spindle in 1% increments from 0% to 200%.

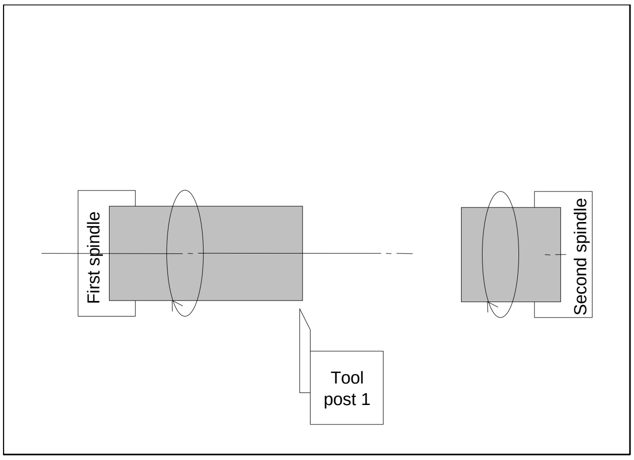
(Note 1) Selection between type 1 and type 2 can be designated by user PLC processing.

8.1.2.3 Multiple-spindle Control

Multi-spindle control is a function that controls the sub-spindle for a machine tool equipped with the second spindle (sub-spindle) in addition to the first spindle (main spindle).
Multi-spindle control I can be switched to multi-spindle control II or vice versa using a parameter and, by so doing, the spindle control method changes.

Multi-spindle control I: Control based on a spindle selection command (such as G43.1) and spindle control command ([S*****;] or [SO=*****;]), etc.

Multi-spindle control II: Control based on an external signal (spindle command selection signal, spindle selection signal) and spindle control command ([S*****;] only), etc.
Spindle selection commands [SO=*****;] cannot be used for this control.



8.1.2.3.1 Multiple-spindle Control I

	E60	E68
M system	–	○
L system	–	○

(1) Spindle selection command

With the spindle selection command (such as G43.1 [G group 20]), it is possible to switch the spindles between 1st spindle and 2nd spindle and determine as to which one of the two axes the subsequent S command (S*****) is applied to.

Command format

G43.1;	First spindle control mode ON
G44.1;	Selected spindle control mode ON; the selected spindle number is set using a parameter.
G47.1;	All spindles simultaneous control mode ON

(2) Spindle control command (Using extended word address (S○=**))**

In addition to using the "S*****" S commands, it is also possible to assign commands in which the 1st spindle and 2nd spindle are differentiated by using the S○=*****.

Command format

S○=*****;

○	:	Number assigned as the spindle number (1: first spindle; 2: second spindle); variables can be designated.
*****	:	Rotational speed or surface speed value assigned by 5-digit analog command; variables can be designated.

8.1.2.3.2 Multiple-spindle Control II

	E60	E68
M system	–	○
L system	–	○

With this function, one S command is used to command to the spindle, and which spindle is selected is decided depending on a signal from the PLC.
A parameter is used to switch between multi-spindle control II and the conventional multi-spindle control I function.

Spindle command selection, spindle selection

The S command for the spindle is output as the rotation speed command to the spindle which has been selected by the spindle selection signal ON from the PLC. The selected spindle rotates at the rotation speed that was output. The spindles that were de-selected by spindle selection signal OFF continue to rotate at the same rotation speed as the speed immediately before their de-selection. As a result, each of the spindles can be made to rotate simultaneously at a different rotation speed.

8.1.3 Position Control

8.1.3.1 Spindle Orientation

	E60	E68
M system	Δ	○
L system	Δ	○

(a) Orient

This function stops the spindle rotation at a certain position when using the digital spindle.
When the orient command is used, the spindle will rotate several times and then stop at the orient point. The orient point is the Z-phase position when using encoder orient (PLG and external encoder/ring sensor).

(b) Multi-point orient

This function performs orientation to a position other than the Z-phase position by inputting a shift amount with the parameter or PLC. The shift amount is 0 to 4095. (Unit: 360°/4096)

(Note 1) Multi-point orient cannot be executed when using the magnetic sensor.

(Note 2) Orient is possible only when the gear ratio is 1:1 for the PLG orient.
(The orient is completed at the PLG encoder's Z-phase, so when using reduction gears, the orient points will be generated at several points during one spindle rotation.)

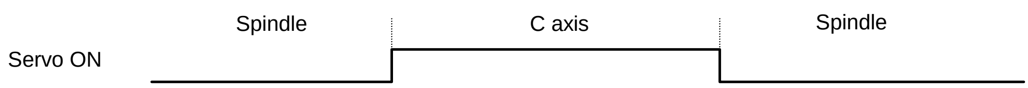
(c) Pre-positioning orient (spindle orient 2)

When the in-position pre-positioning parameter is valid and the second in-position is valid, this function turns ON the orient finish signal as soon as the spindle reaches within the pre-positioning in-position width. It also turns ON the second in-position signal as soon as the spindle reaches within the orient in-position width. (OINP) actually.
Since orient completion can be predicted using this function, it is possible to eliminate the sequence delay time, etc. for tool changes and other such operations, thereby achieving a faster tact time.

8.1.3.2 Spindle Position Control (Spindle / C Axis Control)

	E60	E68
M system	Δ	○
L system	Δ	○

This function enables one digital spindle (SP) to be used also as the C axis (rotary axis) using an external signal.
The C axis servo ON signal is used to switch between the spindle and C axis.



At servo OFF: ---------- Spindle (C axis cannot be controlled).
C axis is in the reference position return incomplete status.

At servo ON: ----------- C axis (spindle cannot be controlled).
C axis is in the reference position return incomplete status.
A parameter is used to initiate reference position return (orient) when servo ON is started.

8.1.3.3 Spindle Synchronization

8.1.3.3.1 Spindle Synchronization I

	E60	E68
M system	–	–
L system	–	○

In a machine with two spindles, this function controls the rotation speed and phase of one spindle (synchronized spindle) in synchronization with the rotation of the other spindle (basic spindle).
It is used in cases where, for instance, workpiece clamped to the basic spindle is to be clamped to the synchronized spindle instead or where the spindle rotation speed is to be changed while one workpiece remains clamped to both spindles.

The synchronous spindle is designated and the start/end of the synchronization are commanded with the G command in the machining program.

Command format

Spindle synchronization control cancel (G113)
This command releases the state of synchronization between two spindles whose rotation has been synchronized by the spindle synchronization command.

```
G113;
```

Spindle synchronization control ON (G114.1)
This command is used to designate the basic spindle and the spindle to be synchronized with the basic spindle, and it places the two designated spindles in the synchronized state.
By designating the synchronized spindle phase shift amount, the phases of the basic spindle and synchronized spindle can be aligned.

G114.1 Hh1 Dd1 Rr1 Aa1 ;

Hh1	: Selects the basic spindle.
Dd1	: Selects the spindle to be synchronized with the basic spindle.
Rr1	: Designates the synchronized spindle phase shift amount.
Aa1	: Designates the spindle synchronization acceleration/deceleration time constant.

8.1.3.3.2 Spindle Synchronization II

	E60	E68
M system	–	○
L system	–	○

In a machine with two spindles, this function controls the rotation speed and phase of one spindle (synchronized spindle) in synchronization with the rotation of the other spindle (basic spindle).
It is used in cases where, for instance, workpiece clamped to the basic spindle is to be clamped to the synchronized spindle instead or where the spindle rotation speed is to be changed while one workpiece remains clamped to both spindles.

The selection of the spindles to be synchronized, the start of the synchronization and other settings are all designated from the PLC.

The spindle synchronization control mode is established by inputting the spindle synchronization control signal. While this mode is established, the synchronized spindle is controlled in synchronization with the rotation speed assigned for the basic spindle.

8.1.3.11 Spindle Holding Power Improvement

	E60	E68
M system	Δ	○
L system	Δ	○

Spindle holding force can be increased when the spindle holding force up request signal is turned ON by ladders while the spindle is stopped for orientation function or spindle/C-axis selected, etc.
When the spindle holding force up request signal is turned ON, the system changes the gain and validates disturbance observer. As a result, the spindle holding force is increased.
Lathe milling, such as D cutting and eccentric drilling, is possible by the spindle holding force up.
For inquiries about spindle drive unit compatible with this function, contact MITSUBISHI.

(Note 1) Depending on the size of load applied to the spindle during machining, a mechanism in which the spindle is mechanically locked may be required. In such a case, consider mounting a mechanical lock on the spindle in advance when designed.

8.2 Tool Functions (T)

8.2.1 Tool Functions

	E60	E68
M system	○	○
L system	○	○

(1) M system

When an 8-digit number following address T (T00000000 to T99999999) is assigned, 8-digit code data and start signal will be output to PLC.
Only one set of T commands can be commanded in a block.
Processing and complete sequences must be incorporated on the PLC side for all T commands.

(Note 1) This function requires a built-in PLC.

(Note 2) There are some screens that cannot display all eight digits.

(2) L system

The command is issued with an 8-digit number following address T (T0 to T99999999).The high-order 6 digits or 7 digits are designated as the tool No., and the low-order 2 digits or 1 digit are designated as the offset No. Which method is to be used is designated with parameters.

Txxxxxxxx

Tool offset No.

Tool No.

Txxxxxxxx

Tool offset No.

Tool No.

The 6-digit (or 7-digit) tool No. code data and start signal will be output to the PLC.
Processing and complete sequences must be incorporated on the PLC side for all T commands.

(Note 1) This function requires a built-in PLC.

(Note 2) There are some screens that cannot display all eight digits.

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

	E60	E68
M system	○	○
L system	○	○

When an 8-digit number (M00000000 to M99999999) is assigned following address M, the 8-digit code data and start signal are output to PLC.

M2-digit BCD output is used for the standard PLC specifications.
When a 2-digit number following address M (M00 to M97) is assigned, the code data and start signal will be output to the PLC.
Apart from the above signals, various special independent signals are also output for the following signals.

M00 : Program stop
M01 : Optional stop
M02 : Program end
M30 : Program end

Respective processing and complete sequences must be incorporated on the PLC side for all M commands from M00 to M97.
M98 and M99 have specific purposes and can not be used.

(Note 1) There are some screens that cannot display all eight digits.

8.3.2 Multiple M Codes in 1 Block

	E60	E68
M system	○	○
L system	○	○

Four sets of M commands can be issued simultaneously in a block.
Apart from the above signals, various special independent signals are also output for the following signals.
Respective processing and completion sequences are required for all M commands included in a block (except M98 and M99).

(Note 1) This function requires a built-in PLC. In this case, the code data and start signal are transferred simultaneously as the M command in the same block from the controller to the PLC, and so high-speed machine control can be done by the PLC processing sequence.

8.3.3 M Code Independent Output

	E60	E68
M system	○	○
L system	○	○

When the M00, M01, M02 or M30 command is assigned during an automatic operation (tape, memory, MDI) or by a manual numerical command, the signal of this function is output. It is turned OFF after the miscellaneous function finishes or by the reset & rewind signal.

Machining program	M code output Independent	Response to controller
M00	M00	Fin1 or Fin2
M01	M01	Fin1 or Fin2
M02	M02	Reset & rewind
M30	M30	Reset & rewind

If movement or dwell command exists in the same block as these M commands, this signal is output upon completion of the movement or dwell command.

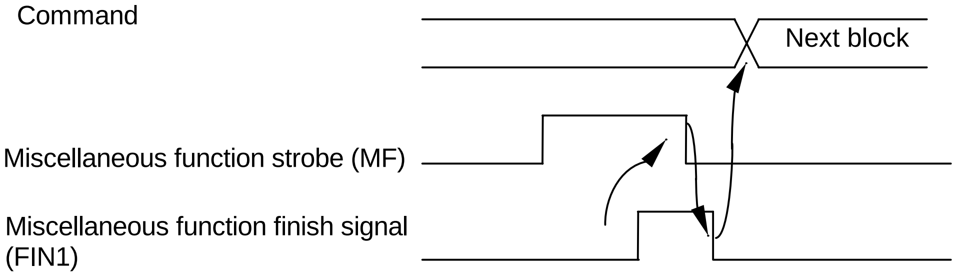
8.3.4 Miscellaneous Function Finish

	E60	E68
M system	○	○
L system	○	○

These signals inform the CNC system that a miscellaneous function (M), spindle function (S), tool function (T) or 2nd miscellaneous function (A, B, C) has been assigned and that the PLC which has received it has completed the required operation. They include miscellaneous function finish signal 1 (FIN1) and miscellaneous function finish signal 2 (FIN2).

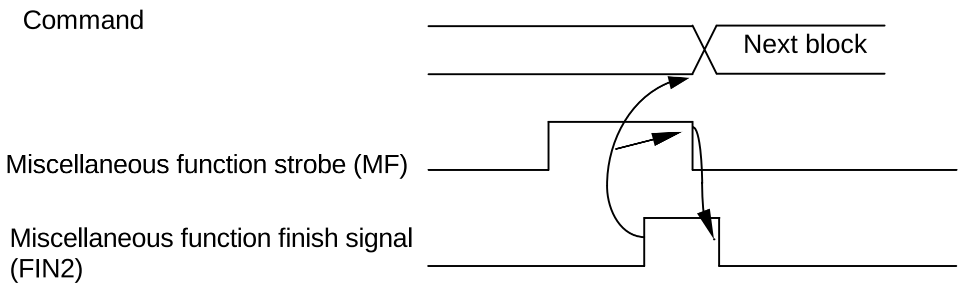
Miscellaneous function finish signal 1 (FIN1)

When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



Miscellaneous function finish signal 2 (FIN2)

When the controller checks that FIN2 is ON, it sets the function strobes OFF and simultaneously advances to the next block. The PLC checks that the strobe signals are OFF and sets FIN2 OFF.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



8.4 2nd Miscellaneous Functions (B)

8.4.1 2nd Miscellaneous Functions

	E60	E68
M system	○	○
L system	○	○

The code data and start signals are output when an 8-digit number is assigned following the address code A, B or C — whichever does not duplicate the axis name being used.
Processing and complete sequences must be incorporated on the PLC side for all 2nd miscellaneous commands.

(Note 1) This function requires a built-in PLC.

(Note 2) There are some screens that cannot display all eight digits.

9. Tool Compensation

9.1 Tool Length / Position Offset

9.1.1 Tool Length Offset

	E60	E68
M system	○	○
L system	○	○

These commands make it possible to control the axis movement by offsetting the position of the end point of the movement command by an offset amount set on the TOOL OFFSET screen.
Using this function, it is possible to offset the difference in distance between the actual position of the machine's tool nose and the program coordinate position made by the tool length and to enhance both the programming and operational efficiency.

(1) M system

G43 G44	Zz1 Zz1	Hh1 Hh1	; ;	Tool length offset can be provided not only for the Z axis but for all other axes which can be controlled in the system (X, Y, etc.).
Offset direction	Offset axis	Offset No.		
G49 ;				Tool length offset cancel

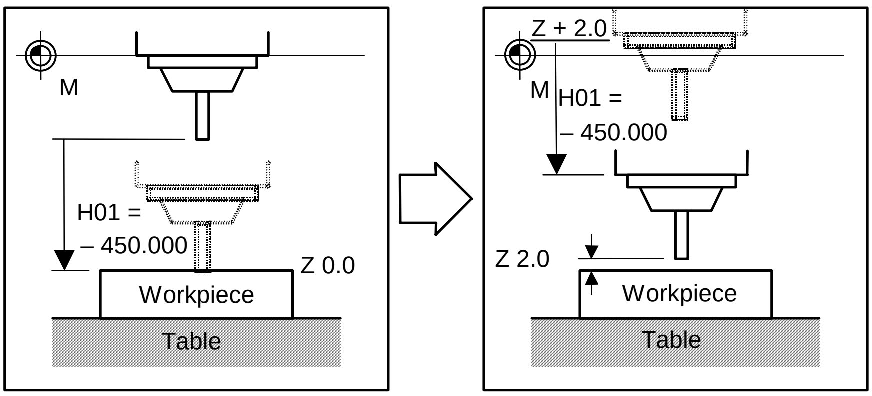
The offset direction is determined by the G command.
G43: Forward direction (z1 + h1)
G44: Reverse direction (z1 – h1)

Offset can be canceled by the following G commands.
G49;
G43 H0;
G44 H0;
(Note) When the tool length offset axis is returned to the reference position, the offset of that axis is canceled.

(Example) Example of tool length offset using a combination with tool length measurement type I

```
G28  X0  Y0  Z0 ;
T01 ;
T02  M06 ;
G91  G00  G43
Z2.0  H01 ;
```

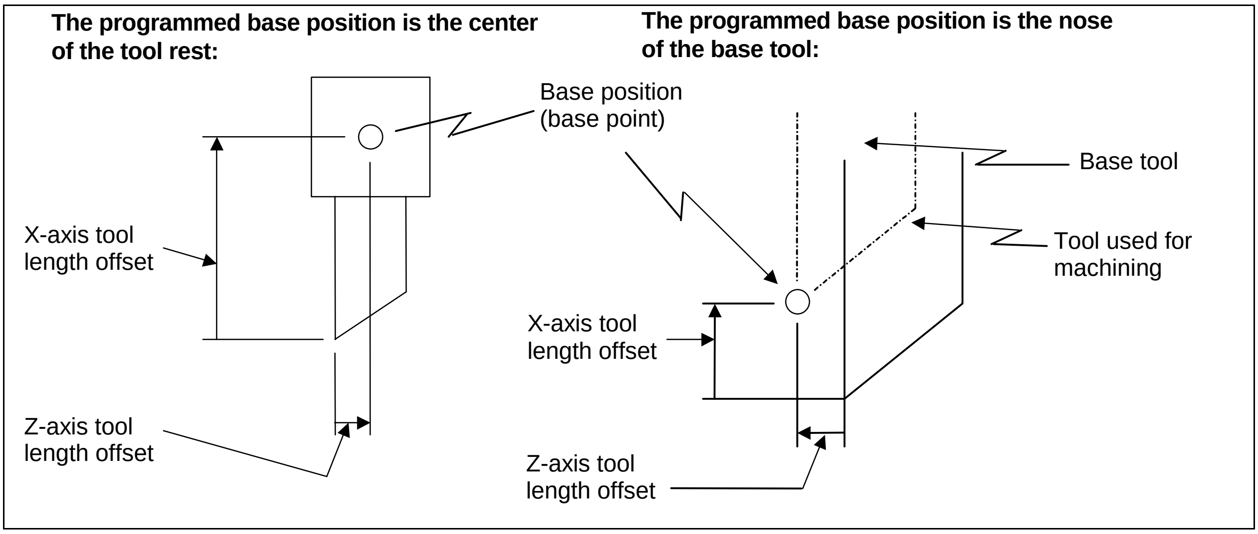
(Note) The tool length offset amount is set as a negative value such as
H01 = –450.000.



(2) L system

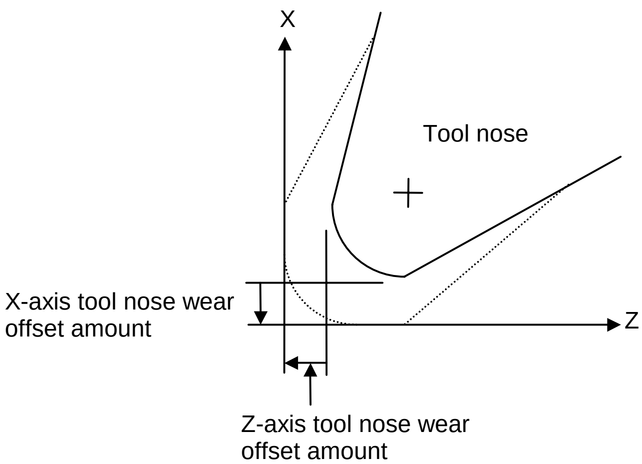
(a) Shape offset

Tool length is offset in reference to the programmed base position. The programmed base position is usually the center of the tool rest or the nose position of the base tool.



(b) Wear offset

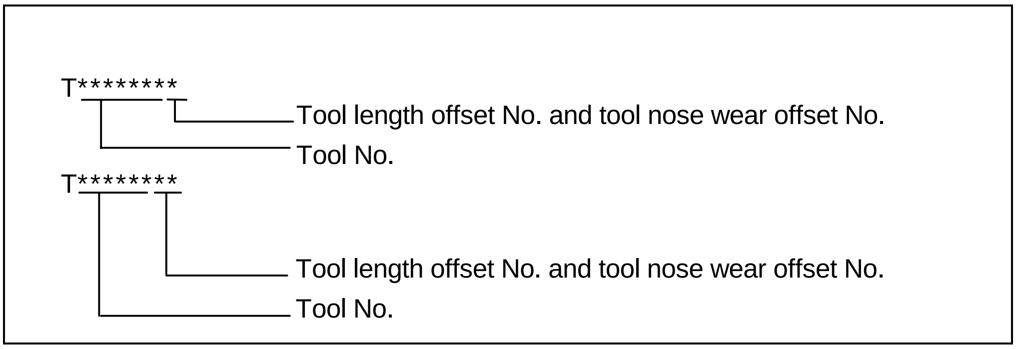
The wear of a tool nose can be offset.



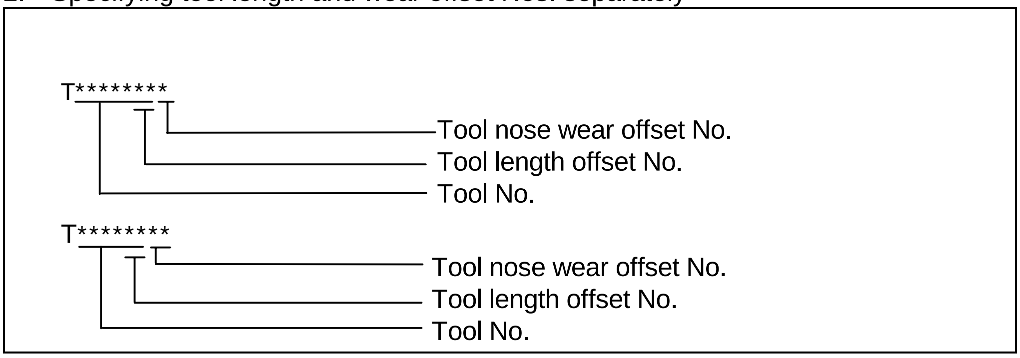
(c) Command format

Tool offset is performed by a T command. It is specified in eight digits following address T. Tool offset is divided into two types: tool length offset and tool nose wear offset. The Nos. of such two types of offsets are specified by a parameter. Also a parameter is used to specify whether the offset Nos. is specified by one or two low-order digits of a T command.

1. Specifying tool length and wear offset Nos. together using one or two low-order digits of the T command



2. Specifying tool length and wear offset Nos. separately



The tool offset for the lathe is valid only for the X and Z axes. If the 3rd axis (Y axis) is added, the tool offset will be validated for the 3rd axis.
With E68, whether to set the additional axis tool offset for the 3rd axis or 4th axis can be changed by the parameter.

9.1.2 Tool Position Offset

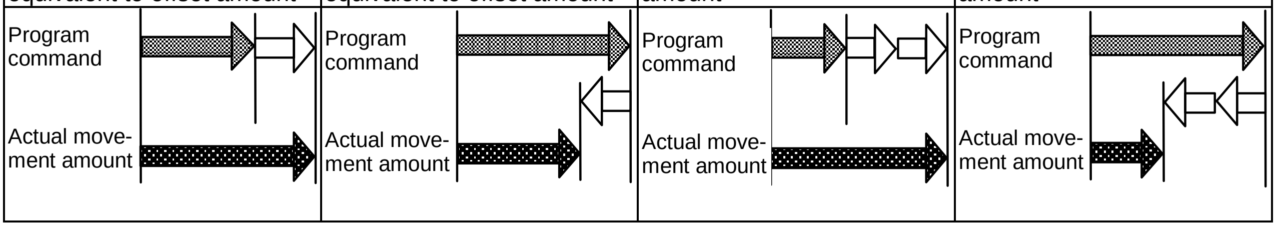
	E60	E68
M system	–	○
L system	–	–

This function uses commands to control the movement by changing the positions of the end points of the movement commands to positions which have been extended or reduced by an amount equivalent to the tool compensation amount.
This function can be used to compensate for the difference in distance between the position where the tool on the machine is actually mounted and the programmed coordinate position based on the tool position and thereby improve the efficiency of both machining and operation.

G45 G00 Xx1 Yy1 Dd1 ;	
G45	: Tool position offset command
Xx1, Yy1	: Movement axes
Dd1	: Offset No.

With tool position offset, the offset operation is performed only for blocks containing a G45 to G48 command.

G45 command	G46 command	G47 command	G48 command
Extension by amount equivalent to offset amount	Reduction by amount equivalent to offset amount	Extension by twice the offset amount	Reduction by twice the offset amount



(1) If the start and end points are on an axis, the radius can be extended or reduced only for one-quarter, one-half and three-quarter arcs.

(2) In the case of absolute commands, the position is extended or reduced in each axial direction from the end point of the previous block along the line of the movement toward the position commanded in the block containing the G45 (or G46, G47 or G48) command.

(3) In the case of simultaneous n axes command, the same amount of offset is applied to all the axes that have the command within the range of the number of the axes which can be simultaneously controlled. Tool position offset is also valid for additional axes.

9.1.3 Tool Offset for Additional Axes

	E60	E68
M system	–	–
L system	○	○

Tool compensation for the L series is valid for the X and Z axes. If an additional axis (such as the Y axis) has been added to these axes, tool compensation is valid for the additional axis.

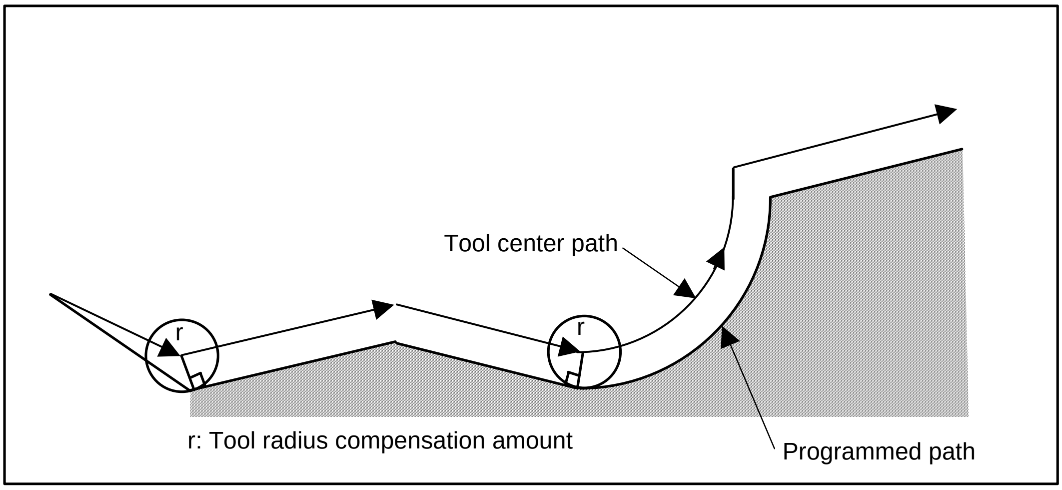
9.2 Tool Radius

9.2.1 Tool Radius Compensation

	E60	E68
M system	○	○
L system	–	–

These commands function to provide tool radius compensation. Through a combination with the G command and D address assignment, they compensate for the actual tool center path either inside or outside the programmed path by an amount equivalent to the tool radius.
The tool path is calculated by the intersection point arithmetic system and, as a result, excessive cut amounts on the inside of corners are avoided.

G code	Function
G38	Vector change during tool radius compensation
G39	Corner arc during tool radius compensation
G40	Tool radius compensation cancel
G41	Tool radius compensation left command
G42	Tool radius compensation right command



The tool radius compensation command controls the compensation from that block in which G41 or G42 is commanded. In the tool radius compensation mode, the program is read up to five blocks ahead including blocks with no movement, and interference check using tool radius is conducted up to three blocks ahead in any of those blocks with movement.

G17 G00 G41 Xx1 Yy1 Dd1 ;

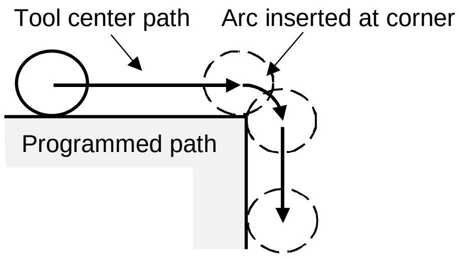
G17	: Compensation plane
G01	: Cutting
G41	: Left compensation
Xx1.Yy1	: Movement axis
Dd1	: Offset No.

The compensation plane, movement axes and next advance direction vector are based on the plane selection command designated by G17 to G19.

G17: XY plane, X, Y, I, J
G18: ZX plane, Z, X, K, I
G19: YZ plane, Y, Z, J, K

An arc is inserted at the corner by the following command during tool radius compensation.

G39 Xx1 Yy1 ;	
Xx1, Yy1	: Movement amount



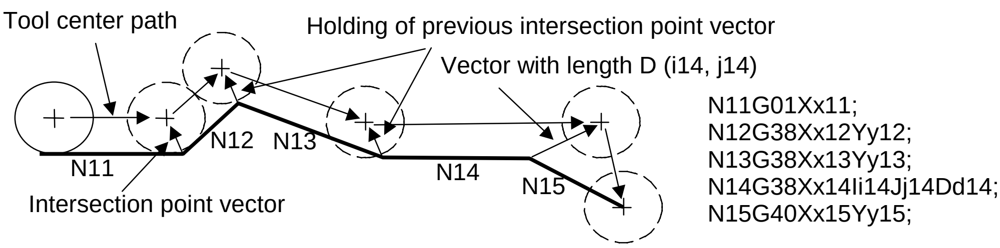
The compensation vector can be changed in following two ways.

G38 Xx1 Yy1 ;	
Xx1, Yy1	: Movement amount

The tool radius compensation vector amount and direction are retained.

G38 Xx1 Yy1 Ii1 Jj1 Dd1 ;	
Xx1, Yy1	: Movement amount
Ii1, Jj1	: Compensation vector direction
Dd1	: Compensation vector length

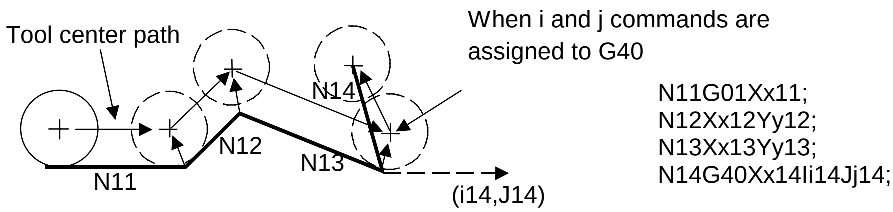
The tool radius compensation vector direction is updated by I and J.



The tool radius compensation is canceled by the following command.

G40 Xx1 Yy1 Ii1 Jj1 ;	
Xx1, Yy1	: Movement amount
Ii1, Jj1	: Compensation vector direction

The vector prior to canceling is prepared by calculating the intersection point with the I and J direction.

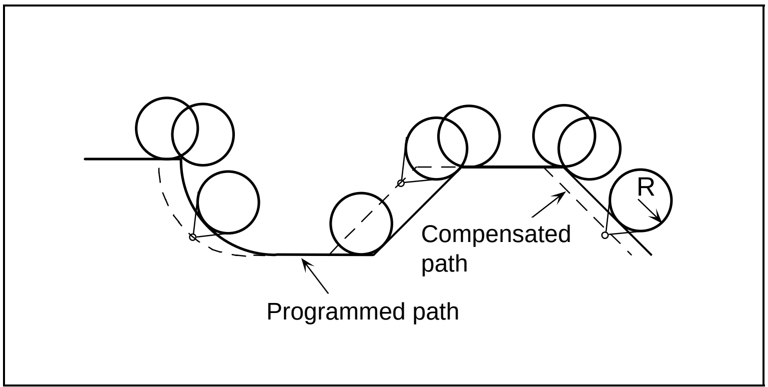


9.2.3 Tool Nose Radius Compensation (G40/41/42)

	E60	E68
M system	–	–
L system	○	○

Corresponding to the tool No., the tool nose is assumed to be a half circle of radius R, and compensation is made so that the half circle touches the programmed path.

G code	Function
G40	Nose R compensation cancel
G41	Nose R compensation left command
G42	Nose R compensation right command



Nose R interference check

In the nose radius compensation mode, the program is read up to five blocks ahead including blocks with no movement, and an interference check using the nose radius is conducted up to three blocks ahead in any of those blocks with movement.

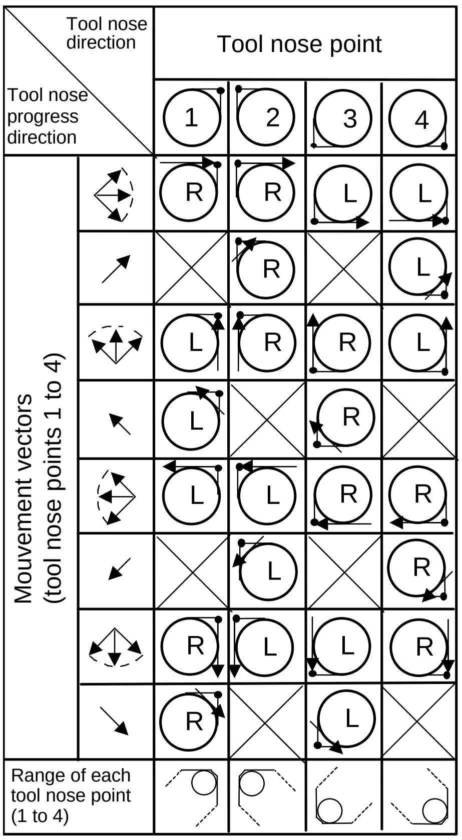
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

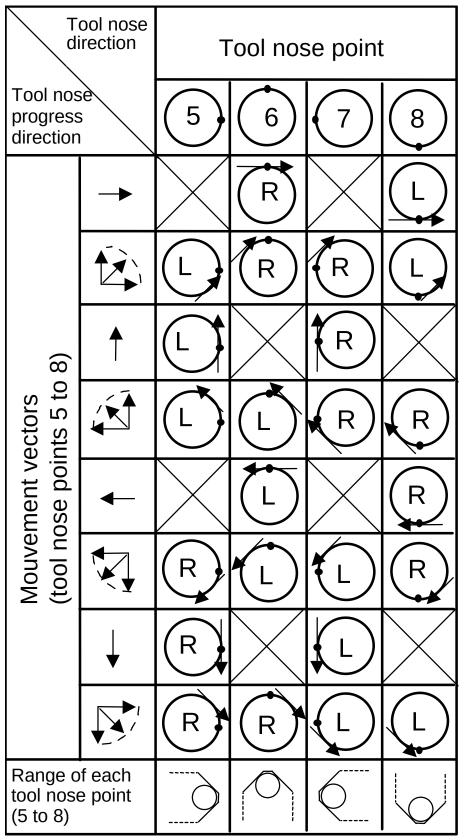
	E60	E68
M system	–	–
L system	○	○

The nose radius compensation direction is automatically determined from the tool nose point and the specified movement vector.

G code	Function
G40	Nose radius compensation cancel
G46	Nose radius compensation ON (Automatic decision of compensation direction)

The compensation directions based on the movement vectors at the tool nose points are as follows:





9.3 Tool Offset Amount

9.3.1 Number of Tool Offset Sets

The number of tool offset sets is as follows.

80 sets

	E60	E68
M system	–	–
L system	○	○

200 sets

	E60	E68
M system	○	–
L system	–	–

400 sets

	E60	E68
M system	–	○
L system	–	–

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

	E60	E68
M system	○	○
L system	○	○

This function registers the tool shape offset and wear offset amounts among the positions of the tools moving in the direction parallel to the control axis. Compensation may encompass two or more axes.

1. Shape offset amount

The tool length offset amount, tool radius compensation amount, nose radius compensation amount, nose radius imaginary tool tip point or tool width can be set as the shape offset amount. The compensation amount that can be set and used differs depending on whether offset amount setting type 1, 2 or 3 is used.

2. Wear offset amount

When the tip of the tool used has become worn, the wear offset amount is used to offset this wear. Types of wear offset amounts include the tool length wear offset amount, tool radius wear compensation amount, and nose radius wear compensation amount.
The wear offset amount can be used with offset amount setting types 2 and 3, and it is added to the shape offset amount for compensation.

(a) Type 1: 1-axis offset amount [M system]

This is the value that is used by rotary tools.
As the tool length offset amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the longitudinal direction of the rotary tool is registered. The tool length offset amount is set as a minus value.

As the tool radius compensation amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the radial direction of the rotary tool is registered. The tool radius compensation amount is set as a plus value.

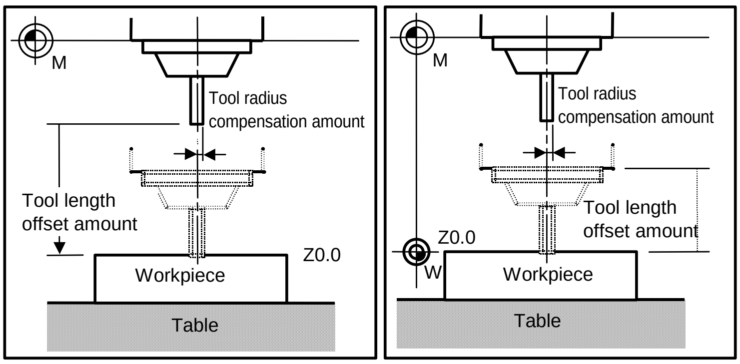
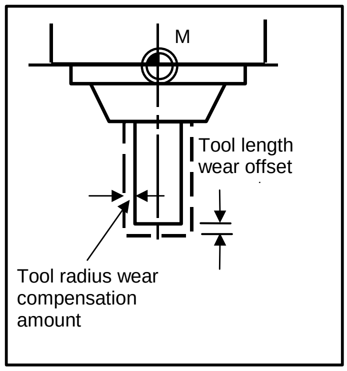
One offset amount data is registered in one offset number, and the offset Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, offset is provided in the form of the tool radius; when it is assigned by an H address command, it is provided in the form of the tool length.

(b) Type 2: 1-axis offset amounts/with wear offset [M system]

As with type 1, type 2 is for the offset amounts used by rotary tools.

With type 2, four kinds of offset amount data are registered in one offset No.: the tool length offset amount, tool length wear offset amount, tool radius compensation amount, and tool radius wear compensation amount.

When an offset No. is assigned by address D as the offset amount, the tool radius is compensated using the amount obtained by adding the tool radius compensation amount and tool radius wear compensation amount. Further, the tool length is offset using the amount obtained by adding the tool length offset amount and tool length wear offset amount.

Figure: Example of how the offset amount is handled when using the type 1 tool length offset amount (Offset types I and II are available for handling offset amounts.)	Wear offset amount when using type 2
Offset type I / Offset type II 	

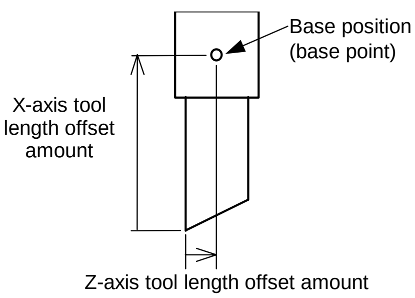
(c) Type 3: 2-axis offset amounts [L system]

Type 3 is for the offset amounts used by non-rotary tools.

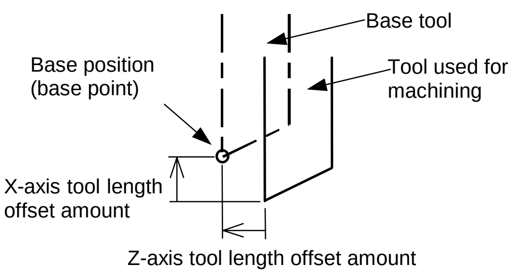
As the offset amounts, the tool length along the X, Y and Z axes and the wear amount along each of these axes, the nose radius and nose radius wear amount, tool tip point P and tool width can be registered.

Offset is provided in the directions of the X, Y and Z axes from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

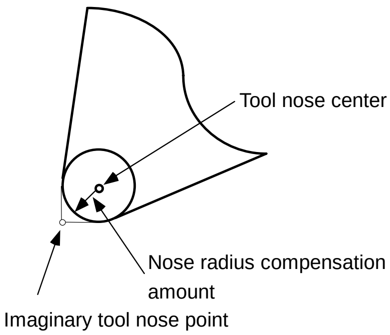
1. The programmed base position is the center of the tool rest:

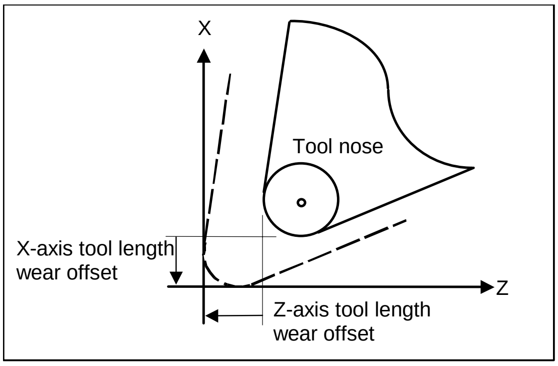


2. The programmed base position is the tip of the base tool:



The tool tip contour arc radius (nose radius) of a non-rotary tool with an arc (nose radius) at its tip is registered as the nose radius offset amount.



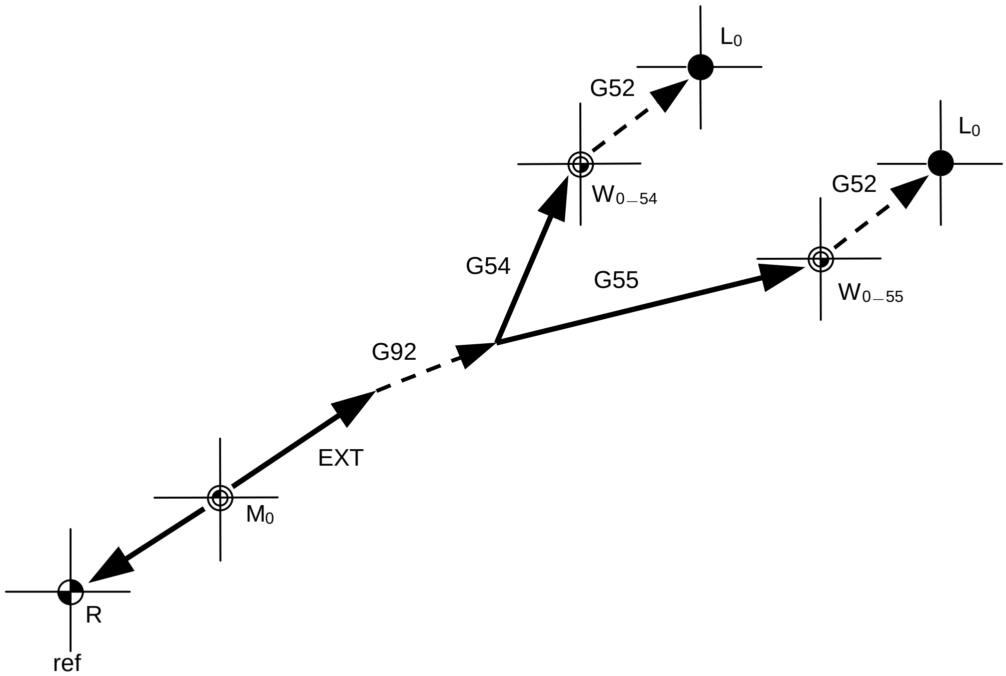


The X-axis tool length offset amount, Z-axis tool length offset amount and nose radius compensation amount are set as plus amounts.
The offset type (1, 2 or 3) is set using a parameter.

10. Coordinate System

10.1 Coordinate System Type and Setting

The coordinate system handled by the NC is shown below.
The points that can be commanded with the movement command are points on the local coordinate system or machine coordinate system.



L_0	Local coordinate system zero point
G52	Local coordinate system offset *1)
W_{0-54}	Workpiece coordinate system zero point (G54)
W_{0-55}	Workpiece coordinate system zero point (G55)
G54	Workpiece coordinate system (G54) offset *1)
G55	Workpiece coordinate system (G55) offset
G92	G92 coordinate system shift
EXT	External workpiece coordinate offset
M_0	Machine coordinate system zero point
ref	Reference position

Offset set with parameters
Offset set with program
(0 when power is turned ON)

*1)The G52 offset is available independently for G54 to G59.

10.1.1 Machine Coordinate System

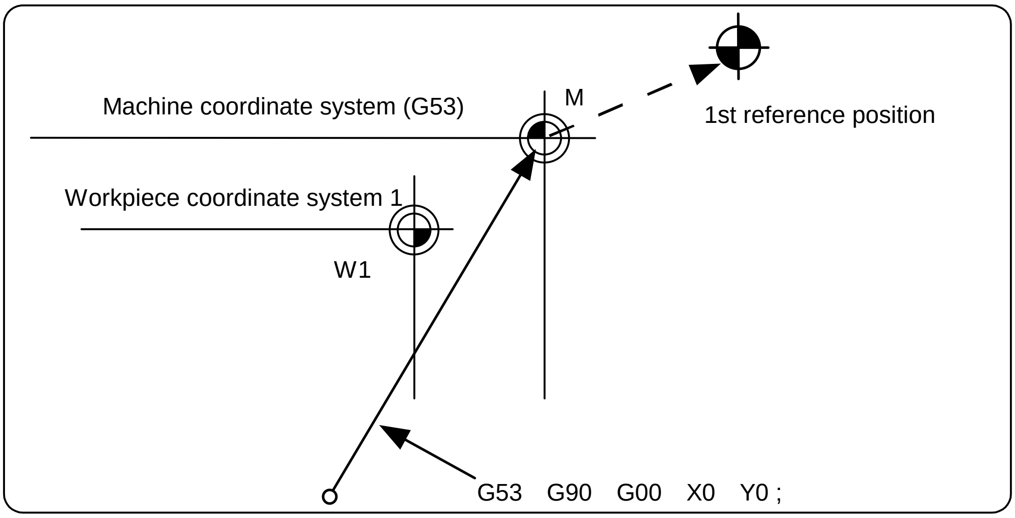
	E60	E68
M system	○	○
L system	○	○

The machine coordinate system is used to express the prescribed positions (such as the tool change position and stroke end position) characteristic to the machine, and it is automatically set immediately upon completion of the first dog-type reference position return after the power has been turned ON or immediately after the power has been turned ON if the absolute position specifications apply.
The programming format for the commands to move the tool to the machine coordinate system is given below.

G53 (G90) (G00) Xx1 Yy1 Zz1 ;

G53	: Coordinate system selection
G90	: Incremental/absolute commands
G00	: Movement mode [M system]
Xx1, Yy1, Zz1	: End point coordinate on the machine coordinate system

If the incremental or absolute commands and movement mode have been omitted, operation complies with the modal command that prevails at the time.
G53 (movement on machine coordinate system) is an unmodal command that is effective only in the block where it is assigned. The workpiece coordinate system being selected is not changed by this command.

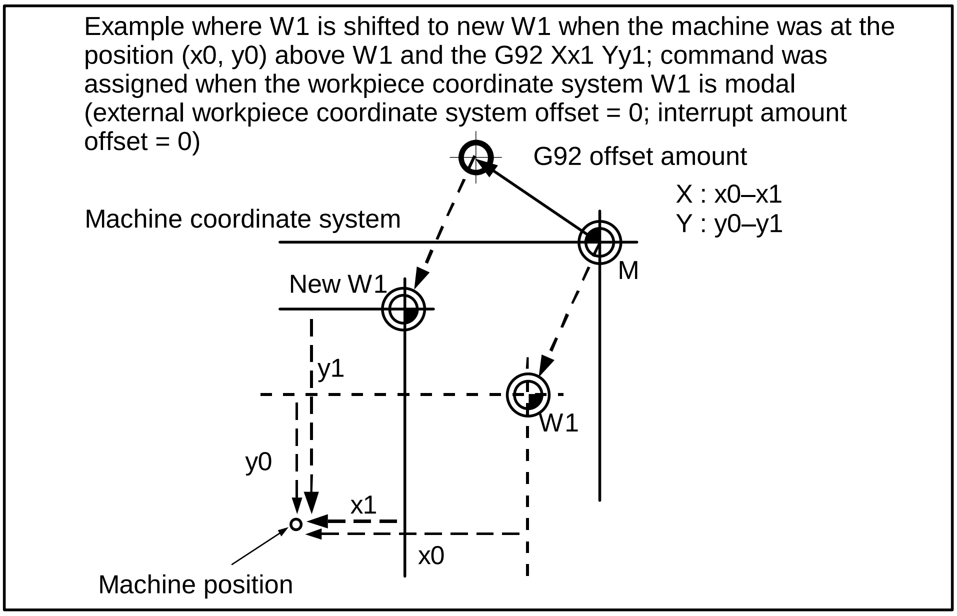


10.1.2 Coordinate System Setting

	E60	E68
M system	○	○
L system	○	○

When a coordinate system setting is assigned using the G92 command, the G92 offset amount is applied so that the machine position in the current workpiece coordinate system is set to the coordinate values assigned by the G92 command, as shown in the figure below, and the workpiece coordinate systems are shifted accordingly. The machine does not move, and all the workpiece coordinate systems from G54 to G59 referenced to the machine coordinate system (or the external workpiece coordinate system if the external workpiece coordinate offset has been set) are shifted.

Offset of coordinate system by G92 coordinate system setting

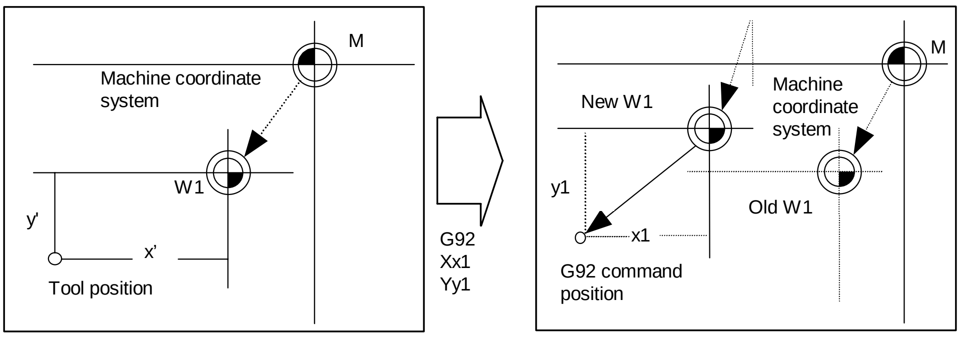


The shifted coordinate system is returned to its original position by dog-type reference position return or the program.

When the coordinate system setting is commanded by G92, all the workpiece coordinate systems from G54 through G59 referenced to the machine coordinate system undergo a shift.

Coordinate system created by automatic coordinate system setting

Coordinate system after coordinate system setting by G92



1. All the workpiece coordinates from G54 to G59 move in parallel.
2. There are two ways to return a shifted coordinate system to its original position.
   a) Dog-type reference position return
   b) Move to machine coordinate system zero point and assign G92 and G53 commands in same block to set the machine coordinate system.

G90 G53 G00 X0 Y0 ;	Positioning at machine coordinate system zero point
G92 G53 X0 Y0 ;	Coordinate system zero setting in machine coordinate system This returns all the workpiece coordinates from G54 to G59 to their original positions.

10.1.3 Automatic Coordinate System Setting

	E60	E68
M system	○	○
L system	○	○

When the tool has arrived at the reference position by means of the first manual or automatic dog type reference position return after the controller power is turned ON, or immediately after the power is turned ON for the absolute position specifications, this function creates the coordinate systems in accordance with the parameters settings.
The coordinate systems created are given below.

1. Machine coordinate system corresponding to G53
2. G54 to G59 workpiece coordinate points
3. Local coordinate systems created under G54 to G59 workpiece coordinate systems

The distances from the zero point of G53 machine coordinate system are set to the controller coordinate related parameters.

10.1.4 Workpiece Coordinate System Selection (6 sets)

	E60	E68
M system	○	○
L system	○	○

When a multiple number of workpieces with the same shape are to be machined, these commands enable the same shape to be machined by executing a single machining program in the coordinate system of each workpiece.
Up to 6 workpiece coordinate systems can be selected.
The G54 workpiece coordinate systems are selected when the power is turned ON or the reset signal which cancels the modal information is input.

G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

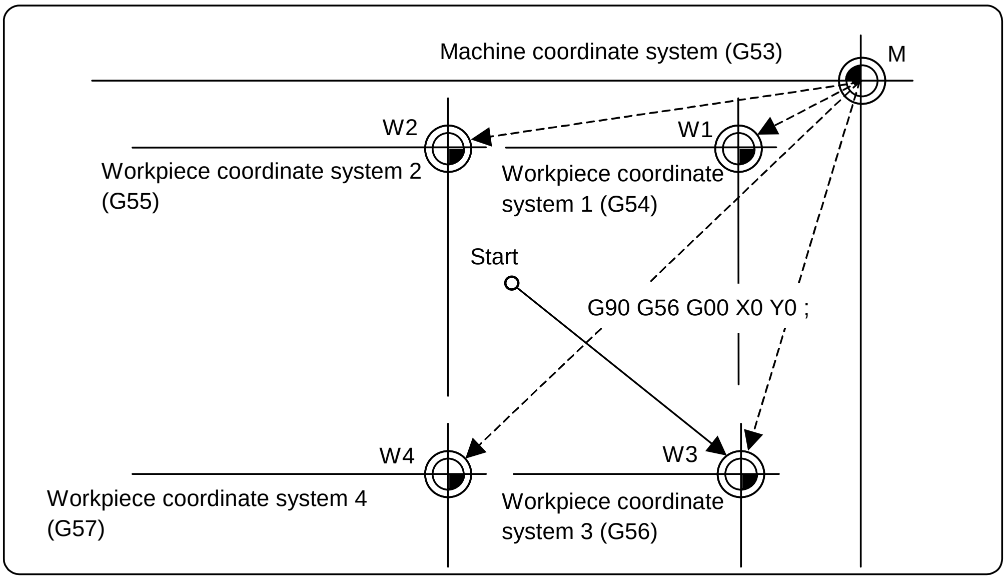
The command format to select the workpiece coordinate system and to move on the workpiece coordinate system are given below.

(G90) G54 G00 Xx1 Yy1 Zz1 ;

(G90) : (Absolute value command)
G54 : Coordinate system selection
G00 : Movement mode
Xx1, Yy1, Zz1 : Coordinate positions of movement destination

The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following three ways:

1. Setting from the screen
2. Setting using commands assigned from the machining program
3. Setting from the user PLC



10.1.5 Extended workpiece coordinate system selection (48 sets) G54.1P1 to P48

	E60	E68
M system	○	○
L system	○	○

In addition to the six workpiece coordinate systems G54 to G59, 48 workpiece coordinate systems can be used by assigning G54.1Pn command.
The command format to select the workpiece coordinate system using the G54.1Pn command and to move on the workpiece coordinate system are given below.

(G90) G54.1Pn G00 Xx1 Yy1 Zz1 ;

G90	: (Absolute command)
G54.1Pn	: Coordinate system selection
G00	: Movement mode
Xx1, Yy1, Zz1	: Coordinate position of end point

The numerical value n of P following G54.1 indicates each workpiece coordinate system. Specify a value between 1 and 48.

The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following three ways:
(1) Setting using the setting and display unit
(2) Setting using commands assigned from the machining program
(3) Setting from the user PLC

(Note) While the G54.1Pn (extended workpiece coordinate system selection) is modal, the local coordinate offset is reduced to zero, and the G52 command cannot be used.

10.1.6 Workpiece Coordinate System Preset (G92.1)

	E60	E68
M system	–	–
L system	–	O

This function presets the workpiece coordinate system, which has been shifted by the programmed command or the manual operation, as the workpiece coordinate system which has been offset by the programmed command (G92.1) from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.

The workpiece coordinate system is shifted from the machine coordinate system when such operations or programmed commands as below have been performed.

- When manual intervention has occurred in the manual absolute OFF status
- When a movement command was performed in the machine lock status
- When movement was initiated by handle interrupt
- When a movement command was performed in the mirror image mode
- When a local coordinate system was set using the G52 command
- When a workpiece coordinate system was shifted using the G92 command

Just as when manual reference position return has been performed, this function presets the workpiece coordinate system that has been shifted once to the workpiece coordinate system that has been offset from the machine zero point by an amount equivalent to the workpiece coordinate offset amount.
Furthermore, whether to preset relative coordinates as well is selected with a parameter.

Command format

G92.1 (G50.3) X0 Y0 Z0 α0 ; (where α is an additional axis)

Designate the addresses of the axes to be preset.
Axes whose addresses have not designated will not be preset.
Depending on the command type, G50.3 command is used in stead.
A program error results when a value other than 0 is commanded.

10.1.7 Local Coordinate System

	E60	E68
M system	○	○
L system	○	○

This function is for assigning a coordinate system on the workpiece coordinate system now being selected. This enables the workpiece coordinate system to be changed temporarily.
The local coordinate system can be selected independently on each workpiece coordinate system G54 to G59.

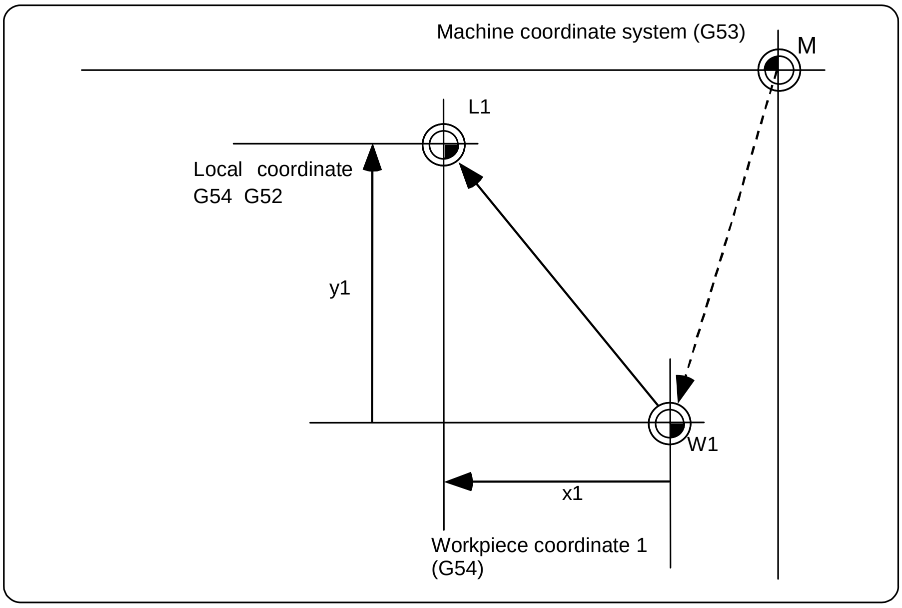
G code	Function
G54 G52	Local coordinate system on the workpiece coordinate system 1
G55 G52	Local coordinate system on the workpiece coordinate system 2
G56 G52	Local coordinate system on the workpiece coordinate system 3
G57 G52	Local coordinate system on the workpiece coordinate system 4
G58 G52	Local coordinate system on the workpiece coordinate system 5
G59 G52	Local coordinate system on the workpiece coordinate system 6

The command format of the local coordinate system is given below.

(G54) G52 Xx1 Yy1 Zz1 ;

G54 : Workpiece coordinate selection
G52 : Local coordinate setting
Xx1, Yy1, Zz1 : Local coordinate offset amount

The local coordinate zero points are provided as distances from the zero point of the designated workpiece coordinate system (local coordinate offset).
In the incremental value mode, the position obtained by adding the local coordinate offset amount to the previously specified offset amount serves as the new local coordinate zero point.
If no workpiece coordinates are designated, the local coordinates will be created on the currently selected workpiece coordinates.
This command is unmodal but the local coordinate system created by G52 is valid until the next G52 command is issued.
The local coordinate system is canceled by the input of the reset signal or by manual or automatic dog-type reference position return.



10.1.8 Coordinate System for Rotary Axis

	E60	E68
M system	○	○
L system	○	○

The axis designated as the rotary axis with the parameters is controlled with the rotary axis' coordinate system.
The rotary axis includes the rotating type (short-cut valid/invalid) and linear type.
The display range is 0 to 359.999° for the rotating type and 0 to ±99999.999° for the linear type.
The range of each coordinate system is 0 to ±359.999° for the rotating type and 0 to ±99999.999° for the linear type.
The rotary axis is commanded with a degree (°) unit regardless of the inch or metric designation.
The rotary axis type is common for all rotary axes and can be set with the parameters.

<table>
<tr><td rowspan="3"></td><td colspan="3">Rotary axis</td><td rowspan="3">Linear axis</td></tr>
<tr><td colspan="2">Rotating type rotary axis</td><td rowspan="2">Linear type rotary axis</td></tr>
<tr><td>Short-cut invalid</td><td>Short-cut valid</td></tr>
<tr><td>Workpiece coordinate position</td><td colspan="2">Displayed in the range of 0° to 359.999°.</td><td colspan="2">Displayed in the range of 0° to ±99999.999°.</td></tr>
<tr><td>Machine coordinate position /relative position</td><td colspan="2">Displayed in the range of 0° to 359.999°.</td><td colspan="2">Displayed in the range of 0° to ±99999.999°.</td></tr>
<tr><td>ABS command</td><td>The incremental amount from the end point to the current position is divided by 360°, and the axis moves by the remainder amount according to the sign.</td><td>The incremental amount from the end point to the current position is divided by 360°, and the axis takes a shortcut to moves by the remainder amount.</td><td colspan="2">In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360°.).</td></tr>
<tr><td>INC command</td><td colspan="4">Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.</td></tr>
<tr><td rowspan="2">Reference position return</td><td colspan="4">Follows the absolute/relative command for a movement to the interim position.</td></tr>
<tr><td colspan="2">Returns to the reference position from the interim position within a 360° movement.</td><td colspan="2">Moves and returns in the reference position direction for the difference from the interim position to the reference position.</td></tr>
</table>

10.1.9 Plane Selection

	E60	E68
M system	○	○
L system	○	○

These G codes are for specifying the planes for the arc, tool radius compensation, coordinate rotation and other such commands.

G17 ; .... Xp-Yp plane designation
G18 ; .... Zp-Xp plane designation
G19 ; .... Yp-Zp plane designation

(1) A parameter can be used to set either the X, Y or Z axis to which the additional axis is to be parallel.
(2) A parameter can be used to set the initialization status (when the power has been turned ON or when the reset status has been entered) to G17, G18 or G19.
(3) The movement commands have no connection with the plane selection.

Example

G19 X100. ;	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, YpZp are selected by G19 and the X axis moves by 100. mm separately from the plane selection.
G17 X100. R50. ;	With these program commands, the Xp-Yp plane is selected by G17 and the arc command is controlled on the X, Y plane by this command.

10.1.10 Origin Set

	E60	E68
M system	○	○
L system	○	○

The coordinate system (current position and workpiece coordinate position) can be set to "0" by operating the screen. This function is the same as the coordinate system setting command " G92 X0 (Y0 or Z0) ; ".

[POSITION]	[WORK(G54)]
X -150.345	X -150.345
Y - 12.212	Y - 12.212
Z - 1.000	Z - 1.000
A - 0.000	A - 0.000

X C.B CAN
Y C.B CAN
Z C.B CAN

[POSOTION]	[WORK(G54)]
X 0.000	X 0.000
Y 0.000	Y 0.000
Z 0.000	Z 0.000
A 0.000	A 0.000

When axes are set to "0" in order, the Y and Z axis can be set by pressing C.B CAN key successively without pressing Y and Z keys.

10.1.11 Counter Set

	E60	E68
M system	○	○
L system	○	○

The position counter display can be change to "0" by operating the screen.
(1) This operation is the same as the operation of "Origin Set", but press INPUT key instead of C.B CAN key.
(2) Only the [POSITION] counter display is changed to "0", and the other coordinate system counter displays are not changed.

10.2 Return

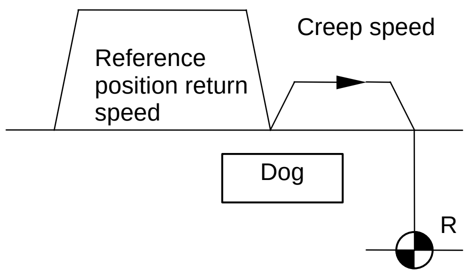
10.2.1 Manual Reference Position Return

	E60	E68
M system	○	○
L system	○	○

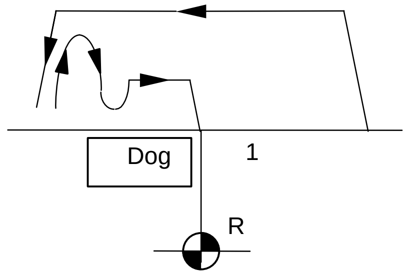
This function enables the tool to be returned manually to the position (reference position) that is characteristic to the machine.

(1) Return pattern to reference position

(a) Dog type

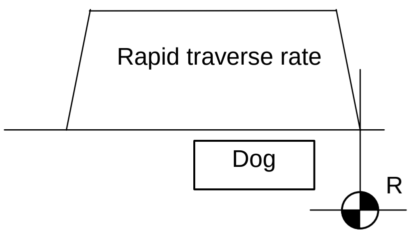


When starting in same direction
as final advance direction



When starting in opposite direction
as final advance direction

(b) High-speed type



(2) Differences according to detection method

	First return after power ON	Second return and following
Incremental position detection method	Dog-type	High-speed
Absolute position detection method	High-speed	High-speed

10.2.2 Automatic 1st Reference Position Return

	E60	E68
M system	○	○
L system	○	○

The machine can be returned to the first reference position by assigning the G28 command during automatic operation. If the interim point is commanded, the machine is moved up to that point by rapid traverse so that it is positioned and then returned separately for each axis to the first reference position.
Alternatively, by assigning the G29 command, the machine can be first positioned separately for each axis at the G28 or G30 interim point, and then positioned at the command position.

G code	Function
G28	Automatic 1st reference position return
G29	Start position return (The tool first returns to the interim position of the 1st reference position return start from the 1st reference position, and then is positioned at the position designated in the program.)

The G28 programming format is given below.

G28 Xx1 Yy1 Zz1 ;

G28 : Return command
Xx1, Yy1, Zz1 : Return control axes (interim point)

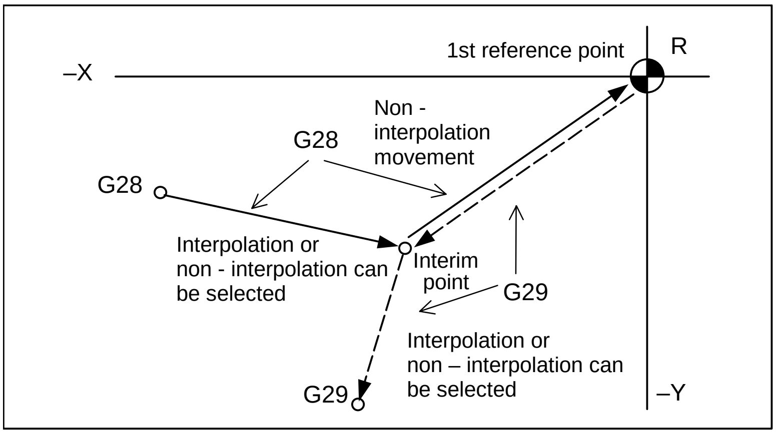
Each axis is first positioned by rapid traverse to the position (interim point) assigned for the assigned axis and then is returned independently to the 1st reference position.

The G29 programming format is given below.

G29 Xx1 Yy1 Zz1 ;

G29 : Return command
Xx1, Yy1, Zz1 : Return control axes (assigned position)

The tool is first moved by rapid traverse to the interim position that is passed through with G28 or G30, and is then positioned by rapid traverse at the position assigned by the program.



If the position detector is for the incremental detection system, the first reference position return for the first time after the NC power has been turned ON will be the dog-type. However, whether the second and subsequent returns are to be the dog type or the high-speed type can be selected by designating a parameter.
The high-speed type is always used when the position detector is for the absolute position detection system.

(Note 1) The automatic 1st reference position return pattern is the same as for manual reference position return.

(Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.

(Note 3) If, at the time of the first reference position return, the tool radius compensation or nose radius compensation has not been canceled, it will be temporarily canceled by the movement to the interim point. The compensation is restored by the next movement after the return.

(Note 4) If, at the time of the first reference position return, the tool length offset has not been canceled, the offset will be canceled by the movement from the interim point to the first reference position, and the offset amount will also be cleared. It is possible to cancel the tool length offset temporarily using a parameter instead. In this case, however, the offset is restored by the next movement command.

(Note 5) Interpolation or non-interpolation can be selected using a parameter for the movement up to the G28 interim point or for the movement from the G29 interim point to the command point. Non-interpolation applies for movement from the G28 interim point to the reference position and movement up to the G29 interim point.

(Note 6) The machine will not stop at the interim point even when a single block is selected.

10.2.3 2nd, 3rd, 4th Reference Position Return

	E60	E68
M system	○	○
L system	○	○

As with automatic 1st reference position return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference positions) characteristic to the machine. The 2nd, 3rd and 4th reference positions can be set by parameters.

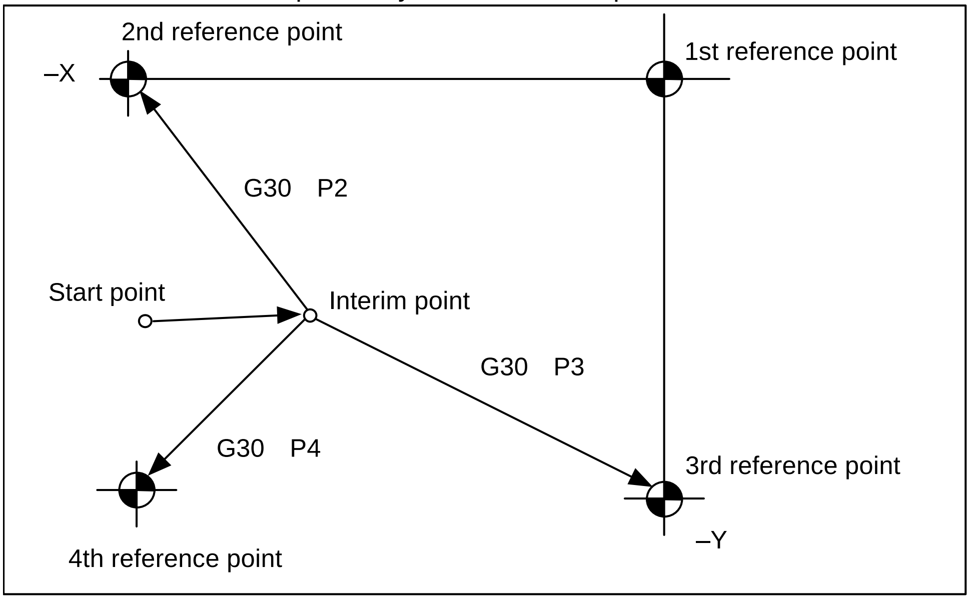
G code	Function
G30 P2	2nd reference position return
G30 P3	3rd reference position return
G30 P4	4th reference position return

The G30 programming format is given below.

G30 Xx1 Yy1 Zz1 Pp1 ;

G30	: Return command
Xx1, Yy1, Zz1	: Return control axes (interim point)
Pp1	: Return position No.

The tool is first positioned by rapid traverse to the interim point commanded for the assigned axis and then is returned independently to the reference position.



(Note 1) The second reference position return is performed if the P command is omitted.

(Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.

(Note 3) If, at the time of the reference position return, the tool radius compensation has not been canceled, it will be temporarily canceled by the movement up to the interim point. The compensation is restored by the next movement command after the return.

(Note 4) If, at the time of the reference position return, the tool length offset has not been canceled, it will be canceled and the offset amount also cleared upon completion of reference position return. The tool length offset can also be canceled temporarily using a parameter. In this case, however, the tool offset is restored by the next movement command.

(Note 5) Whether interpolation or non-interpolation is to apply to the movement up to the interim point can be selected using a parameter. Non-interpolation applies for movement from the interim point to each of the reference positions.

(Note 6) The machine will not stop at the interim point even when a single block is selected.

10.2.4 Reference Position Verification

	E60	E68
M system	○	○
L system	○	○

By commanding G27, a machining program, which has been prepared so that the tool starts off from the reference position and returns to the reference position, can be checked to see whether the tool will return properly to the reference position.
The G27 programming format is given below.

G27 Xx1 Yy1 Zz1 Pp1 ;

G27 : Verification command
Xx1, Yy1, Zz1 : Return control axes
Pp1 : Verification No.
P1 : 1st reference position verification
P2 : 2nd reference position verification
P3 : 3rd reference position verification
P4 : 4th reference position verification

The assigned axis is first positioned by rapid traverse to the commanded position and then, if this is the reference position, the reference position arrival signal is output.

When the address P is omitted, the first reference position verification will be applied.

(Note 1) The number of axes for which reference position verification can be performed simultaneously depends on the number of simultaneously controlled axes.

(Note 2) An alarm results unless the tool is positioned at the reference position upon completion of the command.

(Note 3) Whether interpolation or non-interpolation is to apply to the movement can be selected using a parameter.

10.2.5 Absolute Position Detection

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

The absolute position detection function holds the relation of the actual machine position and the machine coordinates in the controller with a battery even when the power is turned OFF. When the power is turned ON again, automatic operation can be started without executing reference position return. (High-speed return will always be used for the reference position return command.)
For the absolute position detection method, there are two methods such as the dog-type and dog-less type according to how the zero point is established.

Method		Details	Establishment of zero point	Adjustment of zero point position
Dog-type		Same method as incremental detection dog-type	Zero point is established with dog-type reference position return completion.	The data is set in the parameter of zero point shift amount.
Dog-less type	Marked point method	The zero point position is set from the screen.	The zero point is established by input from the zero point initialization screen.	The value equivalent to the shift amount is set in the zero point initialization screen.
	Machine stopper method	The zero point is established by pressing the machine against a set point on the machine.	The zero point is established when a torque limit is applied on the servo and the torque limit is reached by pressing against the machine stopper.	The value equivalent to the shift amount is set in the zero point initialization screen.

Diagnosis during absolute position detection

(1) The machine position at power OFF and ON can be confirmed on the absolute position monitor screen.
(2) If the amount that the axis is moved during power OFF exceeds the tolerable value (parameter), a warning signal will be output.
(3) An alarm will be output if the absolute position information is lost.
(4) An alarm will be output if the voltage of the battery for backing up the absolute position data drops.

10.2.6 Tool Exchange Position Return

	E60	E68
M system	O	O
L system	O	O

By specifying the tool change position in a parameter and also assigning a tool change position return command in a machining program, the tool can be changed at the most appropriate position. The axes for which returning to the tool change position is performed and the order in which the axes begin to return can be changed by commands.

> **G30.n ;**
>
> n = 1 to 6 : Specify the axes that return to the tool change position and the order in which they return. (For L system, n = 1 to 5)

Command and return order

[M system]

Command	Return order
G30.1	Z axis → X axis • Y axis (→ additional axis)
G30.2	Z axis → X axis → Y axis (→ additional axis)
G30.3	Z axis → Y axis→ X axis (→ additional axis)
G30.4	X axis → Y axis • Z axis (→ additional axis)
G30.5	Y axis → X axis • Z axis (→ additional axis)
G30.6	X axis • Y axis • Z axis (→ additional axis)

[L system]

Command	Return order
G30.1	X axis only (→ additional axis)
G30.2	Z axis only (→ additional axis)
G30.3	X axis → Z axis (→ additional axis)
G30.4	Z axis → X axis (→ additional axis)
G30.5	X axis • Z axis (→ additional axis)

(Note1) An arrow (→) indicates the order of axes that begin to return. A period (•) indicates that the axes begin to return simultaneously.
Example: "Z axis → X axis" indicates that the Z axis returns to the tool change position, then the X axis does.
(Note2) G30.6 is only for the M system.

The tool change position return ON/OFF for the additional axis can be set with parameter for the additional axis. For the order to return to the tool change position, the axes return after the standard axis completes the return to the tool change position (refer to above table).
The additional axis cannot return to the tool change position alone.

10.2.7 C Axis Reference Position Return

	E60	E68
M system	–	–
L system	Δ	O

This function is used to carry out the position control for the spindle with the axis motor, and applied to the machine that can switch the spindle motor connected with the spindle to the axis motor.
The C axis (rotation axis) is generally used for the axis motor, and the specification will be the same as the normal C axis control after switching.
As for the reference position return during the C axis connection, either the normal dog-type reference position return with the C axis or the Z-phase pulse type reference position return with the spindle encoder can be selected with parameter.

Reference position return method
- Dog type (with C axis detector)
- Z-phase pulse type (with spindle encoder)

(Supplements)

(1) Reference position return method

The Z-phase pulse type is applied in the first reference position return after the servo OFF for the C axis (generally means changing to the spindle). The high-speed type is applied in the (second or later) reference position return after the Z-phase pulse type reference position return.

(2) Z-phase pulse type reference position return

When there is the reference position return command, the spindle is rotated until the Z-phase pulse of the spindle encoder is detected, and then stopped. (Figure 1 (1))
Encoder gear ratio 1:1 The Z-pulse is detected within one rotation.
Encoder gear ratio 1:2 The Z-pulse is detected within two rotations.

Next, in order to catch the change of the pulse position and improve the accuracy of the remaining distance, the spindle is rotated with G28crp (approach speed) and then stopped again. (Figure 1 (2))
Set as follows: Standard value G28crp = 8 (°/min) (Encoder gear ratio 1:1)

For the spindle, one rotation to the point where the Z-pulse is detected is made with G28rap (G28 rapid traverse rate). (Figure 1 (3)) This point is applied to the reference position. If the reference position shift amount is included, the spindle is rotated to the point of the figure 1 (4) and then stopped.

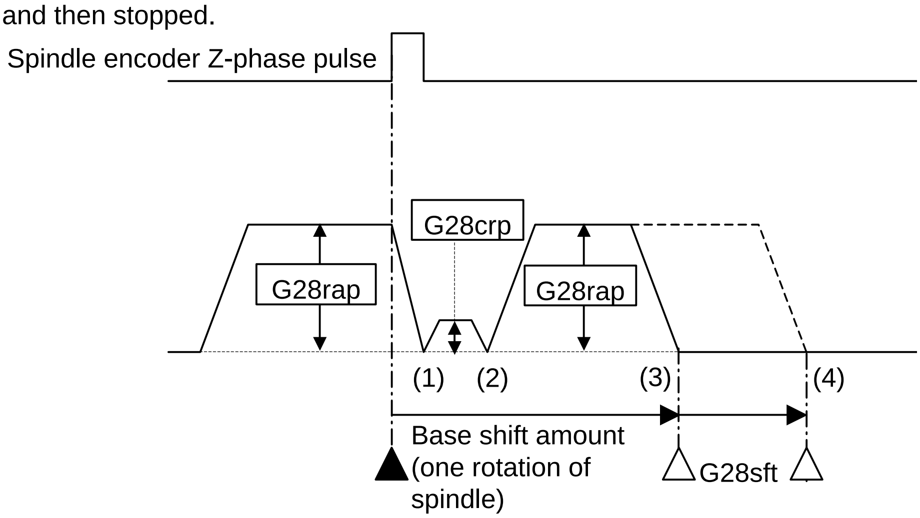


Figure 1 Z-phase pulse type reference position return

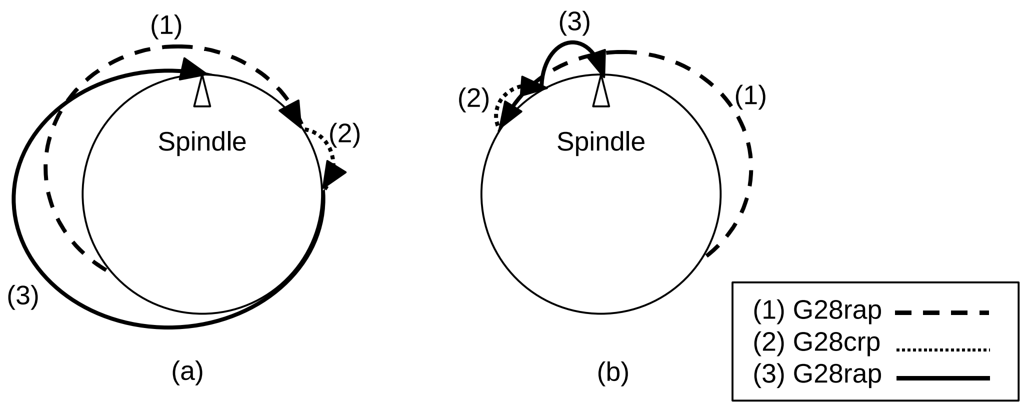


Figure 2 Operation of spindle

With the figure 2 (a), the rotation of the spindle that the G28 rapid traverse is performed in the forward direction (the same direction as the parameter #2030) is looked from the front. With the figure 2 (b), the rotation of the spindle that the G28 rapid traverse is performed in the opposite direction (the opposite direction to the parameter #2030) is looked from the front.

(3) High-speed type reference position return

After the Z-phase pulse type reference position return, the high-speed type reference position return is applied in the second or later reference position return.

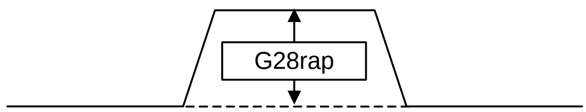


Figure 3 High-speed type reference position return

11. Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

	E60	E68
M system	○	○
L system	○	○

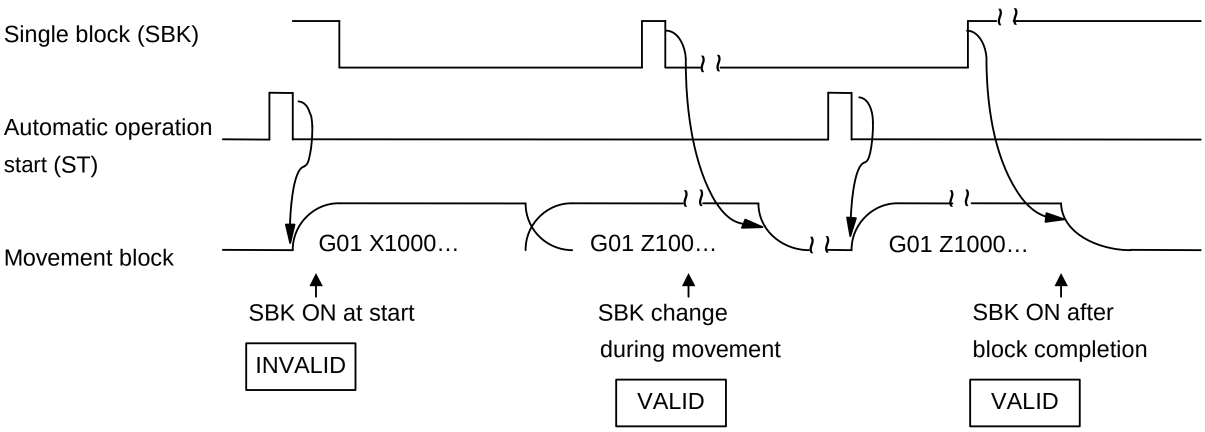
When "/" (slant code) is programmed at the head of a block, and the optional block skip input signal from the external source is turned ON for automatic operation, the block with the "/" code is skipped. If the optional block skip signal is turned OFF, the block with the "/" code will be executed without being skipped.

Programming example	Optional block skip	
	Switch OFF	Switch ON
N1	N1	N1
N2	N2	N2
N3	N3	N3
/N4	N4	
/N5	N5	
N6	N6	N6
N7	N7	N7
:	:	:

11.1.3 Single Block

	E60	E68
M system	○	○
L system	○	○

The commands for automatic operation can be executed one block at a time (block stop) by turning ON the single block input signal. When the single block input signal is turned ON temporarily during continuous operation, the machine will stop after that block has been executed.
When operation is switched to another automatic operation mode (for example, memory operation mode to MDI operation mode) during continuous operation, the machine will stop after that block has been executed.



11.2 Program Test

11.2.1 Dry Run

	E60	E68
M system	○	○
L system	○	○

F code feed commands for automatic operation can be switched to the manual feed rate data of the machine operation board by turning ON the dry run input signal.

Command	Dry run switch ON	
	Rapid traverse selection switch OFF	**Rapid traverse selection switch ON**
G00, G27, G28, G29, G30, G60	Manual feed rate	Rapid traverse rate
G01, G02, G03	Manual feed rate	Cutting clamp speed

11.2.2 Machine Lock

	E60	E68
M system	○	○
L system	○	○

Operation can be executed with the machine in the servo lock status for that axis when the machine lock input signal is turned ON.
The feed rate in the machine lock status is the command speed.
The M, S, T and B commands are executed as usual and operation is completed by returning the FIN signal.

(1) Reference position return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but the block is completed when the interim point is reached.
(2) Machine lock is effective in the signal status applying when the axis has stopped.
(3) Block stop will be applied if the machine lock signal is turned ON and OFF or OFF and ON during automatic operation.

All axes will be simultaneous with the standard PLC.

11.2.3 Miscellaneous Function Lock

	E60	E68
M system	○	○
L system	○	○

The M, S, T and B (2nd miscellaneous function) output signals are not output to the machine or PLC when the miscellaneous function lock signal of external input is turned ON. This function can be used when checking only the movement commands in a program check.
The start signals of the M command are output for the M00, M01, M02 and M30 commands, and so a completion signal must be returned.

(1) Fixed cycle spindle functions containing an S code and any M, S, T or B function assigned by a manual numerical command or in automatic operation will not be executed. The code data and strobe (MF, SF, TF, BF) outputs are stopped.
(2) If this signal is set ON after the code data has already been output, the output is executed as it would normally be executed until the end (until FIN1 or FIN2 is received and the strobe is turned OFF).
(3) Even when this signal is ON, the M00, M01, M02 and M30 commands among the miscellaneous functions are executed, and the decode signal, code data and strobe signals are also output as they would be normally.
(4) Any miscellaneous functions that are executed only inside the controller and not output (M96, M97, M98, M99) are executed as they would be normally even if this signal is ON.

11.2.4 Graphic Check

	E60	E68
M system	○	○
L system	○	○

The movement path of the machine tool can be monitored and traced, and the path of machining programs can be traced and checked using the check and tracing functions based on processing inside the controller.
This function enables the tool path of machining programs to be traced without operating any functions.
For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

11.2.5 Graphic Trace

	E60	E68
M system	○	○
L system	○	○

The machine position of the machine tool is traced. By this operation, the actual movement path in automatic operation or manual operation is traced.
For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

11.3 Program Search / Start / Stop

11.3.1 Program Search

	E60	E68
M system	○	○
L system	○	○

The program No. of the program to be operated automatically can be designated and called.
Upon completion of search, the head of the program searched is displayed.
Machining programs are stored in the memory inside the NC system.

11.3.2 Sequence Number Search

	E60	E68
M system	○	○
L system	○	○

Blocks can be indexed by setting the program No., sequence No. and block No. of the program to be operated automatically.
The searched program is displayed upon completion of the search.

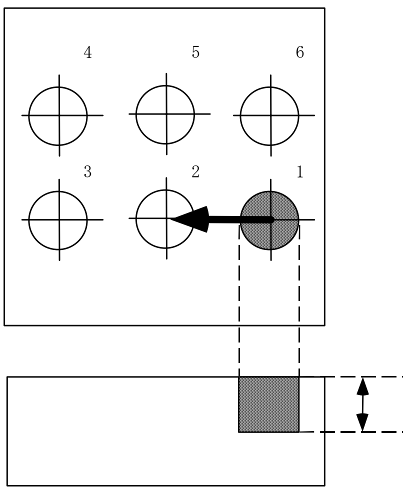
11.3.3 Collation Stop

	E60	E68
M system	–	○
L system	–	○

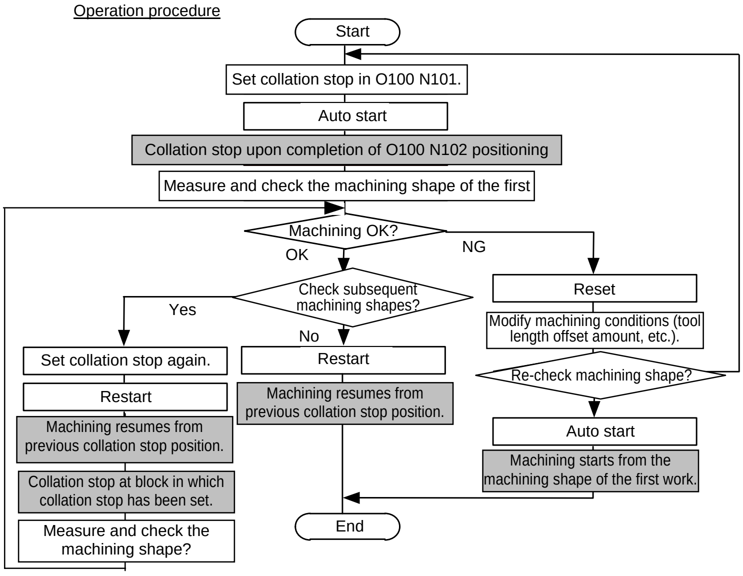
This function enables the single block stop status to be established at any block without having to set the SINGLE BLOCK switch to ON.
It can be used to readily check the machining shape up to the designated block and resume machining.

(Example)

```
O100:
        G91;
        ……
        G00  Z-150.;
N100    G81  X-100.  Z-100.  R-50.  F100;
N101    X-100.
N102    X-100.
N103    Y100.
N104    X100.
N105    X100.
……
……
```



Operation procedure



11.3.4 Program Restart

	E60	E68
M system	○	○
L system	○	○

When a machining program is to be resumed after it has been suspended midway due to tool damage or for some other reason, this function searches the program and the block to be resumed and enables machining to be resumed from the block. When multiple part systems are used, only for 1-part system, the program can be resumed.
There are two resumption methods, type 1 and type 2.

Resumption type 1
Machining is resumed by type 1 if feed hold has been performed due to tool damage, etc. or if resetting has been performed.

(a) Type A (standard specification)
The designated sequence No. and block No. are searched only in the designated program No.. In case of the standard specification, the program No. cannot be omitted.

(b) Type B
The program No. cannot be input since the main program which has been searched itself serves as the target. (A setting error results if the program No. is input.)
The designated sequence No. and block No. are searched in all the programs (including sub-programs) among the searched programs.

Resumption type 2

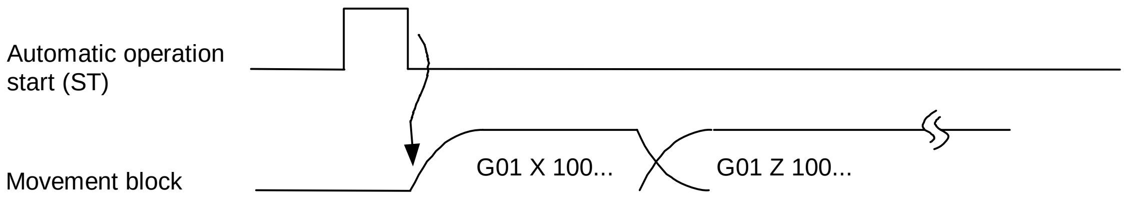
(a) Type A (standard specification)
If, before a resume search is initiated for the machining program to be resumed, a machining program differing from that program was run in the tape or memory mode, the machining program to be resumed will be resumed using type 2. It is also resumed using type 2 in cases where the coordinate system to be used when machining is resumed is to be changed from the coordinate system used during the previous automatic operation.
The operation sequence for type 2 is the same as for type 1. However, the coordinate system settings and other operations that must be performed before running the machining program must all be performed before initiating the resume search. The main program to be resumed should be searched at any time up to the moment immediately prior to starting the resumption of the machining.
The designated sequence No. and block No. are searched only in the designated program No.. In case of the standard specification, the program No. cannot be omitted.

(b) Type B
If, before a resume search is initiated for the machining program to be resumed, a machining program differing from that program was run in the tape or memory mode, the machining program to be resumed will be resumed using type 2. It is also resumed using type 2 in cases where the coordinate system to be used when machining is resumed is to be changed from the coordinate system used during the previous automatic operation.
The operation sequence for type 2 is the same as for type 1. However, the coordinate system settings and other operations that must be performed before running the machining program must all be performed before initiating the resume search.
The program No. cannot be input since the main program which has been searched itself serves as the target. (A setting error results if the program No. is input.)
Therefore, the main program to be resumed must be searched before the resume search is initiated. The designated sequence No. and block No. are searched in all the programs (including sub-programs) among the searched programs.

11.3.5 Automatic Operation Start

	E60	E68
M system	○	○
L system	○	○

With the input of the automatic operation start signal (change from ON to OFF), the automatic operation of the program, which has been operation searched, is started by the controller (or the halted program is restarted).



11.3.6 NC Reset

	E60	E68
M system	○	○
L system	○	○

This function enables the controller to be reset.

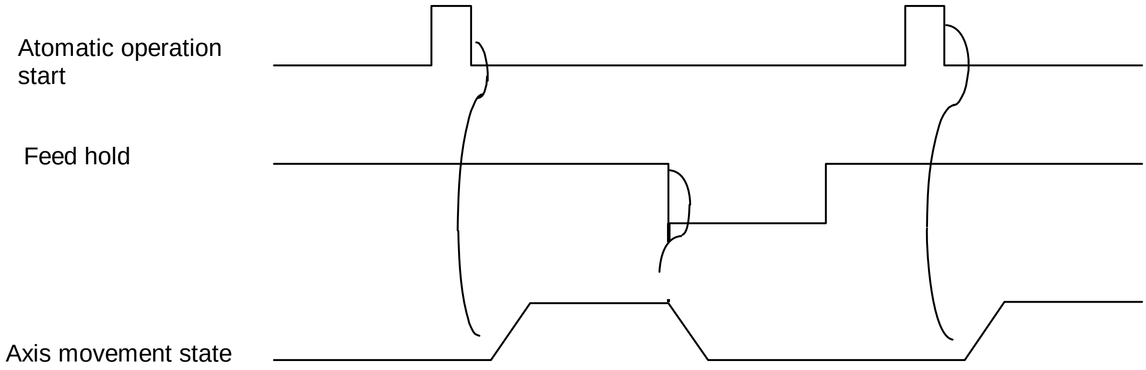
		Reset 1	**Reset 2**	**Reset & Rewind**
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	Retained
6	M code independent output	OFF	OFF	OFF
7	Control axis moving	Decelerated and stopped	Decelerated and stopped	Decelerated and stopped
8	Output signals	"In reset" signal	"In reset" signal	"In reset" signal "In rewind" signal

11.3.7 Feed Hold

	E60	E68
M system	○	○
L system	○	○

When the feed hold signal is set ON during automatic operation, the machine feed is immediately decelerated and stopped. The machine is started again by the "Automatic operation start (cycle start)" signal.

(1) When the feed hold mode is entered during automatic start, the machine feed is stopped immediately, but the M, S, T and B commands in the same block are still executed as programmed.
(2) When the mode is switched during automatic operation to manual operation (jog feed, handle feed or incremental feed), the feed hold stop mode is entered.
(3) An interrupt operation based on manual operation (jog feed, handle feed or incremental feed) can be executed during feed hold.



11.3.8 Search & Start

	E60	E68
M system	○	○
L system	○	○

If the search & start signal is input in a status where the memory mode is selected, the designated machining program is searched and executed from its head.
If the search & start signal has been input during automatic operation in the memory mode, search & start is executed after resetting.

11.4 Interrupt Operation

11.4.1 Manual Interruption

	E60	E68
M system	○	○
L system	○	○

Manual interrupt is a function that enables manual operations to be performed during the automatic operation. The systems used to select the operation mode are as follows:

- System which initiates the interrupt by switching from the automatic mode to manual mode
- System which initiates the interrupt by selecting the manual mode at the same time as the automatic mode

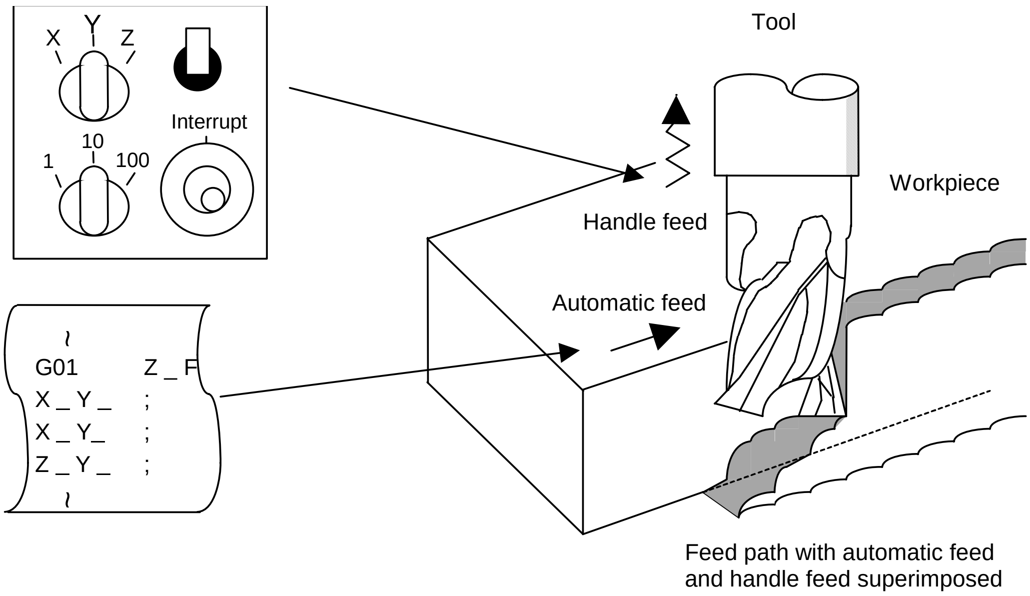
  (Refer to simultaneous operation of manual and automatic modes in section 11.4.9.)

Whether the manual interrupt amount is to be retained and automatic operation is to be continued is determined by setting manual absolute mode ON or OFF (refer to manual absolute mode ON/OFF in section 11.4.3).

11.4.2 Automatic Operation Handle Interruption

	E60	E68
M system	○	○
L system	○	○

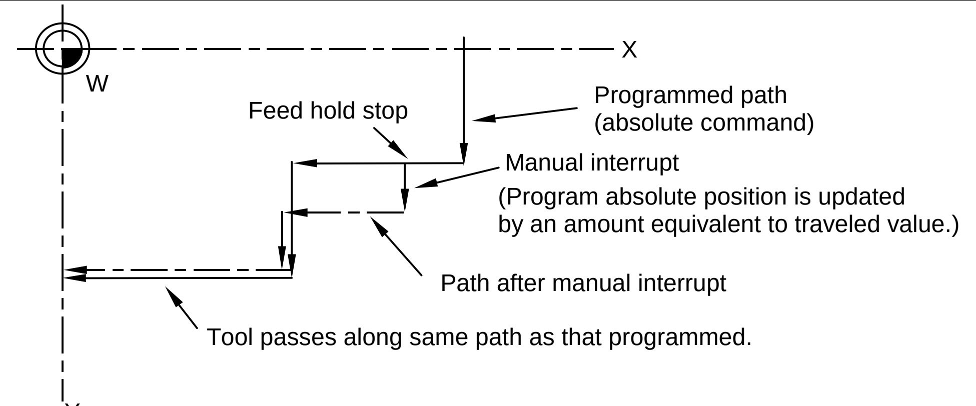
The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during the automatic operation.
If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due to a high rough cutting amount in face machining, for instance, automatic handle interrupt makes it possible to raise the Z surface and reduce the load easily without suspending feed in the automatic operation mode.
Automatic handle interrupt is conducted by setting the "automatic handle interrupt" valid switch provided separately from the "manual operation mode". The axis selection and pulse scale factor operation are conducted as for manual handle feed.
Whether, after an interrupt, to return to the path of the machining program by automatic operation or remain offset by the amount equivalent to the interrupt amount is determined using a parameter.



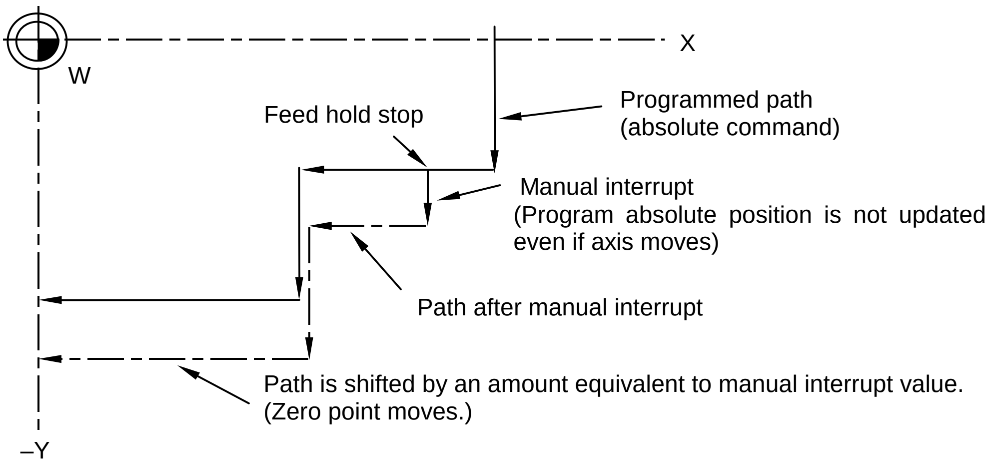
11.4.3 Manual Absolute Mode ON / OFF

	E60	E68
M system	○	○
L system	○	○

The program absolute positions are updated by an amount equivalent to the distance by which the tool is moved by hand when the manual absolute selection input signal is turned ON.
In other words, the coordinate system based on the original program will not shift even if the tool (machine) is moved by hand. Thus, if automatic operation is started in this case, the tool will return to the path before manual movement.



With manual absolute switch ON



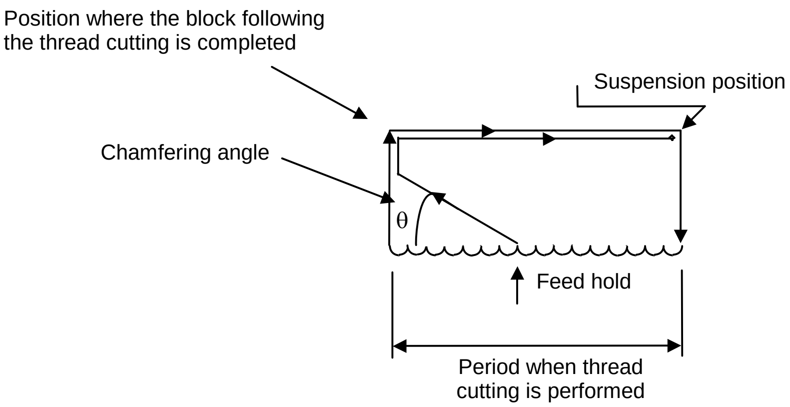
With manual absolute switch OFF

The switch ON state will be entered when the power is turned ON.

11.4.4 Thread Cutting Cycle Retract

	E60	E68
M system	–	–
L system	–	○

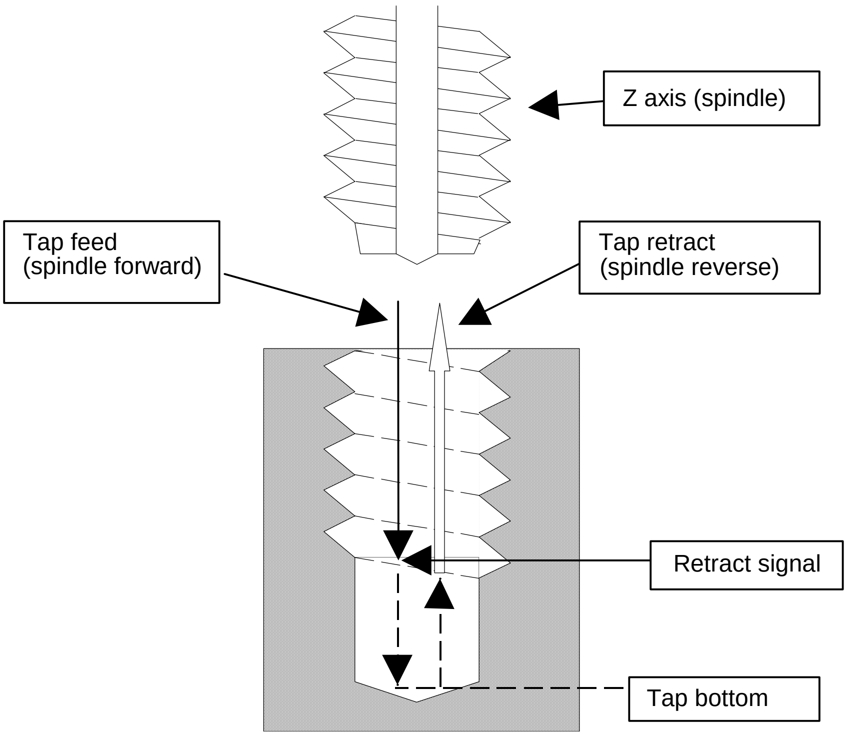
This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting in a thread cutting cycle.
If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is completed.



11.4.5 Tapping Retract

	E60	E68
M system	○	○
L system	○	○

If tapping is interrupted by a reset or emergency stop signal that is input during tapping and the tap is left engaged inside the workpiece, the tap tool engaged inside the workpiece can be rotated in the reverse direction so that it will be disengaged by inputting the tap retract signal.

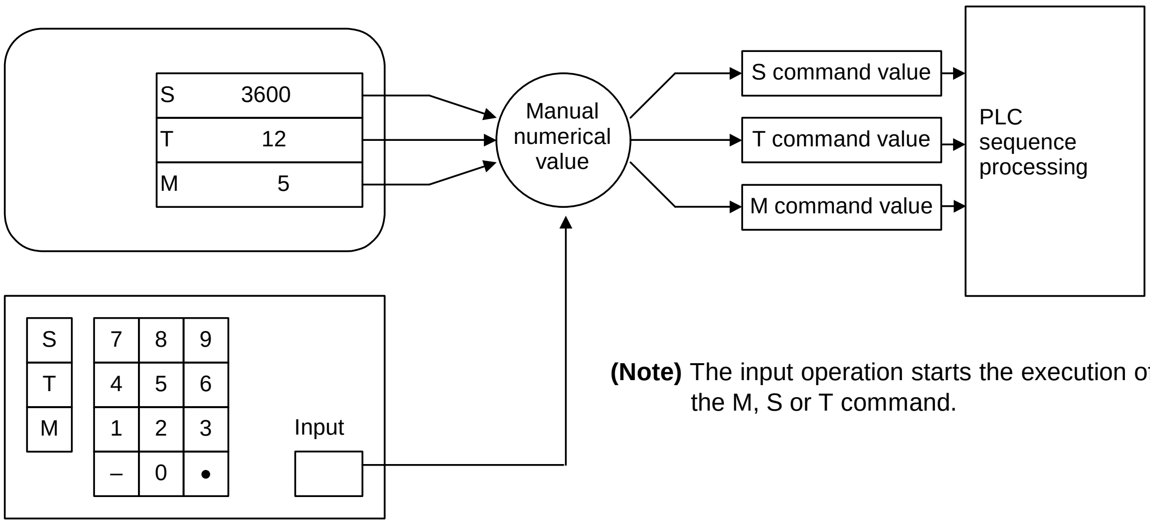


This function can be used by an interruption initiated by reset or emergency stop.
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical value Command

	E60	E68
M system	○	○
L system	○	○

On the screen, the M, S and T (and B when 2nd miscellaneous function is valid) commands can be executed by setting numerical values and pressing [INPUT].
This enables operations such as spindle speed changing, starting, stopping, calling and selecting assigned tools and replacing of the spindle tools to be done easily without having to prepare or revise the machining program. Even in an automatic operation mode, these operations can be conducted with block stop.
Furthermore, the M and T commands can be issued even on the tool offset amount setting and display screen, therefore at the manual tool length measurement, the tools can be called successively to the spindle and measured very simply without having to change the screen page.



(Note) The input operation starts the execution of the M, S or T command.

11.4.8 MDI Interruption

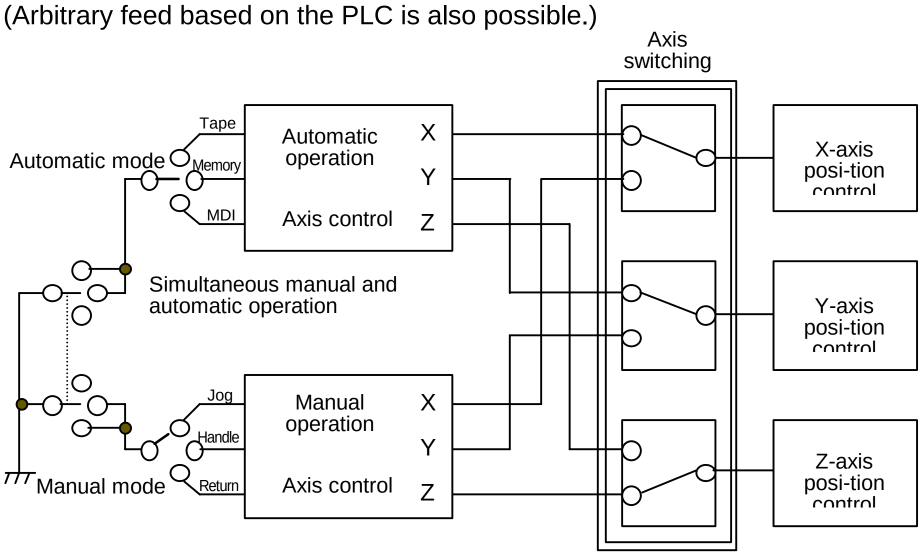
	E60	E68
M system	○	○
L system	○	○

This function enables MDI programs to be executed during automatic operation in the single block stop status. When the modal status is changed in the MDI program, the modal status in the automatic operation mode is also changed.

11.4.9 Simultaneous Operation of Manual and Automatic Modes

	E60	E68
M system	○	○
L system	○	○

This function enables manual operations to be performed during automatic operation by selecting an automatic operation mode (tape, MDI or memory) and manual mode (handle, step, jog or manual reference position return) simultaneously.
(Arbitrary feed based on the PLC is also possible.)



The feed rates for the axes subject to automatic commands and the feed rates for axes subject to manual command are set separately. The acceleration/deceleration modes (rapid traverse, cutting feed) are also set separately. Rapid traverse override, cutting feed override and second cutting feed override are valid both for axes subject to automatic commands and axes subject to manual commands. Override cancel is valid for axes subject to automatic commands. Manual interlock is applied to axes subject to manual commands; automatic interlock is applies to axes subject to automatic commands.

11.4.10 Simultaneous Operation of Jog and Handle Modes

	E60	E68
M system	○	○
L system	○	○

When executing the jog feed and handle feed, both these feeds are available without changing the mode each time by inputting the jog mode signal and simultaneous operation of jog and handle modes signal to the control unit. However, during moving in one of the two modes, the feed in the other mode is not valid.

11.4.11 Reference Position Retract

	E60	E68
M system	○	○
L system	○	○

When the retract signal is turned ON during the automatic and manual operation, this function can retract the tool immediately to a set reference position.
The reference position to be retracted to can be selected from the 1st reference position to 4th reference position with 2-bit input signal.
Set the retracting order of axes with parameter (#2019 revnum).

(1) Other operations

(a) When the retract signal is turned ON, the control unit is reset, the operation is interrupted, and the machining program is indexed.
(b) When the rapid traverse input signal is input, the rapid traverse rate is applied. When the rapid traverse input signal is not input, the manual feed rate is applied.
(c) If the retract signal is input during execution of a tapping cycle, the operation will be the tapping retract, and the normal reference position retract will be executed from the end point of tapping retract operation.
(d) Even if the retract signal is input during the thread cutting cycle, it will be invalid. However, if the retract signal is input in a block other than the thread cutting block, the retracting operation will be executed.
(e) If the retract signal is turned OFF midway during retracting, the operation will decelerate and stop. However, since the machining program is indexed, the block cannot be resumed.
(f) The retract signal is invalid the coordinate system is not established. An operation error will occur when the retract signal is input in such case.

11.4.14 PLC Interruption

	E60	E68
M system	–	○
L system	–	○

The interrupt program set with the R register is executed with the signals from the PLC during single block stop in program operation or during the manual mode.

12. Programming Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

12.1.1.1 Subprogram Control

	E60	E68
M system	○8 layers	○8 layers
L system	○8 layers	○8 layers

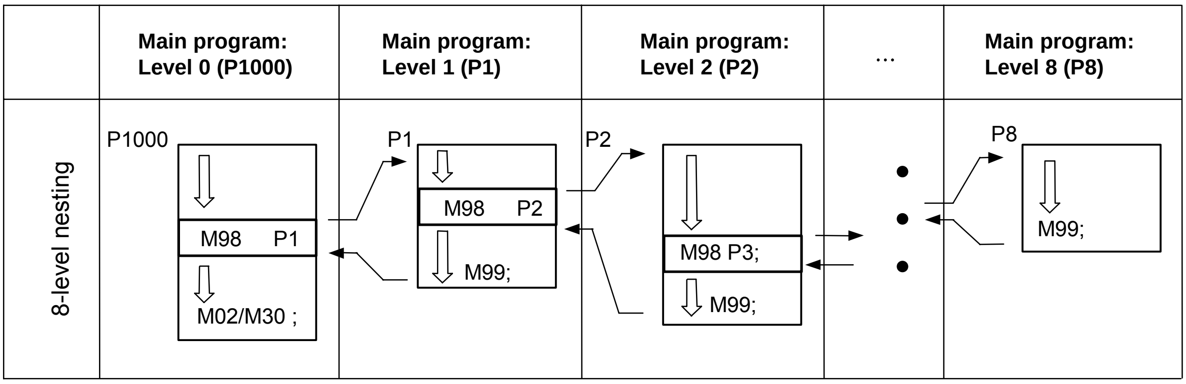
When the same pattern is repeated during machining, the machining pattern is registered as one subprogram and the subprogram is called from the main program as required, thereby realizing the same machining easily. Efficient use of program can be made. The call is designated with the program number and sequence number.

M98 Pp1 Hh1 LL1 ;

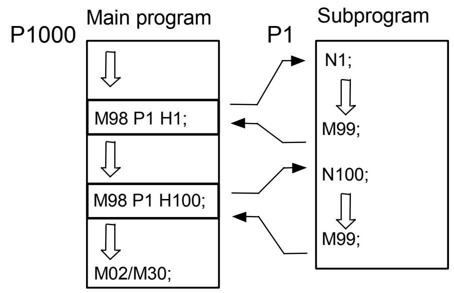
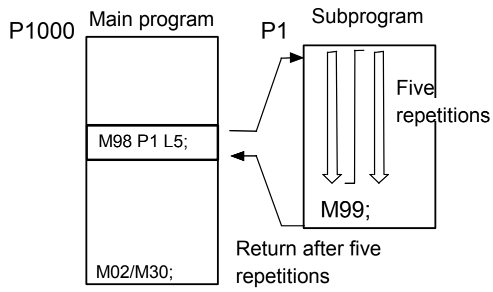
M98 : Call command
Pp1 : Subprogram number
Hh1 : Sequence number
Ll1 : Number of repetitions

(Branch to subprogram)
Op1 (Subprogram)
:
Nh1
:
M99 ; (Return to main program)

Subprograms can be nested up to eight levels deep.



A subprogram branch destination or repetition of a subprogram can be specified.

Specifying a subprogram branch destination	Specifying repetition of a subprogram
	

12.1.1.3 Scaling

	E60	E68
M system	○	○
L system	–	–

The shape commanded by the program can be extended or reduced to the desired size by applying a scale factor to the movement axis command position.

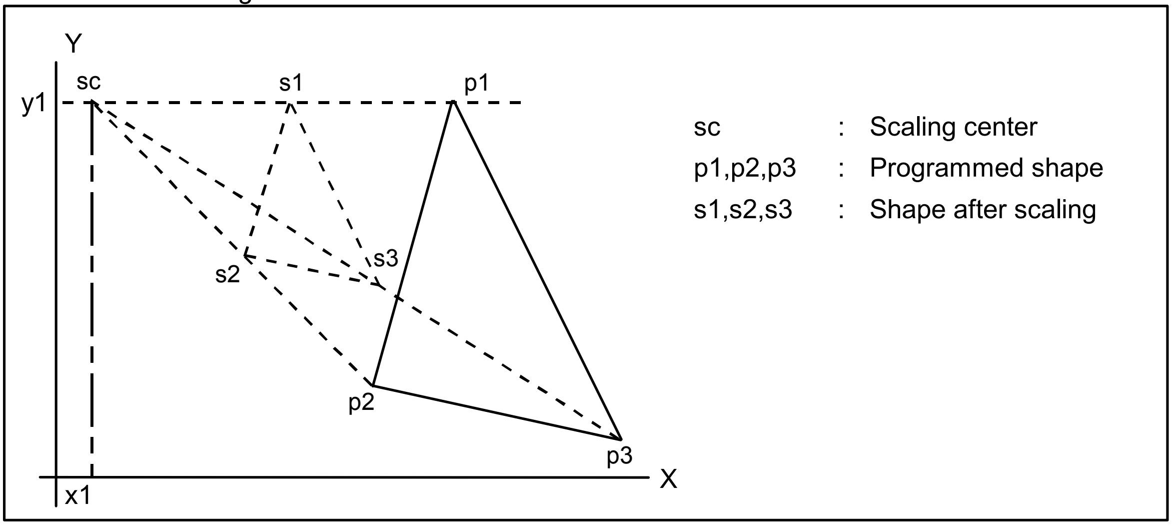
G code	Function
G50 G51	Scaling cancel Scaling ON

The program format is given below.

G51 Xx1 Yy1 Zz1 Pp1 ;

G51	: Call command
Xx1, Yy1, Zz1	: Scaling center coordinate position
Pp1	: Scale factor

The scale factor ranges from 0.000001 to 99.999999 times.



(Note 1) Scaling cannot be applied to compensation amount for tool radius compensation, tool position offset, or tool length compensation, etc. (The compensation and offset are calculated for the scaled shape.)

(Note 2) Scaling applies only to the axes commanded with G51 block; it does not apply to axes that have not been commanded.
When the scale factor is not assigned, the parameter setting applies instead.

12.1.2 Macro Program

12.1.2.1 User Macro

	E60	E68
M system	○4 layers	○4 layers
L system	○4 layers	○4 layers

(1) Macro commands (1) ; G65 to G67

In order to carry through one integrated function, a group of control and arithmetic instructions can be used and registered as a macro program. Furthermore, subprograms with a high degree of expandability can be configured by setting these macro programs as types that are capable of conducting control and arithmetic operations using variable commands.

G code	Function
G65	Macro call (Sample call)
G66	Macro modal call A
G66.1	Macro modal call B
G67	Macro modal call cancel

The program formats are given below.

G65 Pp1 Ll1 Argument ;

G65 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : Variable data assignment

The macro program is called immediately by this command.

G66 Pp1 Ll1 Argument ;

G66 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : Variable data assignment

The macro program is executed from the block with the axis command following this command.

G66.1 Pp1 Ll1 Argument ;

G66.1 : Call command
Pp1 : Program No.
Ll1 : No. of repetitions
Argument : Variable data assignment

The macro program is executed with the word data of each block as the argument.

The following macro command functions are available.

Arithmetic commands	#1 = <Expression> ; Various arithmetic operations can be conducted between variables by the above. "<Expression>" is a combination of constants, variables, functions and operators.
Assignment of priority of arithmetic operations	The portion in which the operator is to be given priority can be enclosed in [ ]. Up to five pairs of square parentheses [ ] including the function [ ] can be used. The normal priority of operation is functions and multiplication/division followed by addition/subtraction.
Control commands	(1) IF [<Conditional expression>] GOTO n ; (2) WHILE [<Conditional expression>] DO m ; . . . END m ; The flow of the program can be controlled by these commands. "n" denotes the sequence numbers of the branching destination. "m" is an identification number, and 1 to 127 can be used. Note that only 27 nestings can be used.

(Note) The variable commands are provided under the optional specifications independently of the user macros. If they are to be used, specify the optional specifications separately.

(2) Macro commands (2)

Specific G commands and the miscellaneous commands (M, S, T, B) can be used for macro call.

(a) Macro call using G codes

Simply by assigning a G code, it is possible to call user macro programs with the prescribed program number.

Format

GXX <Argument> ;	
GXX	: G code for performing macro call

The correspondence between the G×× code which performs macro call and the program number for the macro to be called is set by a parameter.

Up to 10 codes from G00 to G255 can be used for this command. (Whether to use codes such as G00, G01 or G02 which have already been clearly assigned for specific applications by the EIA standards as macro codes can be changed over using a parameter. [M system].)
Up to 800 codes from G200 to G999 can be used in this command. [L system]

(b) Macro call using miscellaneous commands (M, S, T, B code macro call)

Simply by designating an M (or S, T, B) code, it is possible to call user macro programs with the prescribed program number. (Entered M codes and all S, T and B codes can be used.)

Mm ; (or Ss;, Tt;, Bb;)
Mm (Ss, Tt, Bb) : M (or S, T, B) code for performing macro call

The correspondence between the Mm code which performs macro call and the program number for the macro to be called is set by a parameter. Up to 10 M codes from M00 to M95 can be entered.
Neither the codes basically required by the machine nor M codes M0, M1, M2, M30, M96 to M99 are to be entered.

(Note 1) G commands in G code macro programs are not subject to macro calls but normal G commands. M commands in M code macro programs are not subject to macro calls but normal M commands. (The same applies to S, T and B codes.)

12.1.2.2 Machine Tool Builder Macro

12.1.2.2.1 Machine Tool Builder Macro SRAM

	E60	E68
M system	–	○
L system	–	○

This function enables macro programs exclusively designed for use by the machine builders to be registered in addition to the regular macro programs. These macros can be called from user programs using the same method as the one used for regular macros. Machine builder macros can be locked, preventing them from being viewed unless the key word is input.
Machine builder macro programs are stored in a dedicated area which means that the user program registration area is not reduced in the process.

Call format 1

G65 Pp1 Ll ;	
P	: Machine builder macro program number (0100001000 – 0199999998)
l	: Number of repetitions

Note: Machine builder macros cannot be called using the G66, G66.1 or M98 command.

Call format 2

G* ;**	
***	: G code defined in macro definition program

12.1.2.3 Macro Interruption

	E60	E68
M system	○	○
L system	○	○

By inputting a user macro interrupt signal from the PLC, the program being currently executed is interrupted and other programs can be called instead.
Retract or return operations when tools have been damaged, for instance, and other kinds of restoration operations to be conducted when trouble has occurred are programmed in the interrupt programs. There are two types of interrupts, type 1 and type 2, as described below, and they are selected using a parameter.

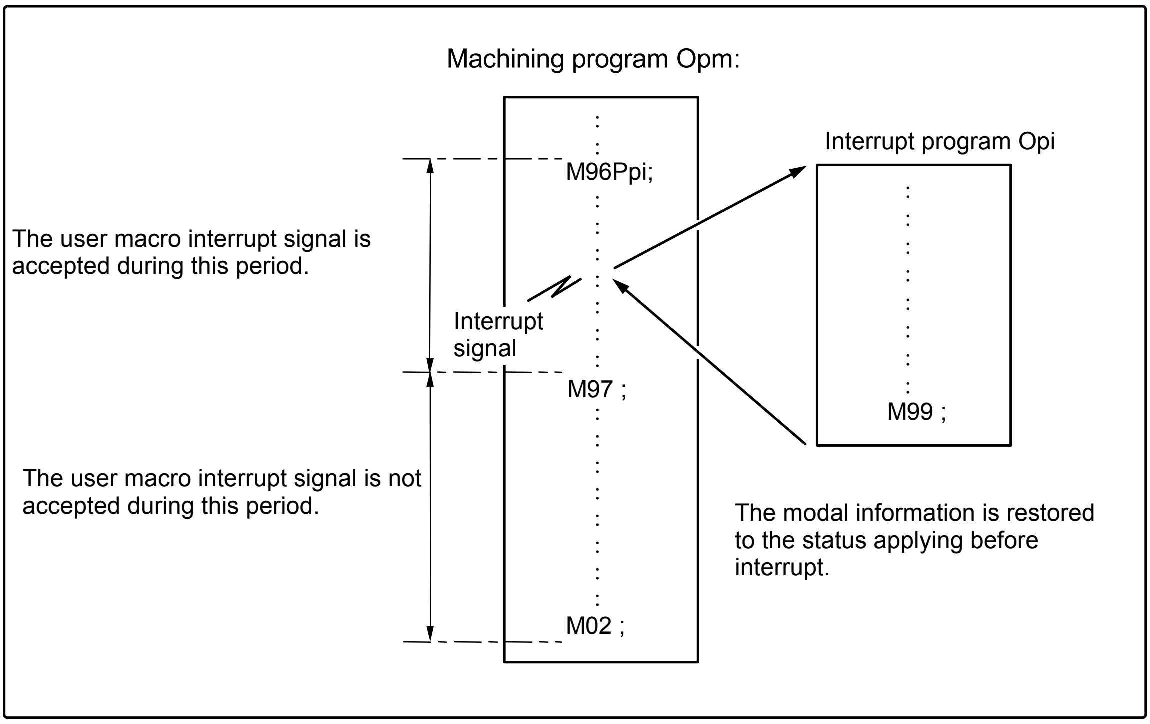
[Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.

[Interrupt type 2] After the block being executed is complete, the interrupt program is executed.

The command format is given below.

M96 P__ H__ ; **User macro interrupt valid**
M97 ; **User macro interrupt invalid**

P : Interrupt program No.
H : Interrupt sequence No.



12.1.2.4 Variable Command

	E60	E68
M system	○200	○300
L system	○200	○300

Programming can be given flexible and general-purpose capabilities by designating variables instead of directly assigning numbers for addresses in programs and by supplying the values of those variables as required when running the programs.
Arithmetic operations (adding, subtracting, multiplying and dividing) can also be conducted for the variables.

Number of variable sets specifications

The numbers of common variable sets depend on the options, and are as follows.

200 sets #100 ~ #199, #500 ~ #599

- Variable names can be set for #500 ~ #519.

Variable expressions

Variable	: # Numerical value (Numerical value: 1, 2, 3, ...)	#100
	: # [Expression]	#100
Expression	:Numerical value	
	: Variable	
	: Expression Operator Expression	#100 + #101
	: – (minus) Expression	–#120
	: [Expression]	[#110]
	: Function [Expression]	SIN [#110]

Variable definition

Variable = expression

(Note 1) Variables cannot be used with addresses "O" and "N".

12.1.3 Fixed Cycle

List of fixed cycles

Type of fixed cycle	M system	L system				Remarks
	G code system 1	G code system 2	G code system 3	G code system 6	G code system 7	
Fixed cycle for drilling	G70 : G89	G80 : G89	G80 : G89	G80 : G89	G80 : G89	Refer to 12.1.3.1. Refer to 12.1.3.7 (Type II). Refer to 4.5.3.
		G79	G83.2	G79	G83.2	
	G98 G99	G98 G99	G98 G99	G98 G99	G98 G99	
Special fixed cycles	G34 G35 G36	—	—	—	—	Refer to 12.1.3.2.
Fixed cycles for turning machining	—	G90 G92 G94	G77 G78 G79	G90 G92 G94	G77 G78 G79	Refer to 12.1.3.3.
Multiple repetitive fixed cycles for turning machining	—	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	G70 : G76 G76.1 G76.2	Refer to 12.1.3.4. Refer to 12.1.3.5 (Type II).

12.1.3.1 Fixed Cycle for Drilling

	E60	E68
M system	○	○
L system	○	○

(1) M series ; G70 to G89, G88, G99

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.

G code	Function
G70	
G71	
G72	
G73	Step cycle
G74	Reverse tapping cycle
G75	
G76	Fine boring
G77	
G78	
G79	
G80	Fixed cycle cancel
G81	Drilling, spot drilling cycle
G82	Drilling, counterboring cycle
G83	Deep hole drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Backboring cycle
G88	Boring cycle
G89	Boring cycle

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation.

G code	Function
G98	Initial point level return
G99	R point level return

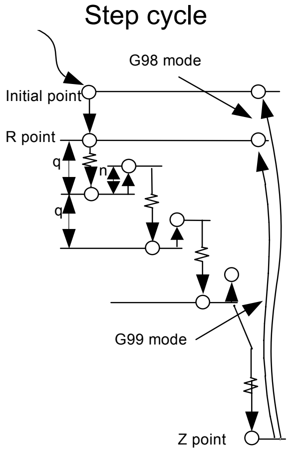
The basic program format for the fixed cycle commands is shown below.

G81 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ll1 Ff1 ;

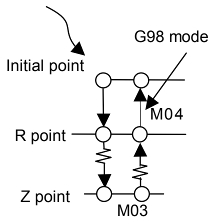
G81	:	Hole drilling mode
Xx1, Yy1	:	Hole position data; X-axis, Y-axis hole drilling position command (rapid traverse) (incremental/absolute)
Zz1	:	Hole machining data; Hole bottom position designation (incremental/absolute)
Rr1	:	Hole machining data; Hole R point designation (incremental/absolute)
Qq1	:	Hole machining data; Depth of cut per pass in G73, G83 cycle (incremental) Shift amount in G76, G87 cycle Depth of cut per pass in pecking tapping, deep hole tapping of G74, G84 cycle
Pp1	:	Hole machining data; Dwell time at hole bottom
Ll1	:	Hole machining data; Number of fixed cycle repetitions
Ff1	:	Cutting feed rate

For details on the synchronous tapping cycle (including pecking tapping cycle and deep-hole tapping cycle), refer to the section "4.5.3 Synchronous tapping".

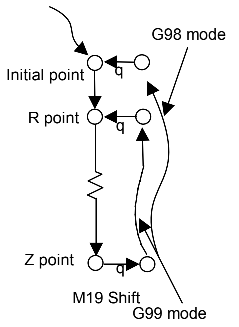
G73
Step cycle



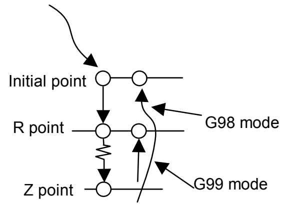
G74
Reverse tapping cycle



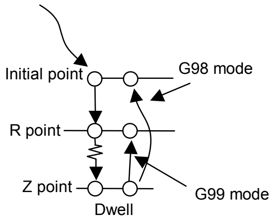
G76
Fine boring cycle



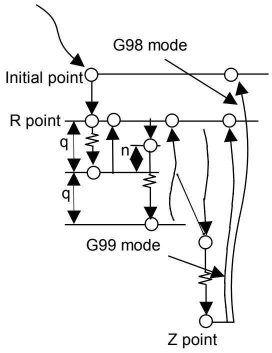
G81
Drilling, spot drilling cycle



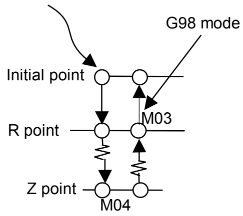
G82
Drilling, counterboring cycle



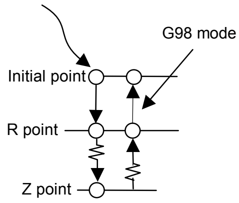
G83
Deep hole drilling cycle



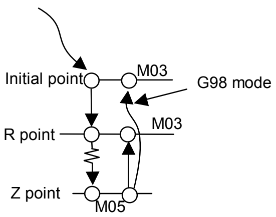
G84
Tapping cycle



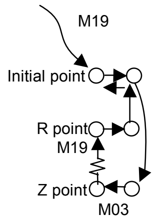
G85
Boring cycle



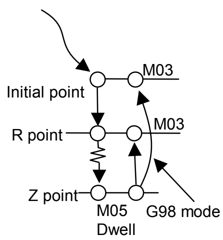
G86
Boring cycle



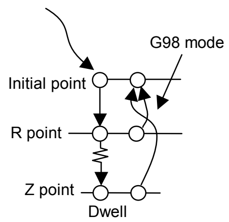
G87
Back boring cycle



G88
Boring cycle



G89
Boring cycle



(2) L series ; G83 to G89, G80

In the fixed cycle for drilling, a machining program such as drilling, tapping, or boring and positioning can be executed for a given machining sequence in 1-block commands.

G code	Drilling axis	Drilling work start	Motion at hole bottom	Return motion	Use
G80	-----	-----	-----	-----	Cancel
G83	Z	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle1
G84	Z	Cutting feed	In-position check Dwell Spindle CCW	Cutting feed	Tapping cycle (Reverse tapping cycle)
G85	Z	Cutting feed	In-position check Dwell	Cutting feed	Boring cycle
G87	X	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle1
G88	X	Cutting feed	In-position check Dwell Spindle CCW	Cutting feed	Tapping cycle (Reverse tapping cycle)
G89	X	Cutting feed	In-position check Dwell	Cutting feed	Boring cycle
G83.2	Z/X	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle2

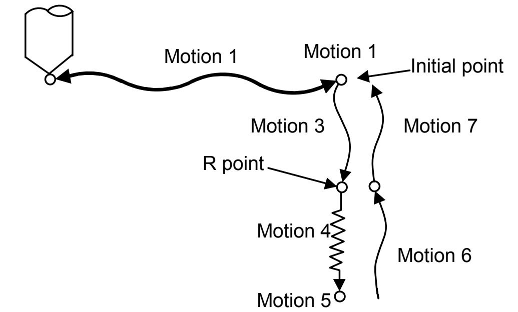
The fixed cycle mode is canceled when a G command of the G80 or G01 group is specified. Data is also cleared simultaneously.

Command format

G83/G84/G85 Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;
G87/G88/G89 Xx1 Cc1 Zz1 Rr1 Qq11 Pp1 Ff1 Kk1 (Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;

G83/G84/G85 G87/G88/G89	:	Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89) The drilling command is modal. Once it is given, it is effective until another drill command is given or drilling fixed cycle cancel command is given.
Xx1, Cc1	:	Data for positioning X (Z) and C axes The data is unmodal. To execute the same hole machining mode consecutively, specify the data for each block.
Zz1, Rr1, Qq11, Pp1, Ff	:	Actual machining data in machining Only Q is unmodal. Specify Q in G83 or G87 for each block whenever the data is required.
Kk1	:	To repeat in a single cycle for hole machining at equal intervals, specify the number of repetitions in the range of 0 to 9999 (no decimal point can be used). It is unmodal and is effective only in the block in which the number of repetitions is specified. If the number of repetitions is omitted, K1 is assumed to be specified. If K0 is specified, hole machining data is stored, but hole machining is not performed. Hole machining data; R point position (incremental value from initial point) designation (sign ignored)
Mm1	:	If axis C clamp M command (parameter setting) is given, the M code is output at the initial point, and after return motion, C axis unclamp M code (clamp M code + 1) is output and the dwell time set in a given parameter is executed.
Ss1	:	Designates spindle rotation speed (When the spindle that is not analog is mounted)
,Ss1	:	Designates spindle rotation speed of return speed (When the spindle that is not analog is mounted)
Dd1	:	Designates tap spindle NO. for G84 (G88) speed (When the spindle that is not analog is mounted)
,Rr1	:	Changes between synchronous/asynchronous in G84 (G88) speed (When the spindle that is not analog is mounted)

The drilling cycle motions generally are classified into the following seven.



Motion 1 : Rapid positioning up to the initial point of X (Z) and C axes.
If the "positioning axis in-position width" is designated, the in-position check is conducted upon completion of the block.

Motion 2 : Output if the C axis clamp M code is given.

Motion 3 : Rapid positioning up to the R point.

Motion 4 : Hole machining at cutting feed.
If the "drilling axis in-position width" is designated, the in-position check is conducted upon completion of the block. However, in the case of deep-hole drilling cycles 1 and 2, the in-position check is not conducted with the drilling of any holes except the last one. The in-position check is conducted at the commanded hole bottom position (last hole drilling).

Motion 5 : Motion at the hole bottom position. It varies depending on the fixed cycle mode.
Spindle CCW (M04), spindle CW (M03), dwell, etc., are included.

Motion 6: Return to the R point.

Motion 7: Return to the initial point at rapid traverse feed.

(Operations 6 and 5 may be conducted as a single operation depending on the fixed cycle mode.

Note: With a synchronous tap command, the in-position check is conducted in accordance with the parameters.

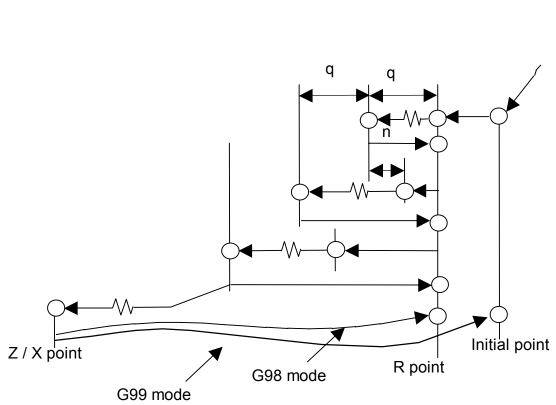
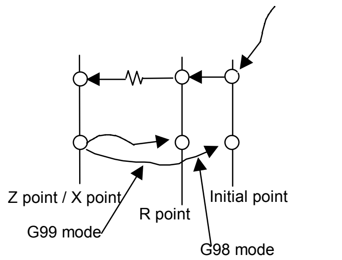
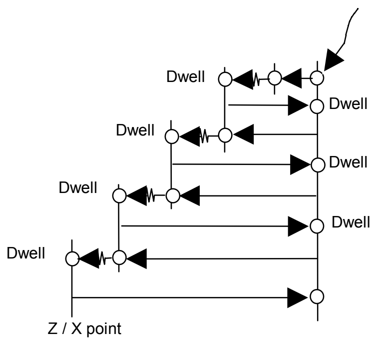
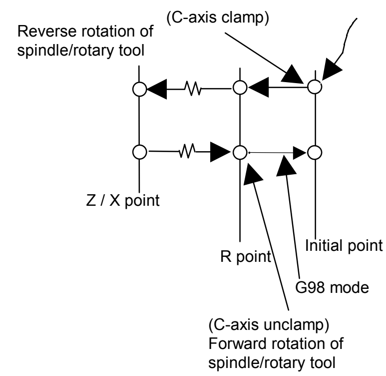
Whether the fixed cycle is complete with motion 6 or 7 can be specified by using either of the following G commands:

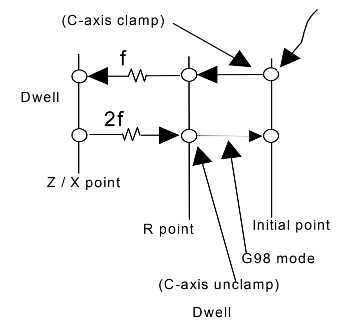
G98: Initial level return

G99: R point level return

These commands are modal. For example, once G98 is given, the G98 mode is entered until G99 is given. The G98 mode is entered in the initial state when the controller is ready.

Deep-hole drilling cycle (G83, G87)

<table>
<tr><td colspan="3">G83/G87
Deep-hole drilling cycle (G83: Z-axis direction, G87: X-axis direction)</td></tr>
<tr><td colspan="2">When Q command is given</td><td>When Q command is not given</td></tr>
<tr><td colspan="2">
</td><td>
</td></tr>
<tr><td>G83.2
Deep-hole drilling cycle

</td><td>G84/88
Tapping cycle


Output or no output can be set using a parameter for the C-axis clamp/unclamp M code</td><td>G85/89
Boring cycle


Output or no output can be set using a paramete for the C-axis clamp/unclamp M code</td></tr>
</table>

There are two levels of hole machining axis return which apply upon completion of the fixed cycle machining operation. (see the figure above)

G code	Function
G98	Initial point level return
G99	R point level return

12.1.3.2 Special Fixed Cycle

	E60	E68
M system	○	○
L system	–	–

Special fixed cycles must always be used in combination with fixed cycles.

(1) Bolt hole circle (G34)

The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius R whose center is the coordinates designated by X and Y, and it drills "n" number of holes at "n" equal intervals along the circumference of that circle. The drilling data for the standard fixed cycle of the G81 or other such command is retained for the drilling operation at each hole position.

All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G34 command.

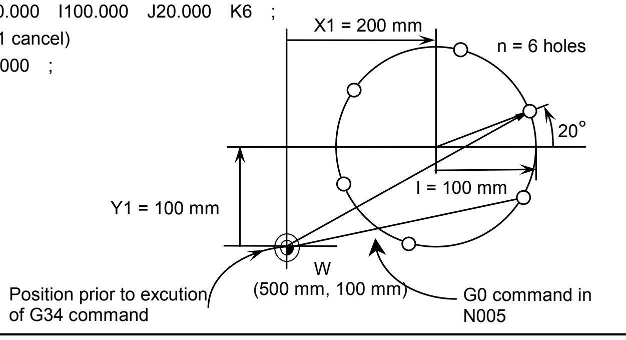
G34 Xx Yy Ir Jθ Kn ;

Xx, Yy	:	Center position of bolt hole circle; this is affected by the G90/G91 commands.
Ir	:	Radius "r" of circle; it is based on the least input increment and is provided using a positive number.
Jθ	:	Angle θ at point to be drilled initially; the counterclockwise direction is taken to be positive.
Kn	:	Number "n" of holes to be drilled; any number of holes from 1 through 9999 can be designated; 0 cannot be assigned. When 0 has been designated, the alarm will occur. A positive number provides positioning in the counterclockwise direction; a negative number provides positioning in the clockwise direction.

(Example)

With 0.001mm least input increment

```
N001  G91  ;
N002  G81  Z – 10.000   R5.000   L0   F200   ;
N003  G90  G34  X200.000   Y100.000   I100.000   J20.000   K6   ;
N004  G80  ;   .........................(G81 cancel)
N005  G90  G0   X500.000   Y100.000   ;
```



As shown in the figure, the tool is positioned above the final hole upon completion of the G34 command. This means that when it is to be moved to the next position, it will be necessary to calculate the coordinates in order to issue the command or commands with incremental values, and so it is convenient to use the absolute value mode.

(2) Line at angle (G35)

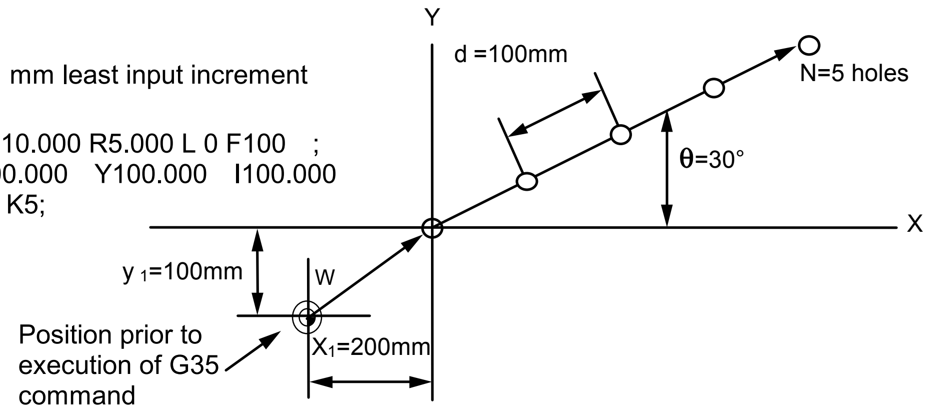
With the starting point at the position designated by X and Y, the tool drills "n" number of holes each at interval "d" in the direction forming angle θ with the X axis. A standard fixed cycle applies for the drilling operation at each of the hole positions and so there is a need to retain beforehand the drilling data (drilling mode and drilling data). All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G35 command.

G35 Xx Yy Id Jθ Kn ;		
Xx, Yy	:	The starting point coordinates; they are affected by the G90/G91 commands.
Id	:	Interval "d"; it is based on the least input increment and when "d" is negative, drilling proceeds in the point symmetrical direction centered on the starting point.
Jθ	:	Angle θ; the counterclockwise direction is taken to be positive.
Kn	:	Number "n" of holes to be drilled including the starting point; any number of holes from 1 through 9999 can be assigned.

(Example)

With 0.001 mm least input increment

```
G91 ;
G81   Z – 10.000 R5.000 L 0 F100   ;
G35   X200.000   Y100.000   I100.000
J 30.000   K5;
```



(3) Arc (G36)

The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius "r" whose center is the coordinates designated by X and Y, and it drills "n" number of holes aligned at angle interval Δθ. As with the bolt hole circle function, the drilling operation at each of the hole positions is based on a hold drilling fixed cycle and so there is a need to retain the drilling data beforehand.
All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G36 command.

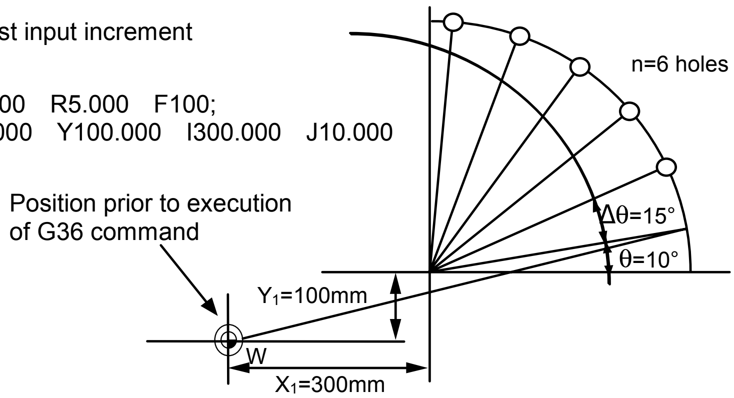
G36 Xx Yy Ir Jθ PΔθ Kn ;

Xx, Yy	:	Center coordinates of arc; they are affected by the G90/G91 commands.
Ir	:	Radius "r" of arc; it is based on the least input increment and is provided with a positive number.
Jθ	:	Angle θ at the point to be drilled initially; the counterclockwise direction is taken to be positive.
PΔθ	:	Angle interval Δθ; when it is positive, the tool drills in the counterclockwise direction and when it is negative, it drills in the clockwise direction.
Kn	:	Number "n" of holes to be drilled; any number of holes from 1 through 9999 can be assigned.

(Example)

With 0.001 mm least input increment

```
N001 G91;
N002 G81   Z-10.000   R5.000   F100;
N003 G36   X300.000   Y100.000   I300.000   J10.000
P   15.000   K6;
```



(4) Grid (G37.1)

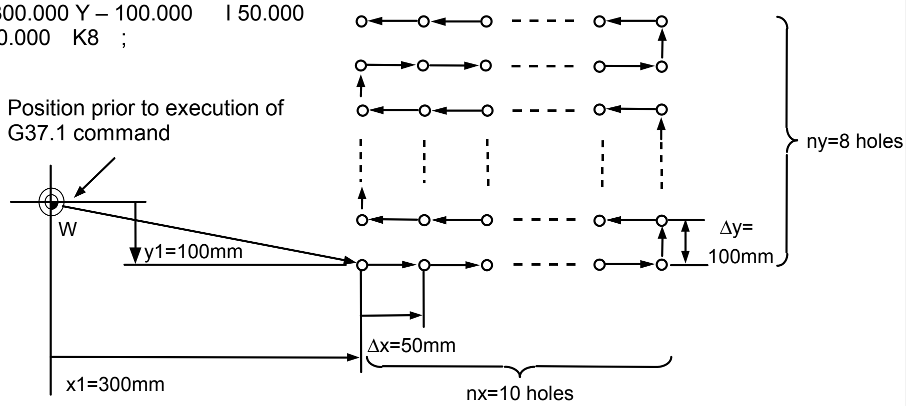
With the starting point at on the position designated by X and Y, this function enables the tool to drill the holes on the lattice with "nx" number of holes at parallel intervals of Δx to the X axis. Drilling proceeds in the X-axis direction. The drilling operation at each of the hole positions is based on a standard fixed cycle and so there is a need to command the drilling data (drilling mode and drilling data) beforehand. All movements between the hole positions are conducted in the G00 mode. The data is not retained upon completion of the G37.1 command.

G37.1 Xx1 Yy1 IΔx Pnx JΔy Kny ;		
Xx, Yy	:	The starting point coordinates; they are affected by the G90/G91 commands.
IΔx	:	X-axis interval Δx; it is based on the least input increment; when Δx is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Pnx	:	Number of holes "nx" in the X-axis direction; any number of holes from 1 through 9999 can be assigned.
JΔy	:	Y-axis interval Δy; it is based on the least input increment; when Δy is positive, the intervals are provided in the positive direction as seen from the starting point and when it is negative, they are provided in the negative direction.
Kny	:	Number of holes "ny" in the Y-axis direction; any number of holes from 1 through 9999 can be assigned.

(Example)

With 0.001 mm least input increment

```
G91 ;
G81 ; Z – 10.000 R5.000   F20   ;
G37.1   X300.000 Y – 100.000     I 50.000
P10   J 100.000   K8   ;
```



12.1.3.3 Fixed Cycle for Turning Machining

	E60	E68
M system	–	–
L system	○	○

The shape normally programmed in several blocks for rough cutting, etc., in the turning machining can be commanded in one block. This function is useful for machining program simplification. The fixed cycles are as follows:

G code	Function
G77	Longitudinal cutting cycle
G78	Thread cutting cycle
G79	Face cutting cycle

Format:

GΔΔ X/U_Z/W_I_K_R_F_(G18 plane)

Each fixed cycle command for turning machining is a modal G code and is effective until another command of the same modal group or a cancel command is given.
The fixed cycle can be canceled by using any of the following G codes:

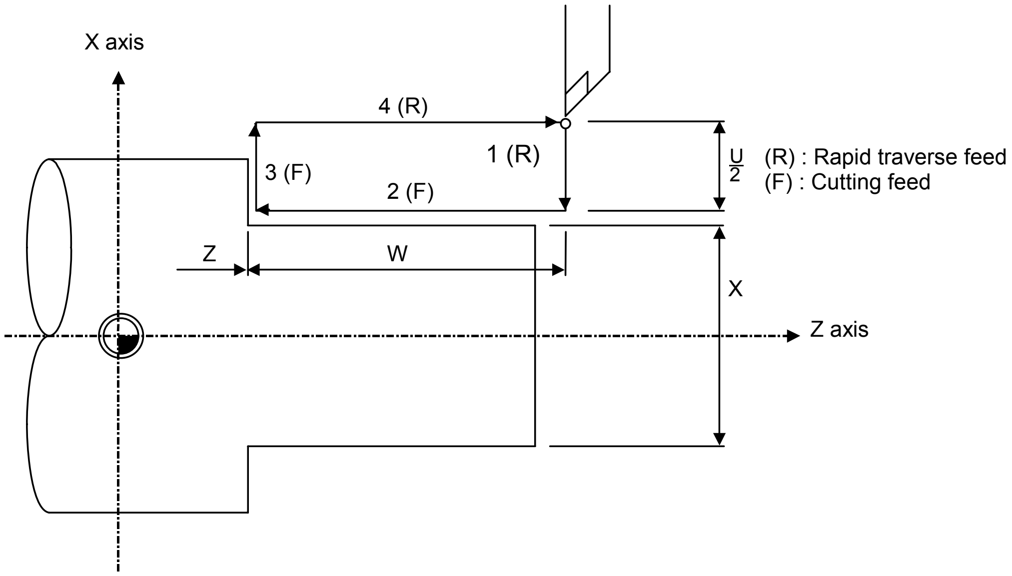
G00, G01, G02, G03
G09
G10, G11
G27, G28, G29, G30
G31
G33, G34
G37
G92
G52, G53
G65

(1) Longitudinal cutting cycle (G77)

(a) Longitudinal cutting

Straight cutting in the longitudinal direction can be performed consecutively by the following block:

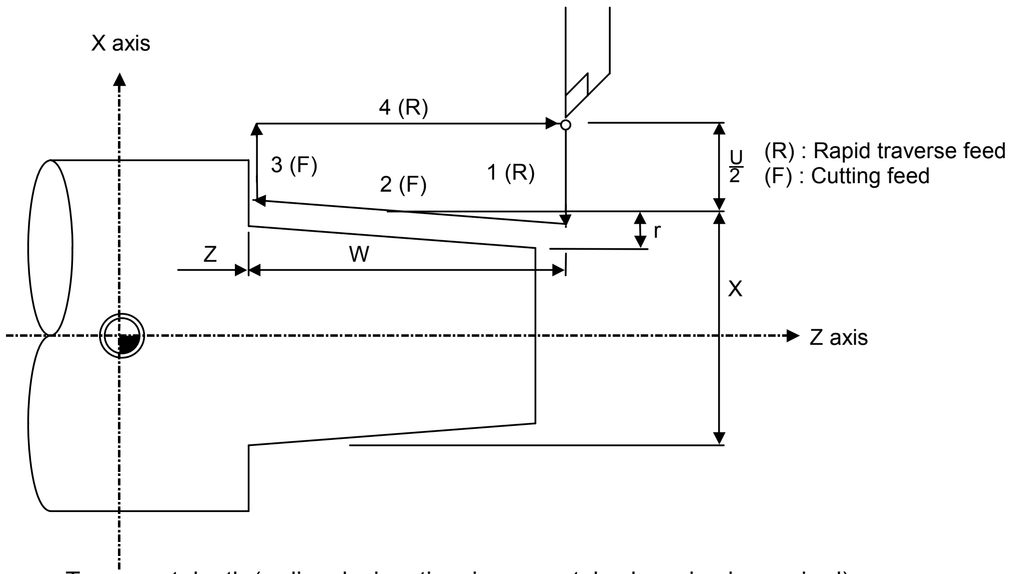
G77X/U_Z/W_F_ ;



(b) Taper cutting

Taper cutting in the longitudinal direction can be performed consecutively by the following block:

G77X/U_Z/W_R_F_ ;



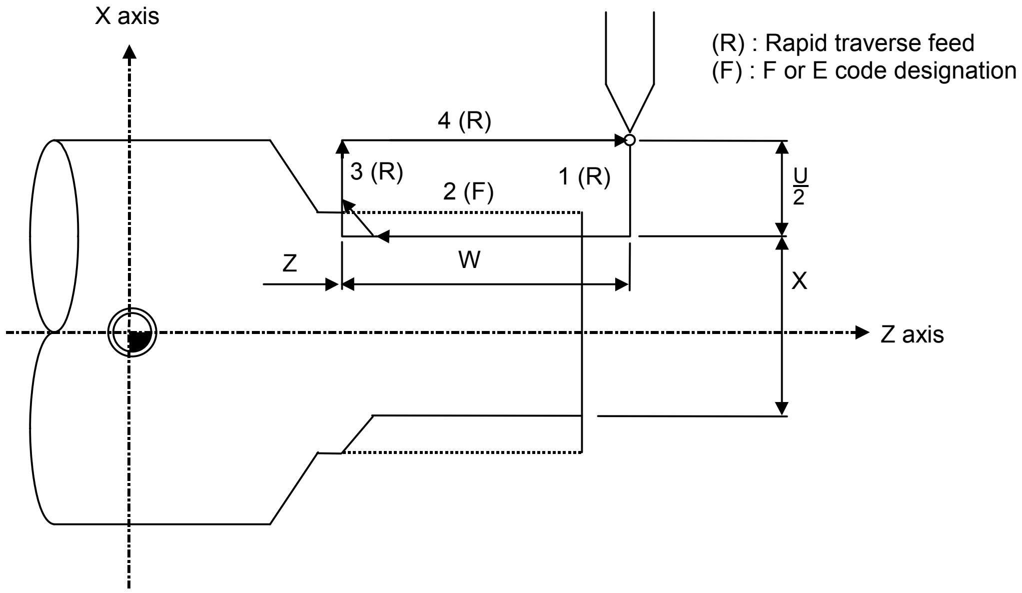
r: Taper part depth (radius designation, incremental value, sign is required)

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

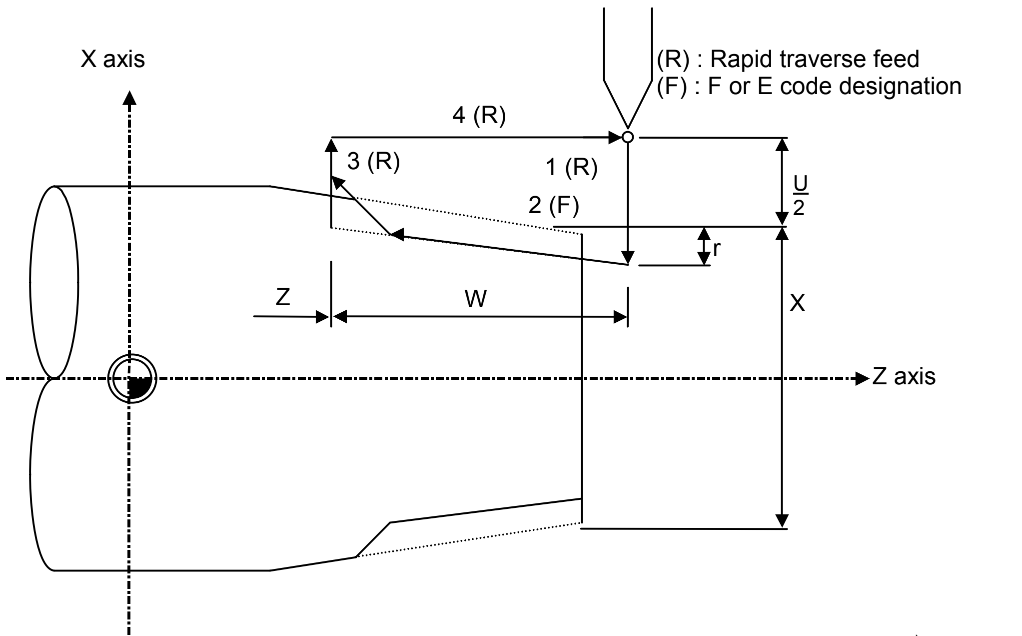
G78X/U_Z/W_F/E_ ;



(b) Taper thread cutting

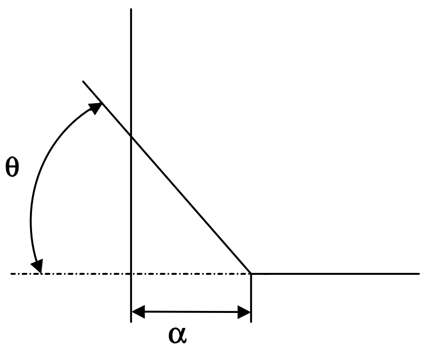
Taper thread cutting can be performed by the following block:

G78X/U_Z/W_R_F/E_ ;



r: Taper part depth (radius designation, incremental value, sign is required)

Chamfering



α : Thread cutting-up amount
Assuming that thread lead is L, the thread cutting-up amount can be set in a given parameter in 0.1L steps in the range of 0 to 12.7L.

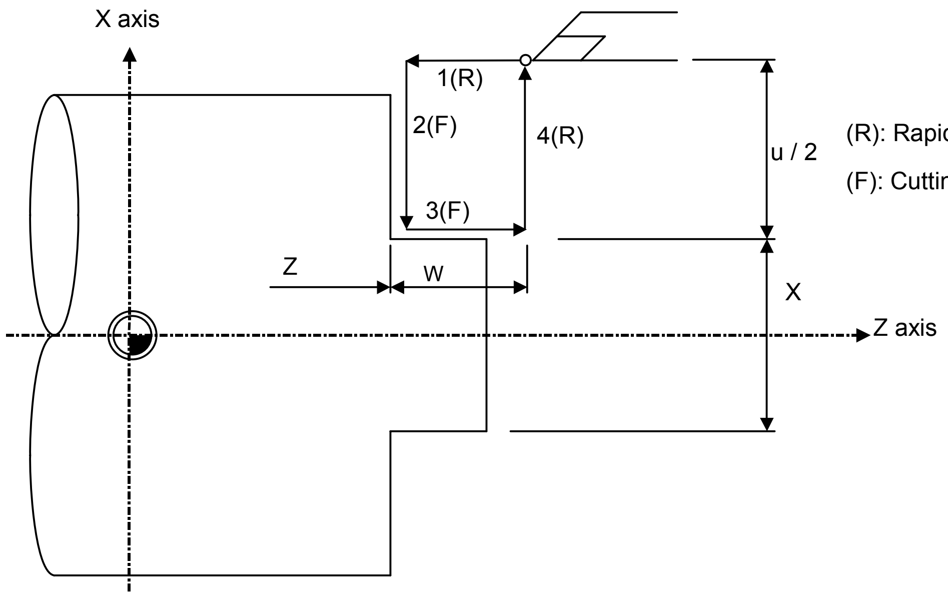
θ : Thread cutting-up angle
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°.

(3) Face cutting cycle (G79)

(a) Straight cutting

Straight cutting in the end face direction can be performed consecutively by the following block:

G79X/U_Z/W_F_ ;

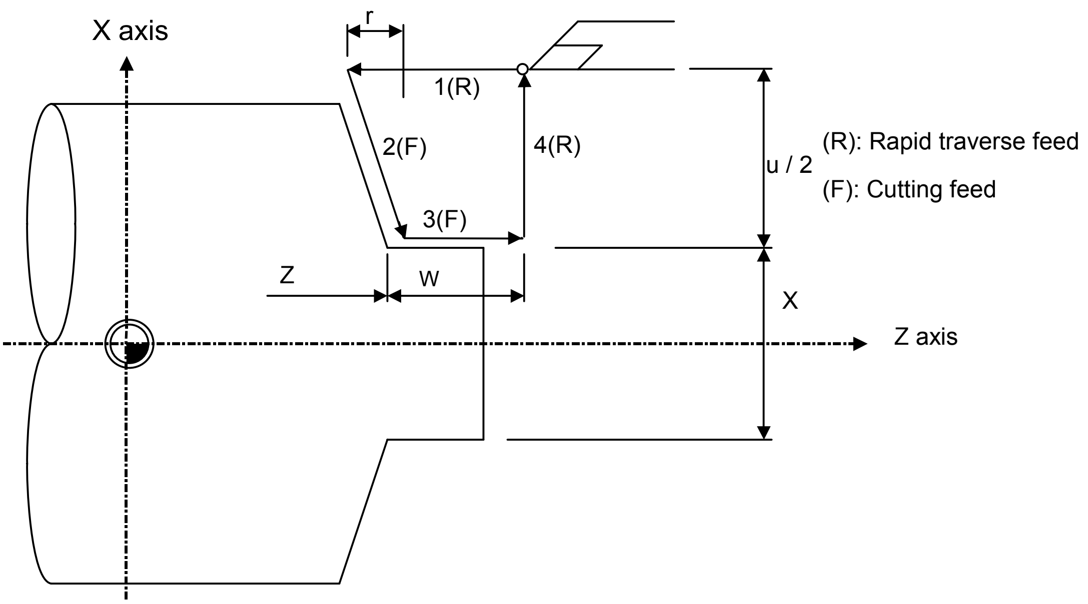


r: Taper part depth (radius designation, incremental value, sign is required)

(b) Taper cutting

Taper cutting in the end face direction can be performed consecutively by the following block:

G79X/U_Z/W_R_F_ ;



r: Taper part depth (radius designation, incremental value, sign is required)

12.1.3.4 Multiple Repetitive Fixed Cycle for Turning Machining

	E60	E68
M system	–	–
L system	○	○

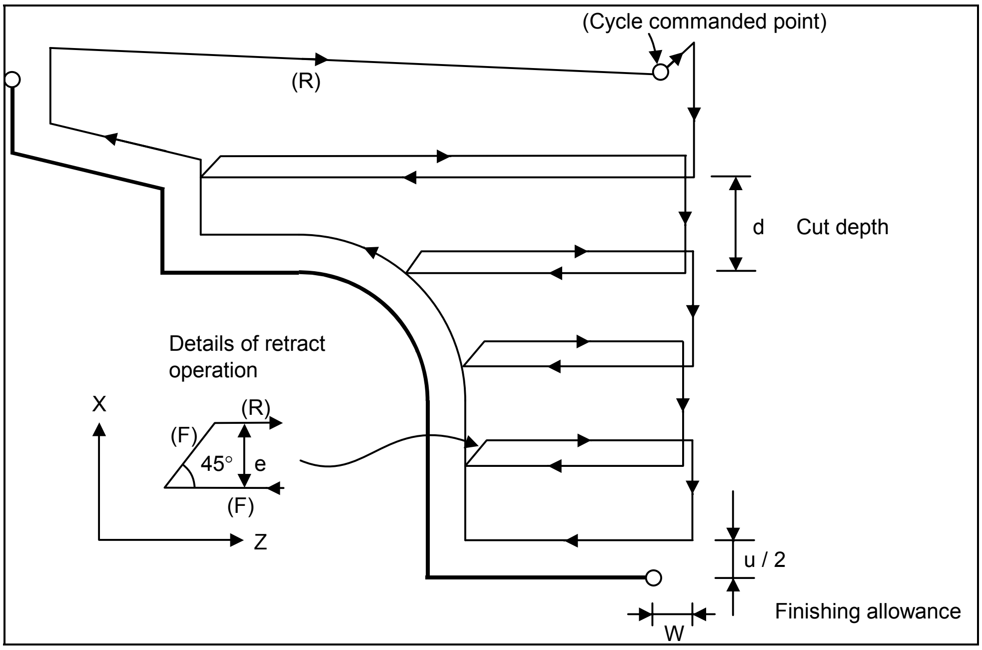
(a) Longitudinal rough cutting cycle I (G71)

The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.
The machining program is commanded as follows.

G71 Ud Re ;

G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ud : Cut depth d. (When P,Q command is not given). (Modal)
Re : Retract amount e. (Modal)
Aa : Finish shape program No. (If it is omitted, the program being executed is assumed to be designated.)
Pp : Finish shape start sequence No. (If it is omitted, the program top is assumed to be designated.)
Qq : Finish shape end sequence No. (If it is omitted, the program end is assumed to be designated.)
However, if M99 precedes the Q command, up to M99.
Uu : Finishing allowance in the X axis direction. (When P, Q command is given). (Diameter or radius designation)
Ww : Finishing allowance in the Z axis direction.
Ff : Cutting feed rate.
Ss : Spindle speed.
Tt : Tool command.
$\}$ F, S, and T command in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.



(b) Face rough cutting cycle (G72)

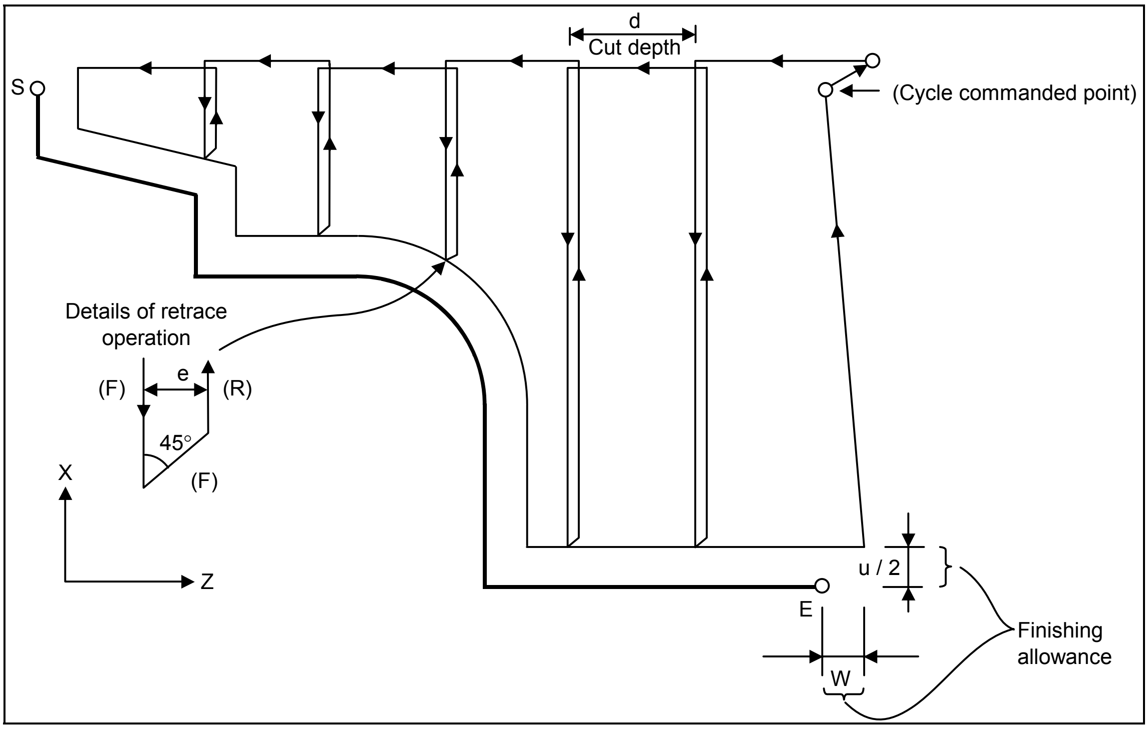
The finish shape program is called, and rough turning is performed in the end face direction while intermediate path is being calculated automatically.
The machining program is commanded as follows.

G72 Wd Re ;

G72 Aa Pp Qq Uu Ww Ff Ss Tt ;

Wd	:	Cut depth d. (When P,Q command is not given). (Modal)
Re	:	Retract amount e. (Modal)
Aa	:	Finish shape program No. (If it is omitted, the program being executed is assumed to be designated.)
Pp	:	Finish shape start sequence No. (If it is omitted, the program top is assumed to be designated.)
Qq	:	Finish shape end sequence No. (If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Q command, up to M99.
Uu	:	Finishing allowance in the X axis direction.
Ww	:	Finishing allowance in the Z axis direction. (When P, Q command is given.)
Ff	:	Cutting feed rate.
Ss	:	Spindle speed.
Tt	:	Tool command.

F, S, and T command in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.



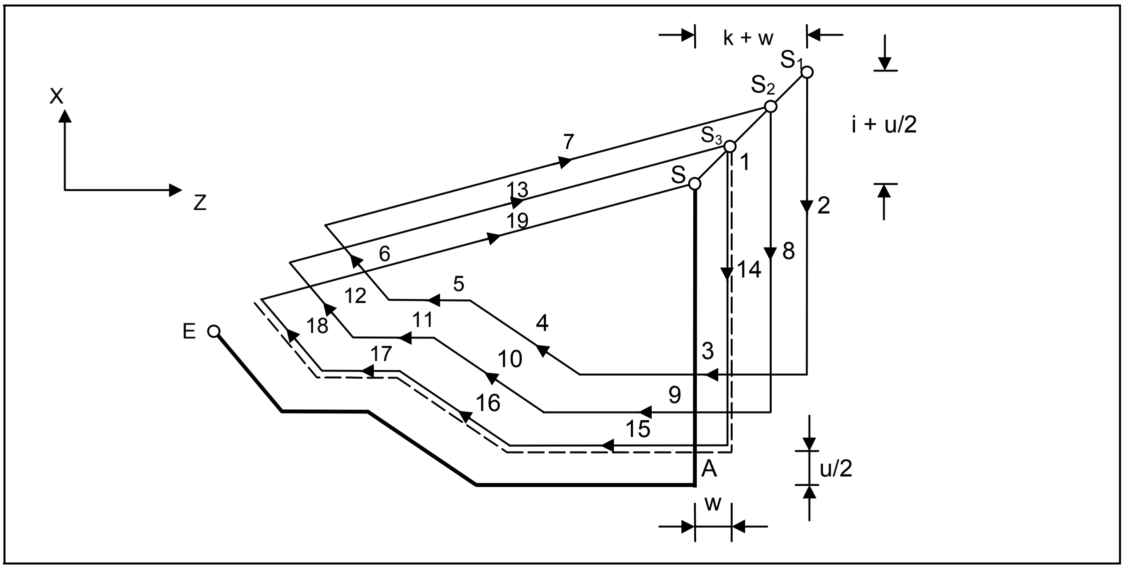
(c) Molding material in rough cutting cycle (G73)

The finish shape program is called. Intermediate path is automatically calculated and rough cutting is performed conforming to the finish shape.
The machining program is commanded as follows.

G73 Ui Wk Rd ;

G73 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ui :	Cutting allowance in the X axis direction	i	• Cutting allowance when P, Q command is not given. • Modal data • Sign is ignored. • Cutting allowance is given with a radius designation.
Wk :	Cutting allowance in the Z axis direction	k	
Rd :	Split count	d	
Aa :	Finish shape program No.		(If it is omitted, the present program is assumed to be designated.)
Pp :	Finish shape start sequence No.		(If it is omitted, the program top is assumed to be designated.)
Qq :	Finish shape end sequence No.		(If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Qq command, up to M99.
Uu :	Finishing allowance in the X axis direction	u	• Finishing allowance when P, Q command is given. • Sign is ignored. • Diameter or radius is designated according to the parameter. • The shift direction is determined by the shape.
Ww :	Finishing allowance in the Z axis direction	w	
Ff :	Cutting feed rate (F function)		The F, S, and T commands in the finish shape program are ignored, and the value in the rough cutting command or the preceding value becomes effective.
Ss :	Spindle speed (S function)		
Tt :	Tool selection (T function)		



(d) Finish cycle (G70)

After rough cutting is performed by using G71 to G73, finish turning can be performed by using the G70 command.
The machining program is commanded as follows.

G70 A_ P_ Q_ ;

A	: Finish shape program number. (If it is omitted, the program being executed is assumed to be designated.)
P	: Finish shape start sequence number. (If it is omitted, the program top is assumed to be designated.)
Q	: Finish shape end sequence number. (If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Q command, up to M99.

(1) The F, S, and T commands in the rough cutting cycle command G71 to G73 blocks are ignored, and the F, S, and T commands in the finish shape program become effective.
(2) The memory address of the finish shape program executed by G71 to G72 is not stored. Whenever G70 is executed, a program search is made.
(3) When the G70 cycle terminates, the tool returns to the start point at the rapid traverse feed rate and the next block is read.

(Example 1) Sequence No. designation

```
   :
N100   G70   P200   Q300  ;        N200 ··· ;
N110                                  :
                                   N300 ·· ;

N120
   :
N200 ⎫
   : ⎬ Finish shape program
N300 ⎭
N310
   :
```

(Example 2) Program No. designation

```
   :
N100   G70
                                  O100
A100   ;
N110 ····· ;                      G01 X100   Z50   F0.5 ;
   :                                   :
                                  M99 ;
```

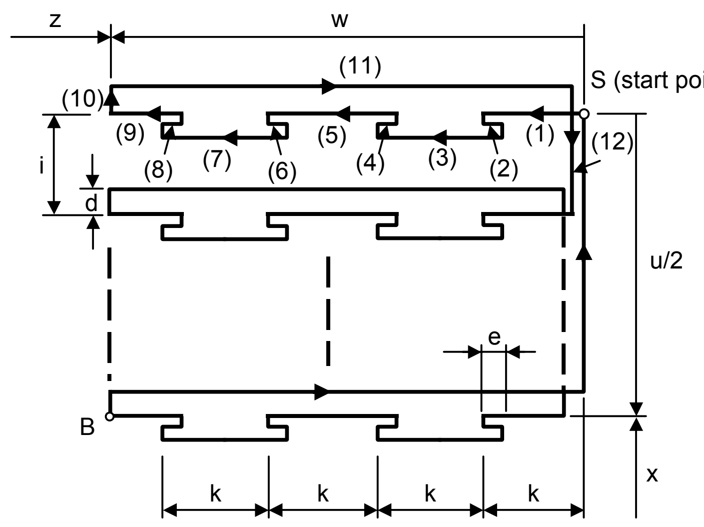
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

(e) Face cutting-off cycle (G74)

When the slotting end point coordinates, cut depth, cutting tool shift amount, and cutting tool relief amount at the cut bottom are commanded, automatic slotting is performed in the end face direction of a given bar by G74 fixed cycle. The machining program is commanded as follows.

G74 Re ;
G74 X/(U) Z/(W) Pi Qk Rd Ff ;

Re	:	Retract amount e (when X/U, Z/W command is not given) (Modal)
X/U	:	B point coordinate (absolute/incremental)
Z/W	:	B point coordinate (absolute/incremental)
Pi	:	Tool shift amount (radius designation, incremental, sign not required)
Qk	:	Cut depth k (radius designation, incremental, sign not required)
Rd	:	Relief amount at cut bottom d (If sign is not provided, relief is made at the first cut bottom. If minus sign is provided, relief is made not at the first cut bottom but at the second cut bottom and later.)
Ff	:	Feed rate



- (9) and (12) just before the last cycle are executed with the remaining distance.
- (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse feed rate.

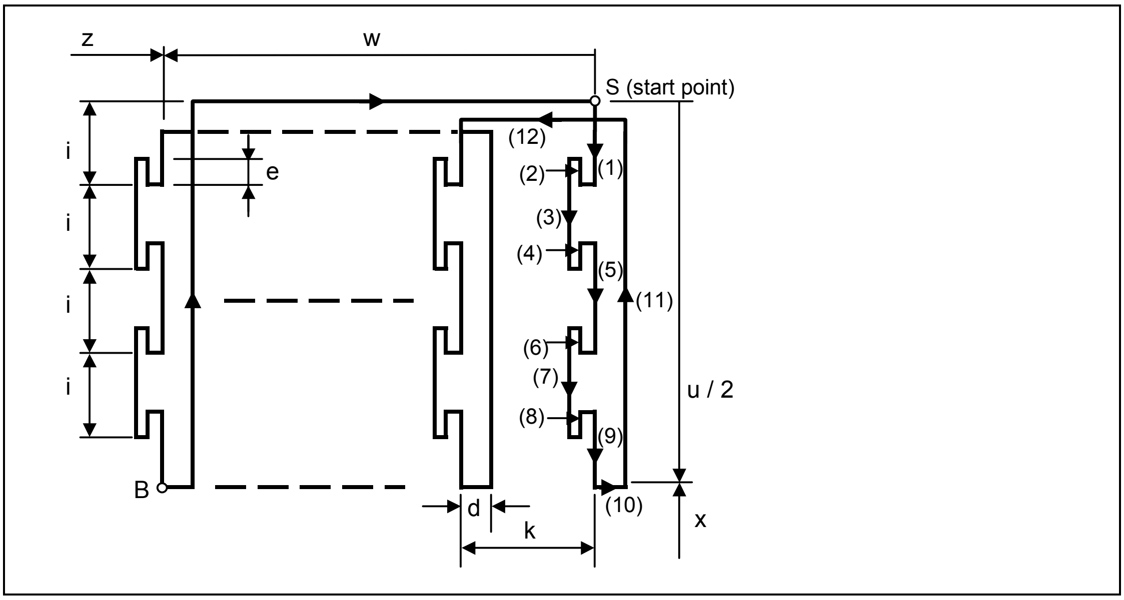
(f) Longitudinal cutting-off cycle (G75)

When the slotting end point coordinates, cut depth, cutting tool shift amount, and cutting tool relief amount at the cut bottom are commanded, automatic slotting is performed in the longitudinal direction of a given bar by G75 fixed cycle. The machining program is commanded as follows.

G75 Re ;

G75 X/(U) Z/(W) Pi Qk Rd Ff ;

Re	:	Retract amount e (when X/U, Z/W command is not given) (Modal)
X/U	:	B point coordinate (absolute/incremental)
Z/W	:	B point coordinate (absolute/incremental)
Pi	:	Tool shift amount (radius designation, incremental, sign not required)
Qk	:	Cut depth k (radius designation, incremental, sign not required)
Rd	:	Relief amount at cut bottom d (If sign is not provided, relief is made at the first cut bottom. If ⊖ sign is provided, relief is made not at the first cut bottom but at the second cut bottom and later.)
Ff	:	Feed rate



(g) Multiple repetitive thread cutting cycle (G76)

When the thread cutting start and end points are commanded, cut at any desired angle can be made by automatically cutting so that the cut section area (cutting torque) per time becomes constant in the G76 fixed cycle.
Various longitudinal threads can be cut by considering the thread cutting end point coordinate and taper height constituent command value.

Command Format

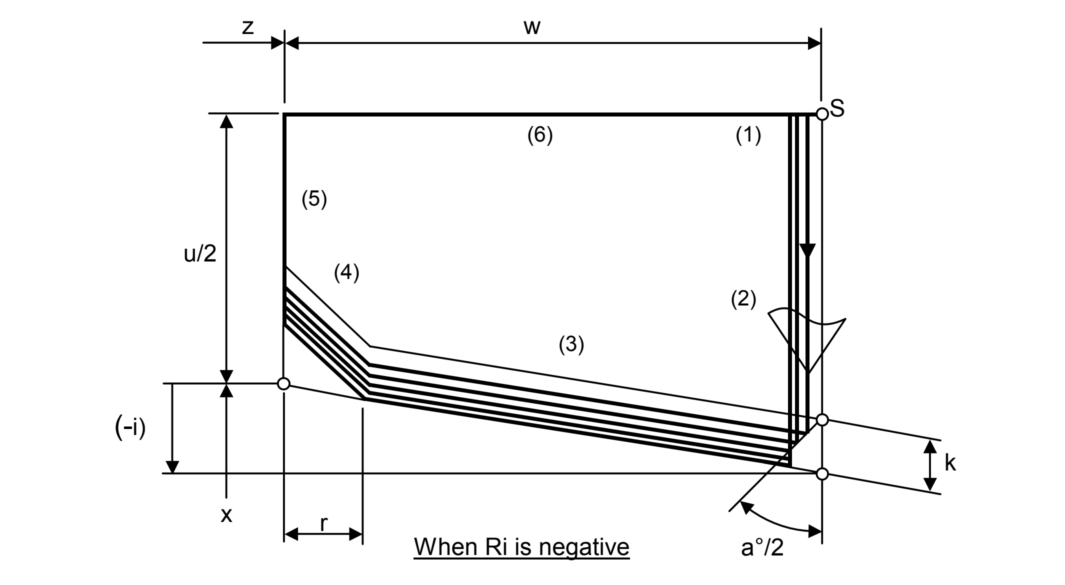
G76 Pmra Rd ;

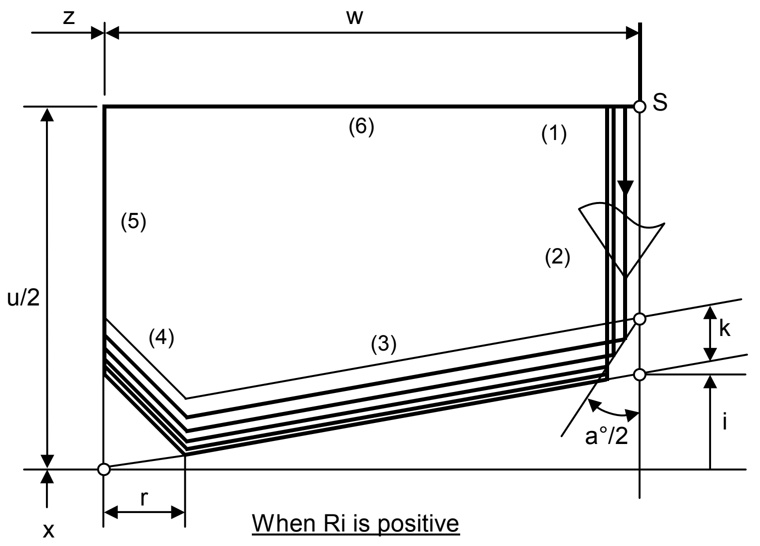
G76 X/U Z/W Ri Pk QΔd Fl ;

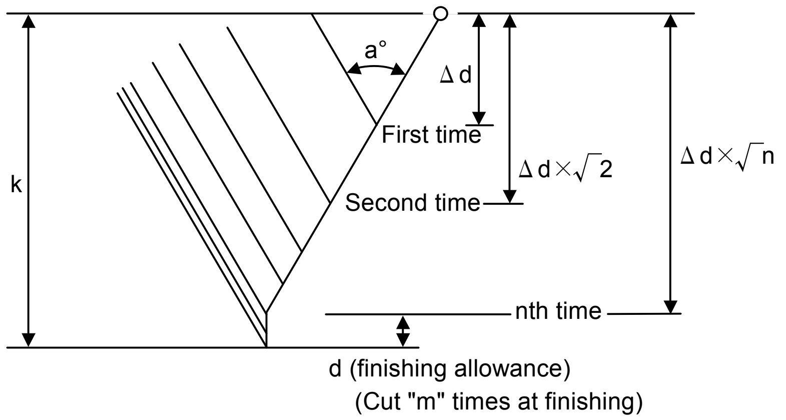
m	:	Cut count at finishing 01 to 99 (modal)
r	:	Chamfering amount 00 to 99 (modal). Set in 0.1-lead increments.
a	:	Nose angle (included angle of thread) 00 to 99 (modal) Set in 1-degree increments.
d	:	Finishing allowance (modal)
X/U	:	X axis end point coordinate of thread part. Designate the X coordinate of the end point in the thread part in an absolute or incremental value.
Z/W	:	Z axis end point coordinate of thread part. Designate the Z coordinate of the end point in the thread part in an absolute or incremental value.
i	:	Taper height constituent in thread part (radius value). When i = 0 is set, straight screw is made.
k	:	Thread height. Designate the thread height in a positive radius value.
Δd	:	Cut depth. Designate the first cut depth in a positive radius value.
l	:	Thread lead

Configuration of one cycle

In one cycle, (1), (2), (5), and (6) move at rapid traverse feed and (3) and (4) move at cutting feed designated in F.




z
w
S
(6)
(1)
(5)
(2)
u/2
k
(4)
(3)
a°/2
i
r
x
When Ri is positive


a°
Δ d
First time
Δ d × √2
Δ d × √n
k
Second time
nth time
d (finishing allowance)
(Cut "m" times at finishing)

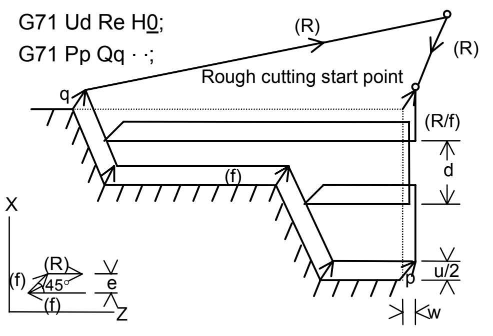
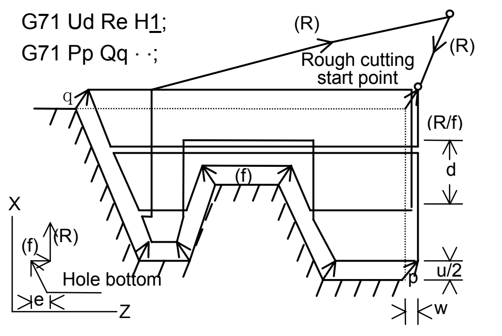
12.1.3.5 Multiple Repetitive Fixed Cycle for Turning Machining (Type II)

	E60	E68
M system	–	–
L system	–	○

Pocket shapes can be machined in the longitudinal rough cutting cycle (G71) and face rough cutting cycle (G72).

Command format (This is a command format when the G71 is commanded. The G72 command is based on the G71 command.)

G71 Ud Re Hh ; ← (can be omitted when values set in parameters are used)
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

<H0: Used for finished shapes without pockets>	<H1: Mainly used for finished shapes with pockets>
	

Hh : Pocket machining (modal) ... reversible parameter ("G71 pocket machining")
0: Select this only for finished shapes without hollow areas (pockets).
* With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle until the finished shape is finally traced.
1: This can be selected regardless of whether the finished shape has hollow (pocket) parts or not.
* A method that traces the finished shape with each cycle is used for the beginning of the pockets.

12.1.3.7 Fixed Cycle for Drilling (Type II)

	E60	E68
M system	–	–
L system	○	○

In the longitudinal hole drilling fixed cycle, the X axis is designated as the hole drilling axis. However, in the longitudinal hole drilling fixed cycle (type II), the Y axis can be designated as the hole drilling axis with the longitudinal hole drilling axis selection function.

The relationship between the longitudinal hole drilling axis selection signal's ON/OFF state and the hole drilling axis of the fixed cycle for drilling is shown below.

G code	Details	Y axis cross tap function selection signal state	Hole drilling axis
G80	Cancel	-	-
G83	Deep hole drilling cycle 1	ON	Z
		OFF	
G84 (G84.1)	Tapping cycle	ON	Z
		OFF	
G85	Boring cycle	ON	Z
		OFF	
G87	Deep hole drilling cycle 1	ON	Y
		OFF	X
G88 (G88.1)	Tapping cycle	ON	Y
		OFF	X
G89	Boring cycle	ON	Y
		OFF	X
G83.2	Deep hole drilling cycle 2	ON	Z/X
		OFF	

12.1.4 Mirror Image

12.1.4.1 Mirror Image by Parameter Setting

	E60	E68
M system	–	○
L system	–	○

A parameter is used to designate the axis for which the mirror image function is to be executed before the machining program is run. When mirror image is set to ON by the parameter, an operation that is symmetrical on the left and right or at the top or bottom is performed. Each axis has its own parameter.

12.1.4.2 External Input Mirror Image

	E60	E68
M system	–	○
L system	–	○

Signals from an external device (PLC) to request the mirror image operation either while a machining program is running or before it is run. When ON has been set for mirror image from an external device, an operation that is symmetrical on the left and right or at the top or bottom is performed. Each axis has its own request signal.

12.1.4.3 G Code Mirror Image

	E60	E68
M system	○	○
L system	–	–

Using a program for the left or right side of an image, this function can machine the other side of the image when a left/right symmetrical shape is to be cut.

Mirror image can be applied directly by a G code when preparing a machining program.

G code	Function
G50.1	G code mirror image cancel
G51.1	G code mirror image ON

The program format for the G code mirror image is shown below.

G51.1 Xx1 Yy1 Zz1 ;

G51.1 : Mirror image on
Xx1, Yy1, Zz1 : Command axes and command positions

With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

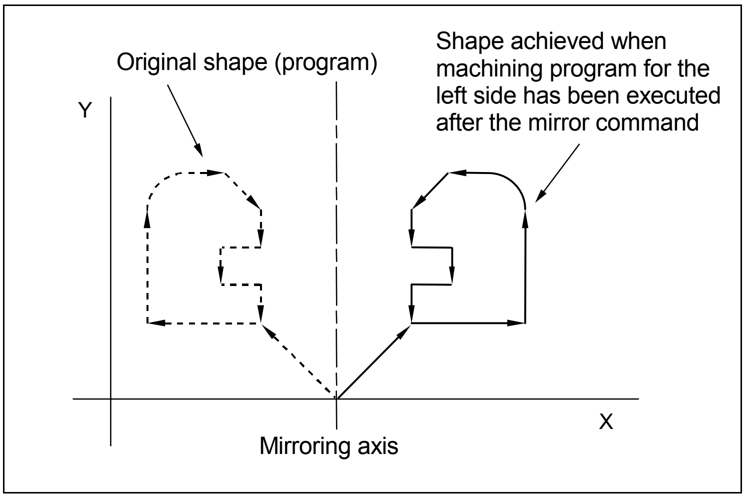
The program format for the G code mirror image cancel is shown below.

G50.1 Xx1 Yy1 Zz1 ;

G50.1 : Mirror image cancel
Xx1, Yy1, Zz1 : Command axes

The coordinate word indicates the axes for which the mirror image function is to be canceled and the coordinates are ignored.

In the case of G51.1 Xx1



12.1.5 Coordinate System Operation

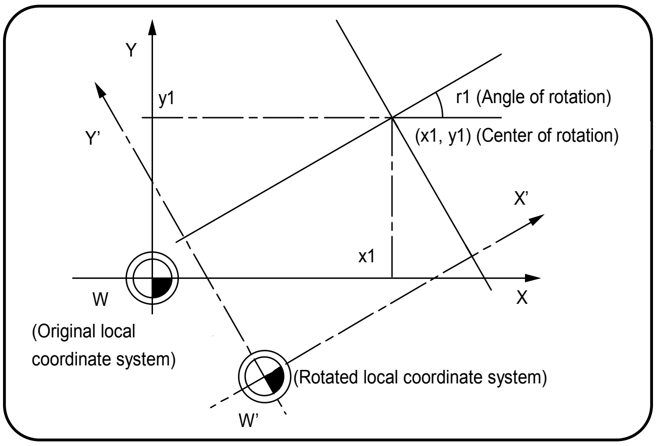
12.1.5.1 Coordinate Rotation by Program

	E60	E68
M system	○	○
L system	–	–

When it is necessary to machine a complicated shape at a position that has been rotated with respect to the coordinate system, you can machine a rotated shape by programming the shape prior to rotation on the local coordinate system, then specifying the parallel shift amount and rotation angle by means of this coordinate rotation command.
The program format for the coordinate rotation command is given below.

G68 Xx1 Yy1 Rr1 ;	Coordinate rotation ON
G69 ;	Coordinate rotation cancel

G68 : Call command
Xx1, Yy1 : Rotation center coordinates
Rr1 : Angle of rotation



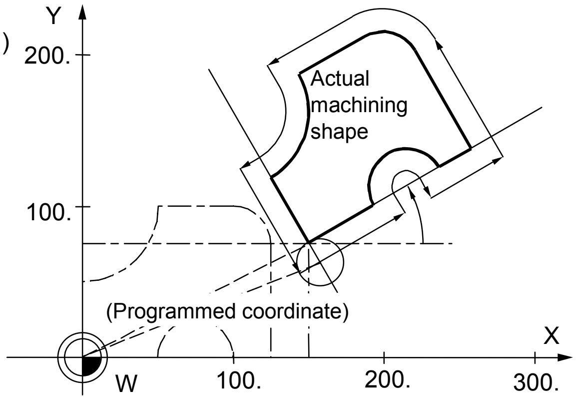
(1) Angle of rotation "r1" can be set in least input increment from –360° to 360° .
(2) The coordinates are rotated counterclockwise by an amount equivalent to the angle which is designated by angle of rotation "r1".
(3) The counter is indicated as the point on the coordinate system prior to rotation.
(4) The rotation center coordinates are assigned with absolute values.

(Example)

```
N01 G28 X Y Z ;
N02 G54 G52 X150. Y75. ;    Local coordinate system assignment
N03 G90 G01 G42 X0 Y0 ;     Tool radius compensation ON
N04 G68 X0 Y0 R30. ;        Coordinate rotation ON
N05 M98 H101 ;              Subprogram execution
N06 G69 ;                   Coordinate rotation cancel
N07 G54 G52 X0 Y0 ;         Local coordinate system cancel
N08 G00 G40 X0 Y0 ;         Tool radius compensation cancel
N09 M02 ;                   Completion
```

Sub program
(Shape programmed with original coordinate system)

```
N101 G90 G01 X50. F200 ;
N102 G02 X100. R25. ;
N103 G01 X125. ;
N104 Y75. ;
N105 G03 X100. Y100. R25. ;
N106 G01 X50. ;
N107 G02 X0 Y50. R50. ;
N108 G01 X0 Y0 ;
N109 M99 ;
```



12.1.6 Dimension Input

12.1.6.1 Corner Chamfering / Corner R

	E60	E68
M system	○	○
L system	○	○

This function executes corner processing by automatically inserting a straight line or arc in the commanded amount between two consecutive movement blocks (G01/G02/G03).
The corner command is executed by assigning the ",C" or ",R" command for the block at whose end point the corner is inserted.

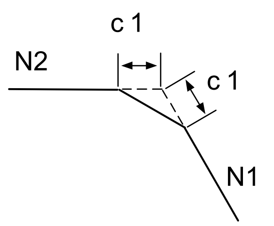
(1) Corner chamfering / Corner rounding I

When ",C" or ",R" is commanded for linear interpolation, corner chamfering or corner rounding can be inserted between linear blocks.

- **Corner chamfering**

Example:

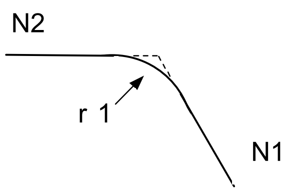
```
N1 G01 Xx1 Zz1,  Cc1  ;
N2 Zz2  ;
```



- **Corner rounding**

Example:

```
N1 G01 Xx1 Zz1,  Rr1  ;
N2 Zz2  ;
```



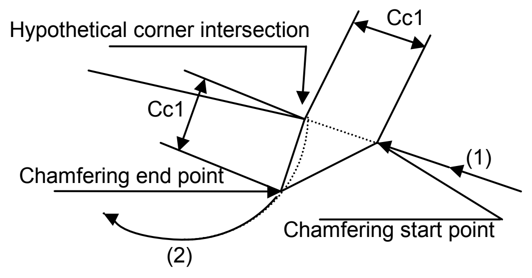
(Note 1) If a corner chamfering or corner rounding command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

(2) Corner chamfering / corner rounding II (L system)

When ",C" or ",R" is command in a program between linear-circular, corner chamfering or corner rounding can be inserted between blocks.

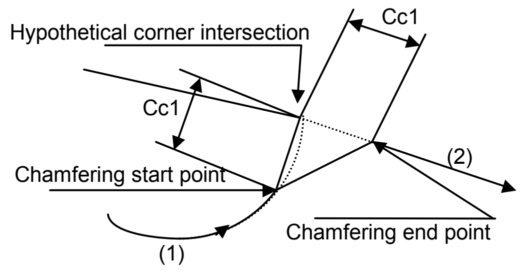
(a) Corner chamfering II (Linear – circular)

Example: G01 X_Z_ ,Cc1 ;
G02 X_Z_ Ii1 Kki ;



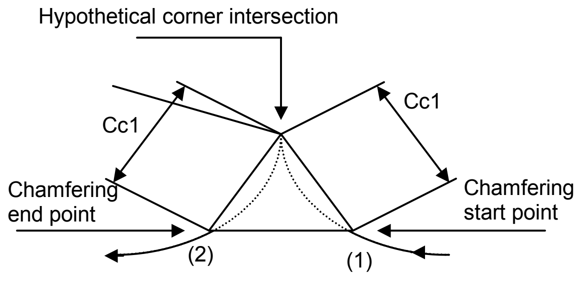
(b) Corner chamfering II (Circular - linear)

Example: G03 X_Z_ Ii1 Kk1 ,Cc1 ;
G01 X_Z_ ;



(c) Corner chamfering II (Circular - circular)

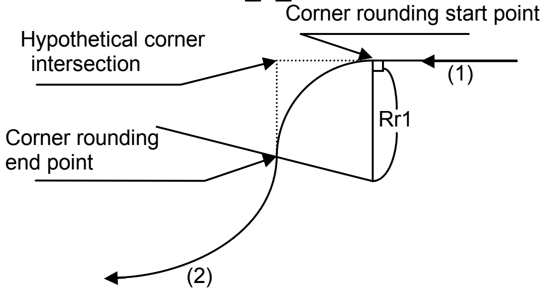
Example: G02 X_Z_ Ii1 Kk1 ,Cc1 ;
G02 X_Z_ Ii2 Kk2 ;



(3) Corner rounding

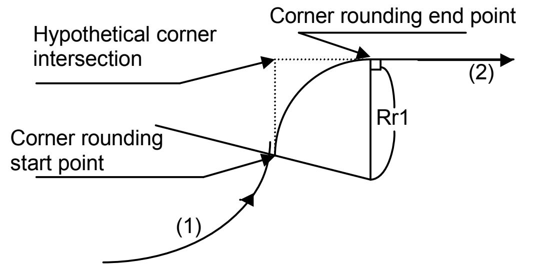
(a) Corner R II (Linear - circular)

Example: G01 X_Z_ ,Rr1 ;
G02 X_Z_ Ii1 Kk1 ;



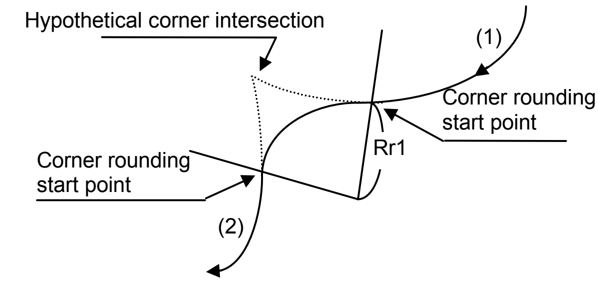
(b) Corner R II (Circular – linear)

Example: G03 X_Z_ Ii1 Kk1 ,Rr1 ;
G01 X_Z_ ;



(c) Corner R II (Circular – circular)

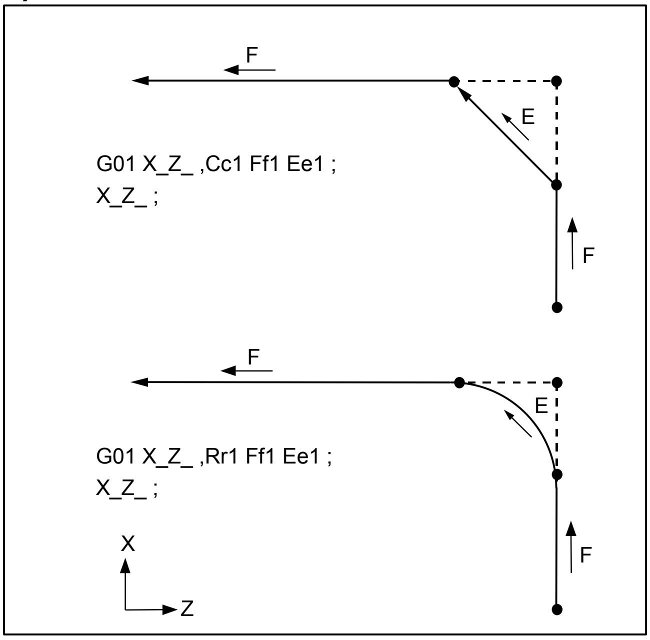
Example: G02 X_Z_ Ii1 Kk1 ,Rr1 ;
G02 X_Z_ Ii2 Kk2 ;



(4) Specification of corner chamfering / corner rounding speed E

An E command can be used to specify the speed for corner chamfering or rounding.
This enables a corner to be cut to a correct shape.

(Example)



An E command is a modal and remains effective for feeding in next corner chamfering or rounding.
An E command has two separate modals: synchronous and asynchronous feed rate modals. The effective feed rate is determined by synchronous (G95) or asynchronous (G94) mode.
If an E command is specified in 0 or no E command has been specified, the feed rate specified by an F command is assumed as the feed rate for corner chamfering or rounding.
Hold or non-hold can be selected (M system only) using a parameter for the E command modal at the time of resetting. It is cleared when the power is turned OFF (as it is with an F command).

(5) Corner chamfering / corner rounding (I, K designation) (L system)

With this command format, by means of parameter settings, corners are chamfered using the "I", "K" or "C" address without a comma, and corners are rounded using the "R" address.
The ",C" and ",R" commands with commas can also be used.

(a) Corner chamfering (I, K designation)

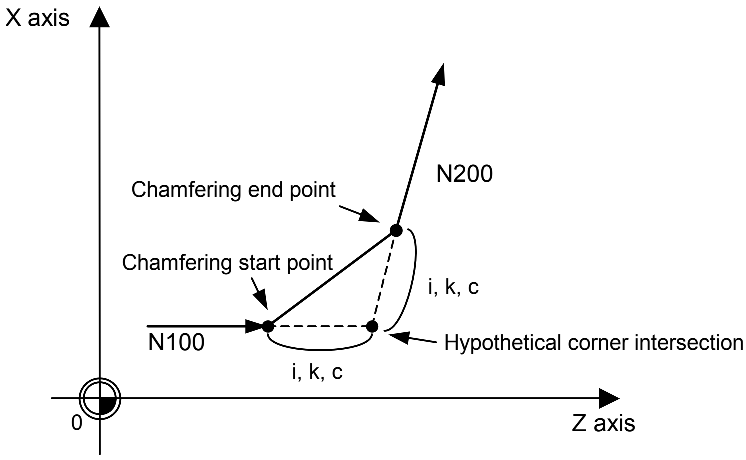
Corners are chamfered using the "I_", "K_" or "C_" address with no comma. Corners can be chamfered to any angle. Signs, if they are provided for the corner chamfering commands, are ignored.

Command format

N100 Xx/Uu Zz/Ww Ii/Kk/Cc ;

N200 Xx/Uu Zz/Ww ;

X/u	:	X-axis end point coordinate
Z/w	:	Z-axis end point coordinate
i/k/c	:	The length from the hypothetical corner intersection to the chamfering start point or chamfering end point is designated using the I, K or C address.



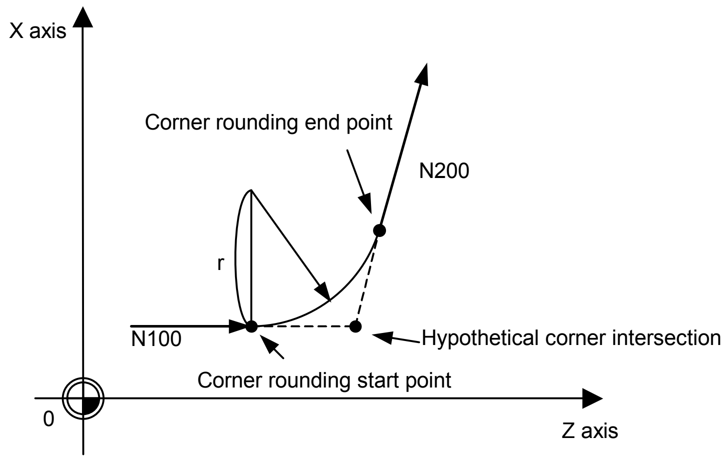
- If multiple "I", "K" or "C" addresses or duplicated addresses have been designated in the same block, the last command will take effect.
- If both corner chamfering and corner rounding commands are present in the same block, the last command will take effect.
- If "C" is used as the name of an axis, corner chamfering commands cannot be designated using the "C" address.
- If "C" is used as a 2nd miscellaneous function, corner chamfering commands cannot be designated using the "C" address.
- Corner chamfering commands using the "I" or "K" address cannot be designated in an arc command block. "I" and "K" are the arc center commands.

(b) Corner rounding (I, K designation)

Corners are rounded using the "R_" address with no comma. Corners can be rounded to any angle. Signs, if they are provided for the corner rounding commands, are ignored.

Command format

N100 Xx/Uu Zz/Ww Rr ;		
N200 Xx/Uu Zz/Ww ;		
x/u	:	X-axis end point coordinate
z/w	:	Z-axis end point coordinate
r	:	Radius of corner rounding arc



- If both corner chamfering and corner rounding commands are present in the same block, the last command will take effect.
- Corner rounding commands using the "R" address cannot be designated in an arc command block. "R" is the arc radius command.

12.1.6.2 Linear Angle Command

	E60	E68
M system	○	○
L system	○	○

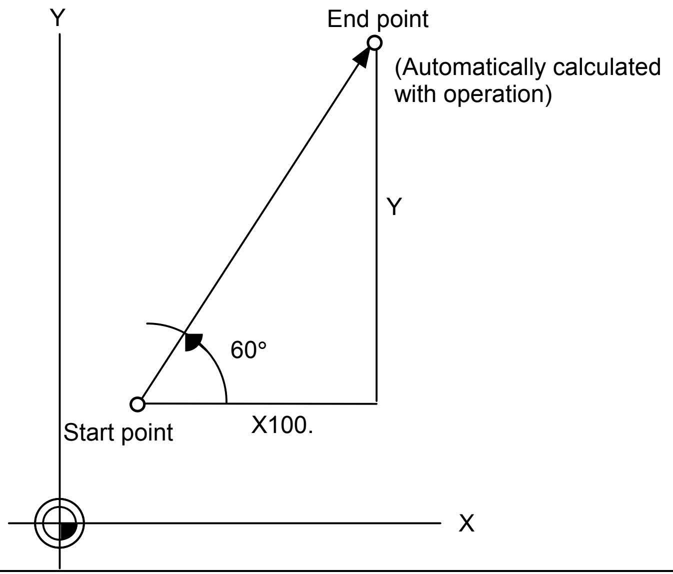
The end point coordinates are automatically calculated by assigning one element (one component of the selected plane) of the end point coordinates and the linear angle.

G17 Xx1 Aa1 ; or G17 Yy1 Aa1 ;

G17	:	Plane selection
Xx1, Yy1	:	1 element of the end point coordinate
Aa1	:	Angle

Example

G17 G91 X100. A60. ;



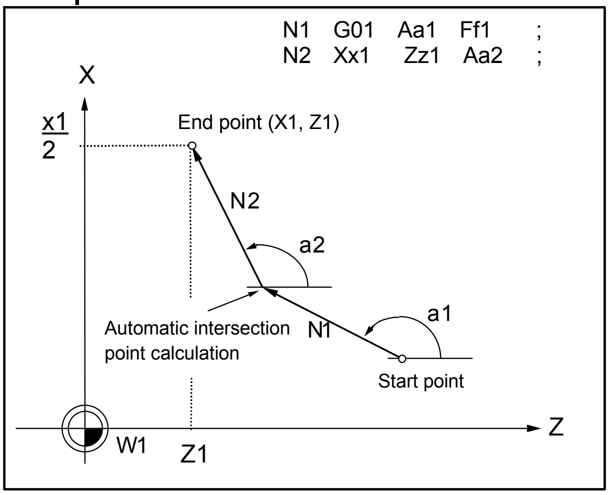
(Note 1) If the axis "A" or 2nd miscellaneous function "A" is used, address "A" is treated as the axis "A" command or the 2nd miscellaneous function, respectively.

12.1.6.3 Geometric Command

	E60	E68
M system	○	○
L system	○	○

When it is difficult to find the intersection point of two straight lines with a continuous linear interpolation command, this point can be calculated automatically by programming the command for the angle of the straight lines.

Example



a: Angle (°) formed between straight line and horizontal axis on plane.
The plane is the selected plane at this point.

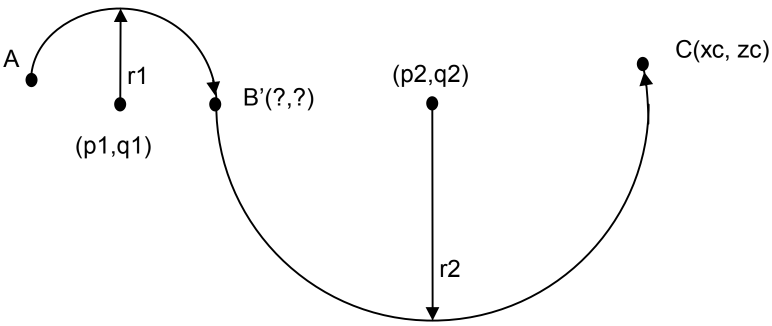
(Note 1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(1) Automatic calculation of two-arc contact

When two continuous circular arcs contact with each other and it is difficult to find the contact, the contact is automatically calculated by specifying the center coordinates or radius of the first circular arc and the end point absolute coordinates and center coordinates or radius of the second circular arc.

Example

```
   G18   G02   Ii1   Kk1   Ff1   ;
         G03   Xxc  Zzx   Ii2    Kk2   Ff2   ;
or
   G18   G02   Ii1   Kk1   Ff1   ;
         G03   Xxc  Zzc   Rr2   Ff2   ;
or
   G18   G02   Rr1   Ff1   ;
         G03   Xxc   Zzc   Ii2   Kk2   Ff2   ;
```



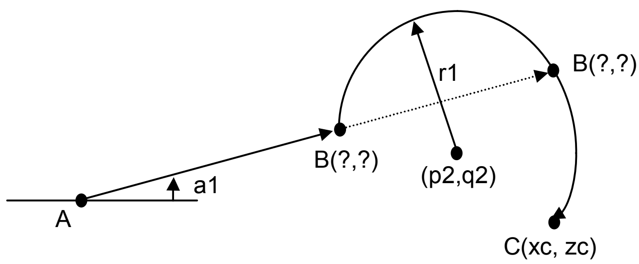
I and K are circular center coordinate incremental values; distances from the start point in the first block or distances from the end point in the second block. P and Q commands (X, Z absolute center coordinates of circular arc) can be given instead of I and K commands.

(2) Automatic calculation of linear-arc intersection

When it is difficult to find the intersections of a given line and circular arc, the intersections are automatically calculated by programming the following blocks.

Example

```
   G18   G01   Aa1   Ff1   ;
         G02   Xxc   Zzc   Ii2   Kk2   Hh2   Ff2   ;
```



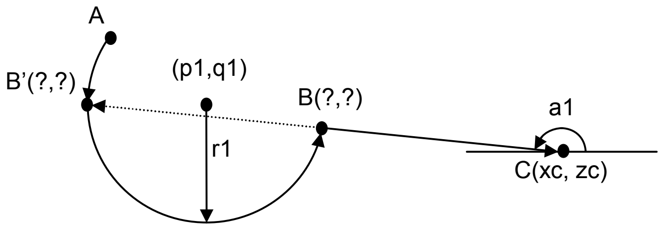
I and K : Incrimental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc
H = 0 : Intersection with shoter line
H = 1 : Intersection with longer line

(3) Automatic calculation of arc-linear intersection

When it is difficult to find the intersections of a given circular arc and line, the intersections are automatically calculated by programming the following blocks.

Example

```
G18   G03   Ii1   Kk1   Hh1   Ff1   ;
      G01 Xxc   Zzc    Aa1   Ff2   ;
```



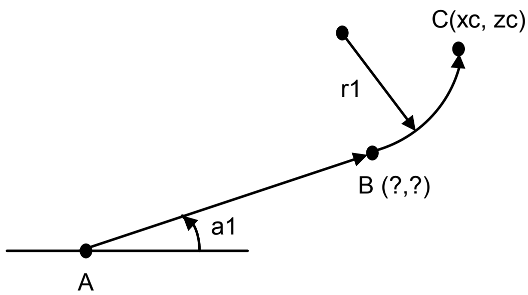
I and K ：Incrimental coordinates from circular end point
P and Q：Absolute center coordinates of circular arc (L3 only)
H = 0 ：Intersection with shoter line
H = 1 ：Intersection with longer line

(4) Automatic calculation of linear-arc contact

When it is difficult to find the contact of a given line and circular arc, the contact is automatically calculated by programming the following blocks.

Example

```
G01   Aa1   Ff1   ;
G03   Xxc   Zzc   Rr1   Ff2   ;
```

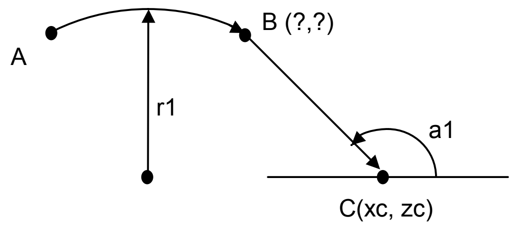


(5) Automatic calculation of arc-linear contact

When it is difficult to find the contact of a given circular arc and line, the contact is automatically calculated by programming the following blocks.

Example

```
G02   Rr1   Ff1   ;
G01   Xxc   Zzc   Aa1   Ff2   ;
```



12.1.6.4 Polar Coordinate Command

	E60	E68
M system	–	○
L system	–	–

With this function, the end point position is commanded with the radius and angle.

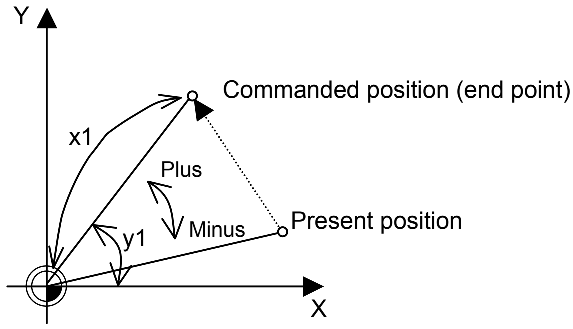
Command format

G16 ;	Polar coordinate command mode ON

G15 ;	Polar coordinate command mode OFF

Example of program

G1x ;	Plane selection for polar coordinate command (G17/G18/G19)
G16 ;	Polar coordinate command mode ON
G9x G01 Xx1 Yy1 F2000 ;	Polar coordinate command
:	G9x : Center selection for polar coordinate command (G90/G91) G90···The workpiece coordinate system zero point is the polar coordinate center. G91···The present position is the polar coordinate center. x1 : 1st axis for the plane···The radius commanded y1 : 2nd axis for the plane···The angle commanded



For G90/G17(X-Y plane)

G15 ;	Polar coordinate command mode OFF

12.1.7 Axis Control

12.1.7.1 High-speed Machining

12.1.7.1.3 High-speed Machining Mode III

	E60	E68
M system	–	Δ
L system	–	–

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed.
It is effective in increasing the speed at which dies with free-form curved surfaces are machined.
High-accuracy machining can be achieved by using this function in combination with the high-accuracy control function.

Command format

G05 P3 ;	High-speed machining mode III ON
G05 P0 ;	High-speed machining mode III OFF

Example of program

```
G28  X0.  Y0.  Z0.  ;
G91  G00  X-100.  Y-100.  ;
G01  F3000  ;
G05  P3  ;                   .... High-speed machining mode III ON
X0.1  ;                      }
X0.1  Y0.001  ;              }  Incremental modal linear cutting
X0.1  Y0.002  ;              }
G90  G00  X0  Y0  Z0  ;      }                   Rapid traverse
G02  X__Y__R__F__  ;         }  Absolute modal   R-designated arc
G03  X__Y__I__J__F__  ;      }                   Arc
•
•
•
G05  P0  ;                   .... High-speed machining mode III OFF
M02;
```

(1) The following G codes can be used in the high-speed machining mode III:
G00, G01, G02, G03, G90, G91, G17, G18 and G19.
(2) In the high-speed machining mode III, only the axis names, movement amounts (no variables or arithmetic operations), F commands and I/J/K/R/P commands can be designated. (Comments can be used.)
(3) The machining speed may be compromised depending on the number of characters in a block.

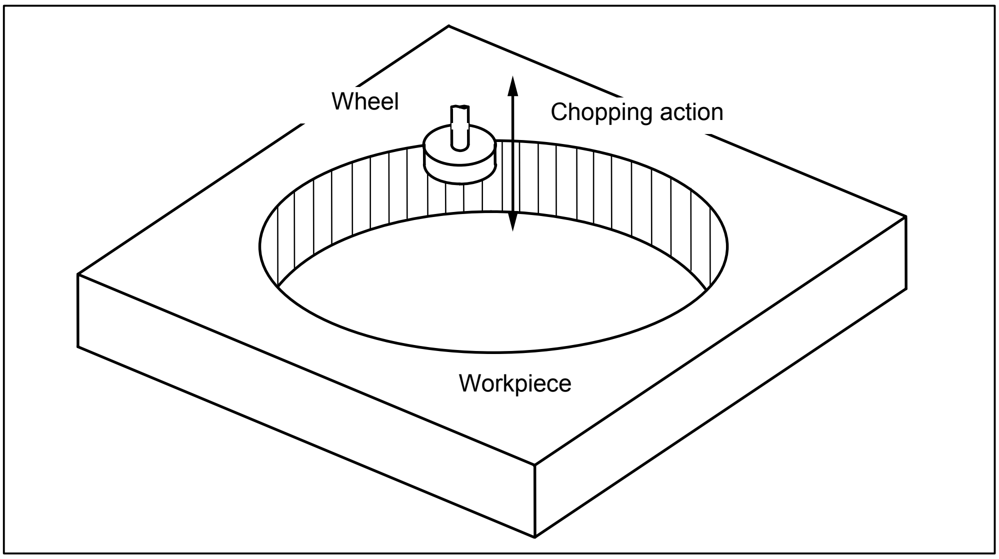
12.1.7.2 Chopping

12.1.7.2.1 Chopping

	E60	E68
M system	–	○
L system	–	○

This function continuously raises and lowers the chopping axis independently of the program operation when workpiece profiles are to be cut. It can be used for grinding operations using machining centers, for instance.
Which of the axes is to serve as the chopping axis is set by a parameter ahead of time.

(1) Chopping action

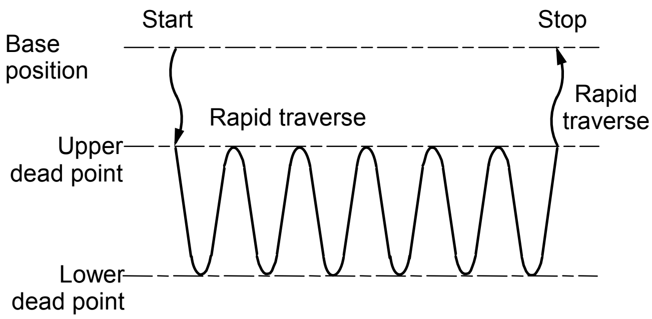


The chopping operation is initiated by setting the upper dead point position, lower dead point position and number of cycles (number of up/down movements per minute) and pressing the chopping start switch.

Note 1: The upper dead point position, lower dead point position and number of cycles are set and the start and stop commands are designated by input signals from the user PLC.

Note 2: The setting for the number of cycles differs according to the motor, inertia and other factors.

The chopping operation is performed as follows.



(a) The axis moves from the base position to the upper dead point by rapid traverse.
(b) Next, the axis moves repeatedly from the upper dead point to the lower dead point and then from the lower dead point to the upper dead point. (Sinusoidal waveforms)
The feed rate is tailored to achieve the number of cycles set for the up/down motion.

Chopping override

Override in 1% increments from 0% to 100% can be applied to the chopping operation.

Note: Bear in mind that the override increment differs according to the machine specifications.

12.1.7.5 Circular Cutting

	E60	E68
M system	○	○
L system	–	–

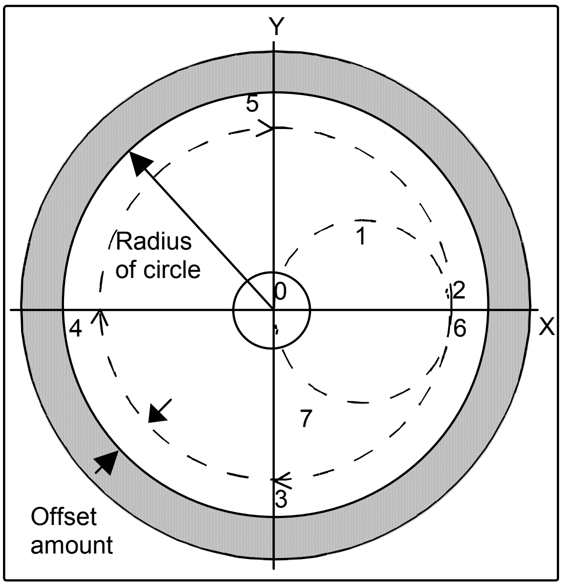
In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed serves as the center of the circle.

G code	Function
G12	CW (clockwise)
G13	CCW (counterclockwise)

The program format is given below.

G12/13 Ii Dd Ff ;

G12/13	: Circular cutting command
Ii	: Radius of complete circle
Dd	: Compensation number
Ff	: Feed rate



When the G12 command is used
(path of tool center)
0 → 1 → 2 → 3 → 4 → 5 → 6 → 7 → 0
When the G13 command is used
(path of tool center)
0 → 7 → 6 → 5 → 4 → 3 → 2 → 1 → 0

(Notes)

1. Circular cutting is undertaken on the plane which has been currently selected (G17, G18 or G19).
2. The (+) and (–) signs for the compensation amount denote reduction and expansion respectively.
3. When [radius (I) – compensation amount] is zero or negative, the alarm results to indicate an error in the circular cutting radius.

12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

	E60	E68
M system	○	○
L system	○	○

The parameters set from the screen can be changed using the machining programs.
The format used for the data setting is shown below.

```
G10  L50 ; ....... Data setting command
P  Major classification No.  A  Axis No.  N  Data No.  H  Bit type data     ;
P  Major classification No.  A  Axis No.  N  Data No.  D  Byte type data    ;
P  Major classification No.  A  Axis No.  N  Data No.  S  Word type data    ;
P  Major classification No.  A  Axis No.  N  Data No.  L  2-word type data  ;
G11 ; ..... Data setting mode cancel (data setting completed)
```

The following types of data formats can be used according to the type of parameter (axis-common and axis-independent) and data type.

With axis-common data
Axis-common bit-type parameter ------------------ P____ N____ H____ ;
Axis-common byte-type parameter ---------------- P____ N____ D____ ;
Axis-common word-type parameter --------------- P____ N____ S____ ;
Axis-common 2-word-type parameter------------- P____ N____ L____ ;

With axis-independent data
Axis-independent bit-type parameter-------------- P__ A___ N___ H___ ;
Axis-independent byte-type parameter ----------- P__ A___ N___ D___ ;
Axis-independent word-type parameter ---------- P__ A___ N___ S___ ;
Axis-independent 2-word-type parameter-------- P__ A___ N___ L___ ;

(Note 1) The order of addresses in a block must be as shown above.
(Note 2) For a bit type parameter, the data type will be H□ (□ is a value between 0 and 7).
(Note 3) The axis number is set in the following manner: 1st axis is "1", 2nd axis is "2", and so forth.
(Note 4) Command G10L50 and G11 in independent blocks. A program error will occur if not commanded in independent blocks.
(Note 5) Depending on the G90/G91 modal status when the G10 command is assigned, the data is used to overwrite the existing data or added.

12.1.9.2 Compensation Data Input by Program

	E60	E68
M system	○	○
L system	○	○

(1) Workpiece coordinate system offset input

The value of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.

G code	Function
G10 L2 P0	External workpiece coordinate system setting
G10 L2 P1	Workpiece coordinate system 1 setting (G54)
G10 L2 P2	Workpiece coordinate system 2 setting (G55)
G10 L2 P3	Workpiece coordinate system 3 setting (G56)
G10 L2 P4	Workpiece coordinate system 4 setting (G57)
G10 L2 P5	Workpiece coordinate system 5 setting (G58)
G10 L2 P6	Workpiece coordinate system 6 setting (G59)
G10 L20 Pn	Extended workpiece coordinate system setting (G54.1 Pn) (n = 1 to 48)

The format for the workpiece coordinate system setting commands is shown below.

G10 L2 Pp1 Xx1 Yy1 Zz1 ;

G10 L2	: Parameter change command
Pp1	: Workpiece coordinate No.
Xx1, Yy1, Zz1	: Settings

(Note) L2 can be omitted. Omitting Pp1 results in a program error. [M system]

(2) Tool offset input

The tool offset amounts, which have been set from the screen, can be input by program commands.
The command format differs between the [M system] and the [L system]. The respective command format must be set by a parameter.

[M system]

G code	Function
G10 L10	Tool length shape offset amount
G10 L11	Tool length wear offset amount
G10 L12	Tool radius shape offset amount
G10 L13	Tool radius wear offset amount

The tool offset input format is as follows.

G10 Ll1 Pp1 Rr1 ;	
G10 Ll1	: Command for setting offset amount
Pp1	: Offset No.
Rr1	: Offset amount

(Note) When Ll1 has been omitted, the tool length shape offset amount is set. Omitting Pp1 results in a program error.

[L system]

G code	Function
G10 L10	Tool length offset amount
G10 L11	Tool wear offset amount

The tool offset input format is as follows.

G10 L10(L11) Pp1 Xx1 Zz1 Rr1 Qq1 ;	
G10 L10(L11)	: Command for setting offset amount
Pp1	: Offset No.
Xx1	: X axis offset amount
Zz1	: Z axis offset amount
Rr1	: Nose R compensation amount
Qq1	: Hypothetical tool nose point

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

	E60	E68
M system	○	○
L system	○	○

When tapping mode commands are issued, the NC system is set to the following internal control modes required for tapping.

1. Cutting override is fixed at 100%.
2. Deceleration commands at joints between blocks are invalid.
3. Feed hold is invalid.
4. Single block is invalid.
5. "In tapping mode" signal is output.

G code	Function
G63	Tapping mode ON

The tapping mode command will be canceled with the following commands:

- Exact stop check mode (G61)
- Automatic corner override (G62) [M system]
- Cutting mode (G64)
- High-accuracy control mode command (G61.1) [M system]

12.1.10.2 Cutting Mode

	E60	E68
M system	○	○
L system	○	○

When a cutting mode command is issued, the NC system is set to the cutting mode that enables smooth cutting surface to be achieved. In this mode, the next block is executed continuously without the machine having to decelerate and stop between the cutting feed blocks: this is the opposite of what happens in the exact stop check mode (G61).

G code	Function
G64	Cutting mode ON

The cutting mode command will be canceled with the following commands:

- Exact stop check mode (G61)
- Automatic corner override (G62) [M system]
- Tapping mode (G63) [M system]
- High-accuracy control mode command (G61.1) [M system]

The machine is in the cutting mode status when its power is turned on.

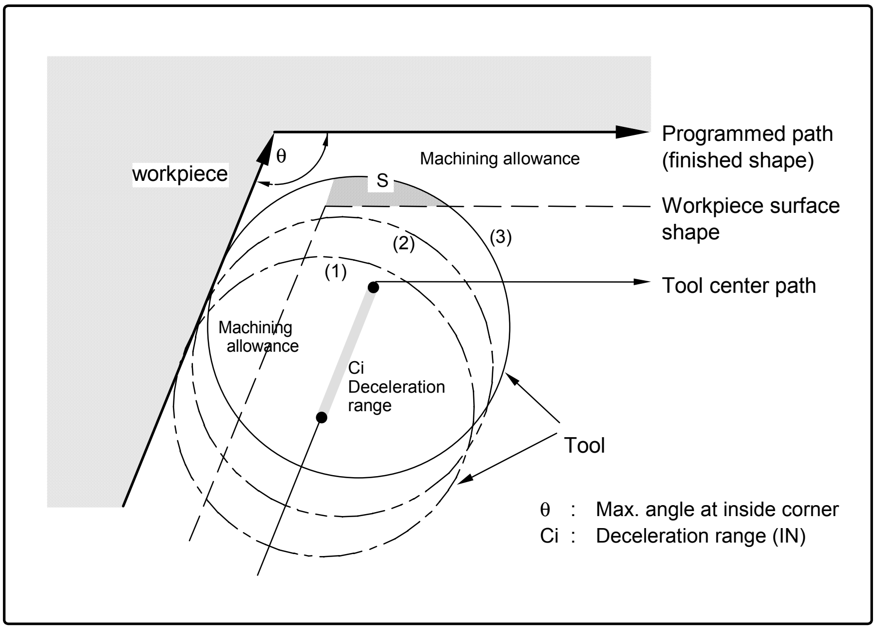
12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

	E60	E68
M system	○	○
L system	○	○

To prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this function automatically applies an override on the cutting feed rate so that the cutting amount is not increased for a set time at the corner.
Automatic corner override is valid only during tool radius compensation.
The automatic corner override mode is set to ON by the G62 command and it is canceled by any of the G commands below.

G40 ...... Tool radius compensation cancel
G61 ...... Exact stop check mode
G63 ...... Tapping mode
G64 ...... Cutting mode
G61.1.... High-accuracy control mode



Operation

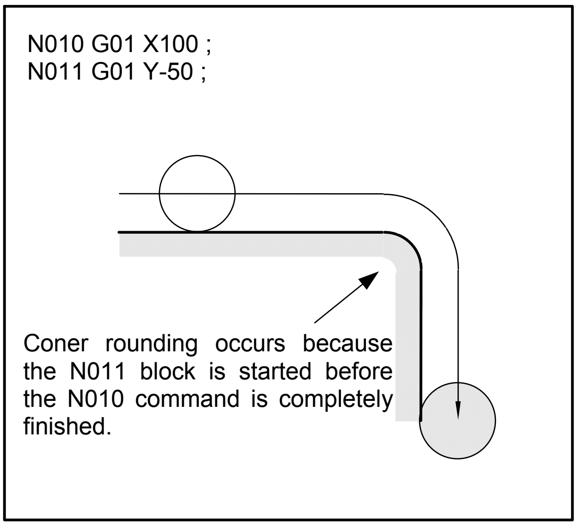
(a) When automatic corner override is not to be applied :
When the tool moves in the order of (1) → (2) → (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.

(b) When automatic corner override is to be applied :
When the inside corner angle θ in the figure above is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

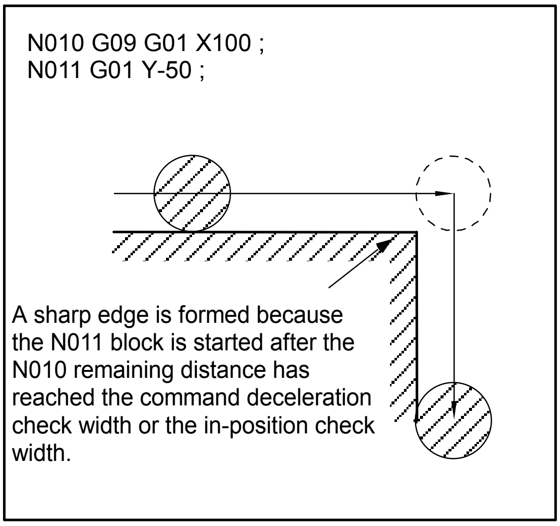
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another before executing the next block. This is effective to alleviate the machine shock and prevent the corner rounding when the feed rate of the control axis changes suddenly.

Without deceleration check



With deceleration check



The conditions for executing deceleration check are described below.

(1) Deceleration check in the rapid traverse mode

In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.

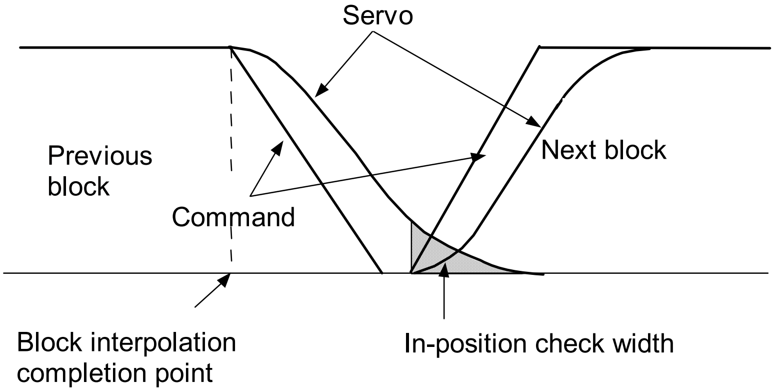
(2) Deceleration check in the cutting feed mode

In the cutting feed mode, the deceleration check is performed at the end of block when any of the conditions below is applicable before executing the next block.

(a) When G61 (exact stop check mode) is selected.
(b) When the G09 (exact stop check) is issued in the same block.
(c) When the error detect switch (external signal) is ON.

(3) Deceleration check system

Deceleration check is a system that executes the next block only after the command deceleration check is executed as shown below, and it has been confirmed that the position error amount, including the servo system, is less than the in-position check width (designated with parameter or with ",I" in same block).



12.2.2.1 Exact Stop Mode

	E60	E68
M system	○	○
L system	○	○

A deceleration check is performed when the G61 (exact stop check mode) command has been selected. G61 is a modal command. The modal command is released by the following commands.

G62 .... Automatic corner override
G63 .... Tapping mode
G64 .... Cutting mode
G61.1/G08P1 ..... High-accuracy control mode [M system]

Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.2 Exact Stop Check

	E60	E68
M system	○	○
L system	○	○

A deceleration check is performed when the G09 (exact stop check) command has been designated in the same block.
The G09 command is issued in the same block as the cutting command. It is an unmodal command.
Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.3 Error Detect

	E60	E68
M system	○	○
L system	○	○

To prevent rounding of a corner during cutting feed, the operation can be changed by turning an external signal switch ON so that the axis decelerates and stops once at the end of the block and then the next block is executed.
The deceleration stop at the end of the cutting feed block can also be commanded with a G code.
Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.4 Programmable In-position Check

	E60	E68
M system	O	O
L system	O	O

This command is used to designate the in-position width, which is valid when a linear interpolation command is assigned, from the machining program. The in-position width designated with a linear interpolation command is valid only in cases when the deceleration check is performed, such as:

- When the error detect switch is ON.
- When the G09 (exact stop check) command has been designated in the same block.
- When the G61 (exact stop check mode) command has been selected.

G01 X_ Z_ F_ ,I_;	
X_,Z_	: Linear interpolation coordinates of axes
F_	: Feed rate
,I_	: In-position width

This command is used to designate the in-position width, which is valid when a positioning command is assigned, from the machining program.

G00 X_ Z_ ,I_;	
X_,Z_	: Positioning coordinates of axes
,I_	: In-position width

In-position check operation

After it has been verified that the position error between the block in which the positioning command (G00: rapid traverse) is designated and the block in which the deceleration check is performed by the linear interpolation command (G01) is less than the in-position width of this command, the execution of the next block is commenced.

12.2.3 High-Accuracy Control

12.2.3.1 High-accuracy Control (G61.1/G08P1)

	E60	E68
M system	–	○
L system	–	–

With this function, the error caused by the accuracy in control system during machining is to be improved. There are parameter (of turning initial high-accuracy ON) method and G code command method available in order to achieve the high-accuracy control mode.
With the normal control method, there are problems as indicated below.

(1) Corner rounding occurs at the corner where two lines are connected because the next command movement starts before the previous command finishes. (Refer to Fig. 1)
(2) When cutting with the circle command, an error occurs further inside the commanded path, resulting in a smaller finish. (Refer to Fig. 2)

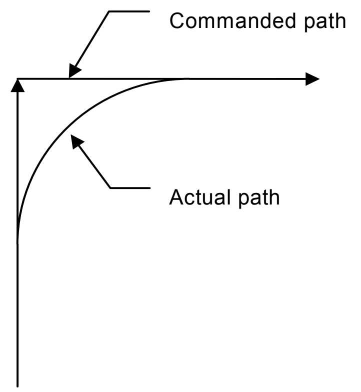


Fig.1 Roundness at linear corner

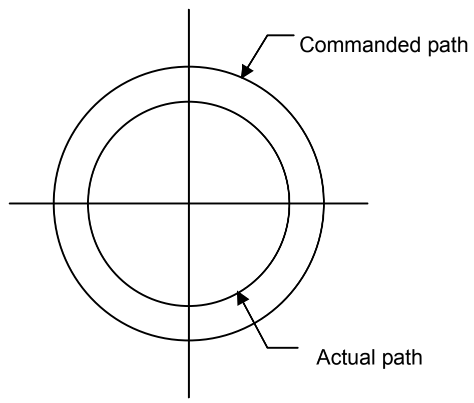


Fig.2 Error by radius reduction with arc command

This function uses the following six functions to reduce the shape error while minimizing the extension of machining time.

(1) Acceleration/deceleration before interpolation (Linear acceleration/deceleration)

(2) Optimum speed control

(3) Vector accuracy interpolation

(4) Feed forward control

(5) Arc entrance/exit speed control

(6) S-pattern filter control

The high-accuracy control is commanded as follows.
G61.1 or G08 can be selected by the parameter.

G61.1 Ff1 ;	
G61.1	: High-accuracy control ON
Ff1	: Feed rate command

High-accuracy control mode is validated from the block containing the G61.1 command.
"G61.1" high-accuracy control mode is canceled with one of the G code group 13's functions.

G08 P1(P0) ;	
G08	: High-accuracy control mode
P1	: High-accuracy control mode start
P0	: High-accuracy control mode end

"G08P1" high-accuracy control mode is canceled with P0 in G08.
Command G08P_ in can independent block.
The decimal places below the decimal point are ignored for P address.

(Note) G code group in G08 is "0"; the priority is given to the function of the G code group 0 over that of the G code group 13. After "G08 P1" is commanded, G code group 13 is changed automatically to G64 (cutting) mode. Other command of "13" results in error. Even if high-accuracy control mode is canceled by "G08 P0" command, G64 (cutting) mode will not be changed. If you want to return to the function of the G code group "13" when "G08 P1" has been commanded, command again after high-accuracy control mode is canceled.

(1) Acceleration / deceleration before interpolation

By accelerating /decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved.
With the arc commands, the radius reduction error can be significantly minimized.
Furthermore, since constant inclination acceleration/deceleration is performed, the time taken for positioning at microscopically small distances in the G00 command is reduced.

(Note 1) Whether acceleration/deceleration before interpolation in the rapid traverse command (G00) is to be performed always or not can be selected using a parameter setting independently from the high-accuracy control assignment.

(2) Optimum speed control

(a) Optimum corner deceleration

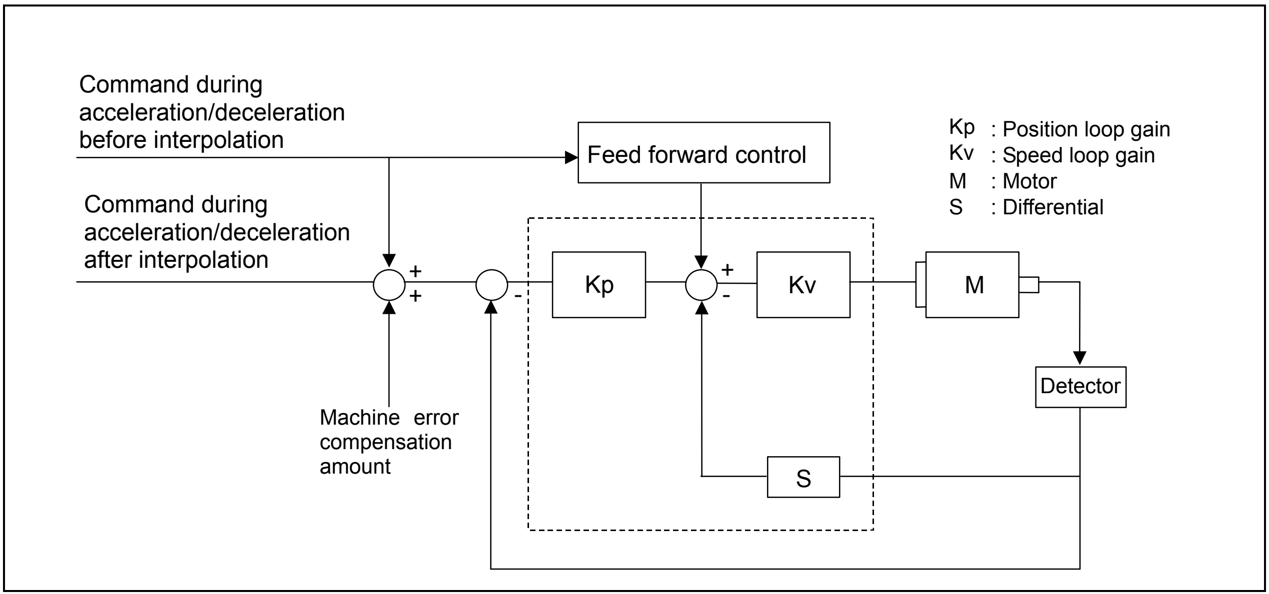
By calculating the angle of the seam between blocks, and carrying out acceleration/deceleration control in which the corner is passed at the optimum speed, highly accurate edge machining can be realized. When the corner is entered, that corners optimum speed (optimum corner speed) is calculated from the angle with the next block. The machine decelerates to that speed in advance, and then accelerates back to the command speed after the corner is passed.

(b) Arc speed clamp

During circular interpolation, even when moving at a constant speed, acceleration is generated as the advance direction constantly changes. When the arc radius is large enough compared to the commanded speed, control is carried out at the commanded speed. However, when the arc radius is relatively small, the speed is clamped so that the generated acceleration does not exceed the tolerable acceleration/deceleration speed before interpolation calculated with the parameters.
This allows arc cutting to be carried out at an optimum speed for the arc radius.

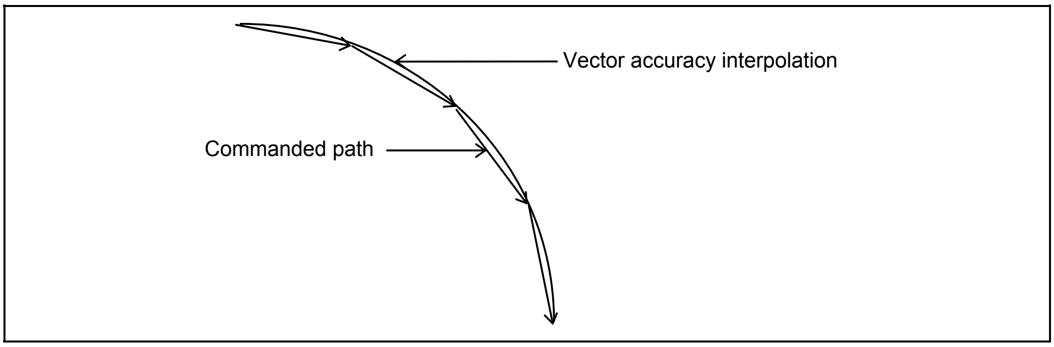
(3) Feed forward control

With this function, the constant speed error caused by the position loop control of the servo system can be greatly reduced. However, if machine vibration occurs as the feed forward coefficient is increased, use this function together with the smooth high gain (SHG) and more stably compensate the delay by the servo system's position loop so that a high accuracy control is realized. As the response is smoother during acceleration/deceleration, the position loop gain can be increased.



(4) Vector accuracy interpolation

When a fine segment is commanded and the angle between the blocks is extremely small (when not using optimum corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.



(5) Arc entrance/exit speed control

There are cases where the speed fluctuates and the machine vibrates at the joint from the straight line to arc or from the arc to straight line.

This function decelerates to the deceleration speed before entering the arc and after exiting the arc to reduce the machine vibration. If this is overlapped with corner deceleration, the function with the slower deceleration speed is valid.

(6) S-pattern filter control

This control interpolates while smoothing the changes in the segments distributed to each axis element with vector accuracy interpolation. With this, the fluctuation amplified by feed forward control is reduced and the effect onto the machine is reduced.

12.3 Programming Support Functions

12.3.1 Playback

	E60	E68
M system	○	○
L system	○	○

By repeatedly operating the controls on the panel, the amounts by which the machine is to move by jog feed, rapid traverse and handle feed can be converted into the command format of the control unit, and by repeatedly writing this data into the memory, machining programs for all the steps can be prepared.

12.3.2 Address Check

	E60	E68
M system	○	○
L system	○	○

When a machining program is to be run, it can be checked in 1-word units. A parameter is used to select whether or not to conduct an address check.

Program address check operation

In addition to the conventional program check, a simple check in 1-word units is conducted. If letters of the alphabet follow successively, a program error results.
(Word: Consists of one letter followed by a number composed of several digits.)
With the conventional method, when a letter was not followed by a number, that the number was assumed to be zero, however, now an error will result when this new check is performed.
An error will not result in the following cases:

1) Machine language
2) Comment statements

Example of a program address check

Example 1: When the letter is not followed by a number
G28X; → Program should be reviewed and changed to G28X0; , etc.
Example 2: When there is an illegal character string
TEST; → Program should be reviewed and changed to "(TEST);", etc.

13. Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

	E60	E68
M system	○	○
L system	○	○

This function compensates for the error (backlash) produced when the direction of the machine system is reversed.
The backlash compensation can be set in the cutting feed mode or rapid traverse mode.
The amount of backlash compensation can be set separately for each axis. It is set using a number of pulses in increments of one-half of the least input unit. The output follows the output unit system.
The "output unit system" is the unit system of the machine system (ball screw unit system).
The amount of compensation for each axis ranges from 0 to ±9999 (pulses).

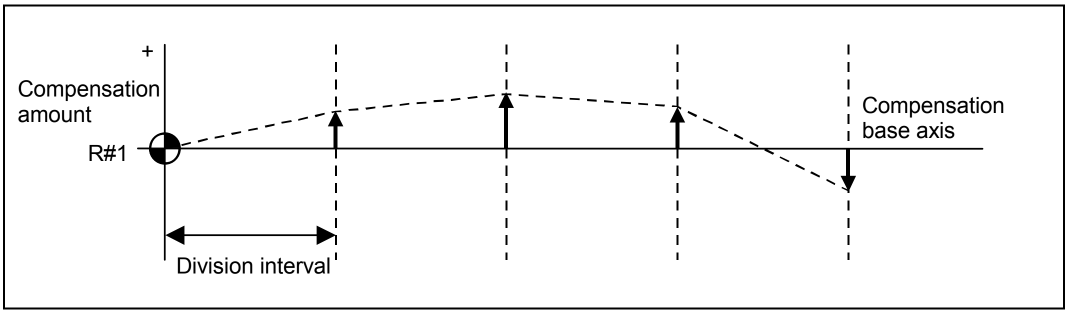
13.1.2 Memory-type Pitch Error Compensation

	E60	E68
M system	○	○
L system	○	○

The machine accuracy can be improved by compensating for the errors in the screw pitch intervals among the mechanical errors (production errors, wear, etc.) of the feed screws.
The compensation positions and amounts are stored in the memory by setting them beforehand for each axis, and this means that there is no need to attach dogs to the machine.
The compensation points are divided into the desired equal intervals.

1. Division intervals of compensation points : 1 to 9999999 (μm)
2. Number of compensation points : 1024
3. Compensation amount : –128 to 127 (output unit)
4. No. of compensated axes : 10 axes (including number of axes for relative position error compensation)

(1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed if return to reference position is not made for the compensation base axis or compensation execution axis after the controller power is turned ON and the servo is turned ON.
(2) When the compensation base axis is a rotary axis, select the dividing intervals so that one rotation can be divided.



(3) As shown in the figure above, highly individualized compensation control is exercised using the minimum output units with linear approximation for the compensation intervals between the compensation points.

(Note 1) Compensation points 1,024 is a total including the points for memory-type relative position error compensation.
(Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.3 Memory-type Relative Position Error Compensation

	E60	E68
M system	○	○
L system	○	○

Machine accuracy can be improved by compensating a relative error between machine axes, such as a production error or time aging.
The compensation base axis and compensation execution axis are set by using parameters.
The compensation points are divided at any desired equal intervals.

1. Compensation point dividing intervals : 1 to 9999999 (μm)
2. Number of compensation points : 1024
3. Compensation amount : –128 to 127 (detection unit)
4. No. of compensated axes : 10 axes (including number of axes for memory type pitch error compensation.)

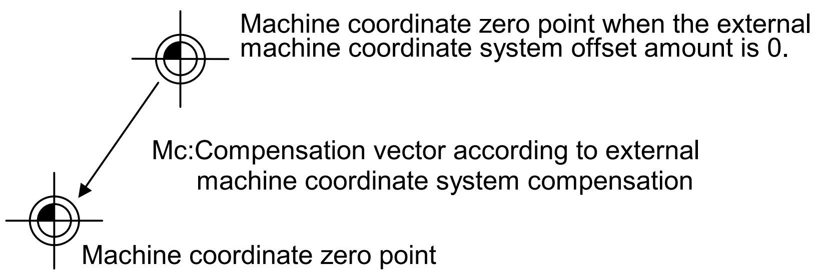
(1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed if return to reference position is not made for the compensation base axis or compensation execution axis after the controller power is turned ON and the servo is turned ON.
(2) When the compensation base axis is a rotary axis, select the dividing intervals so that one rotation can be divided.
(3) Since all coordinate systems of compensation execution axes are shifted or displaced by the compensation amount when the relative position error compensation is made, the stroke check point and machine coordinate system are also shifted or displaced.

(Note 1) Compensation points 1,024 is a total including the points for memory-type pitch error compensation.
(Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.4 External Machine Coordinate System Compensation

	E60	E68
M system	○	○
L system	○	○

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counter (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.

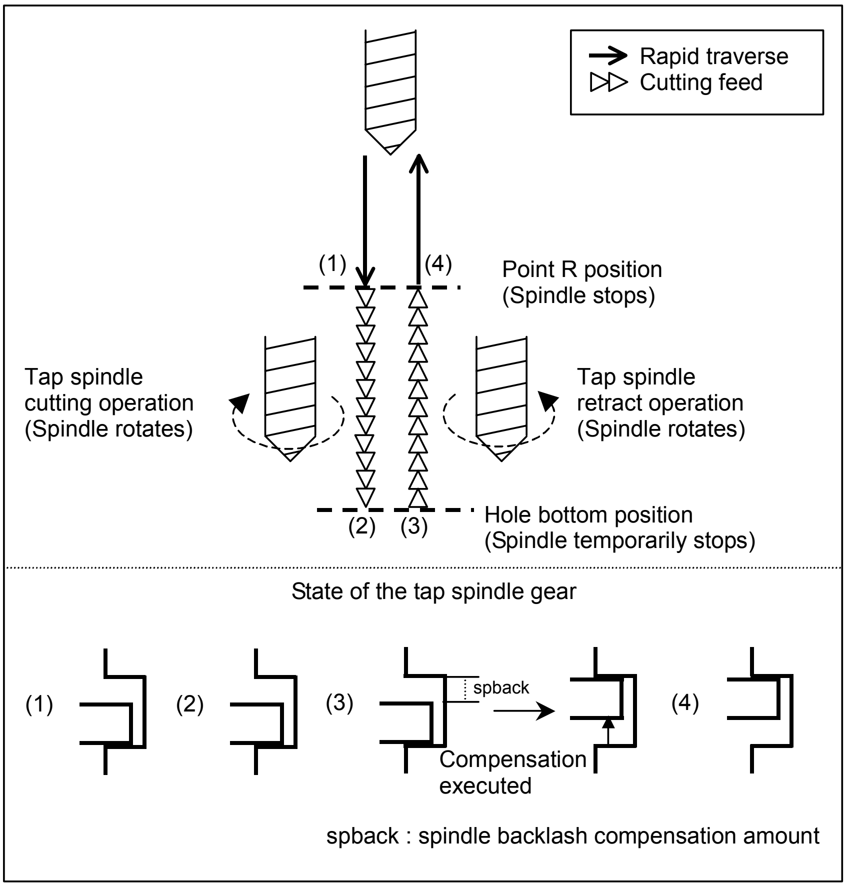


13.1.9 Spindle Backlash Compensation

	E60	E68
M system	–	○
L system	–	○

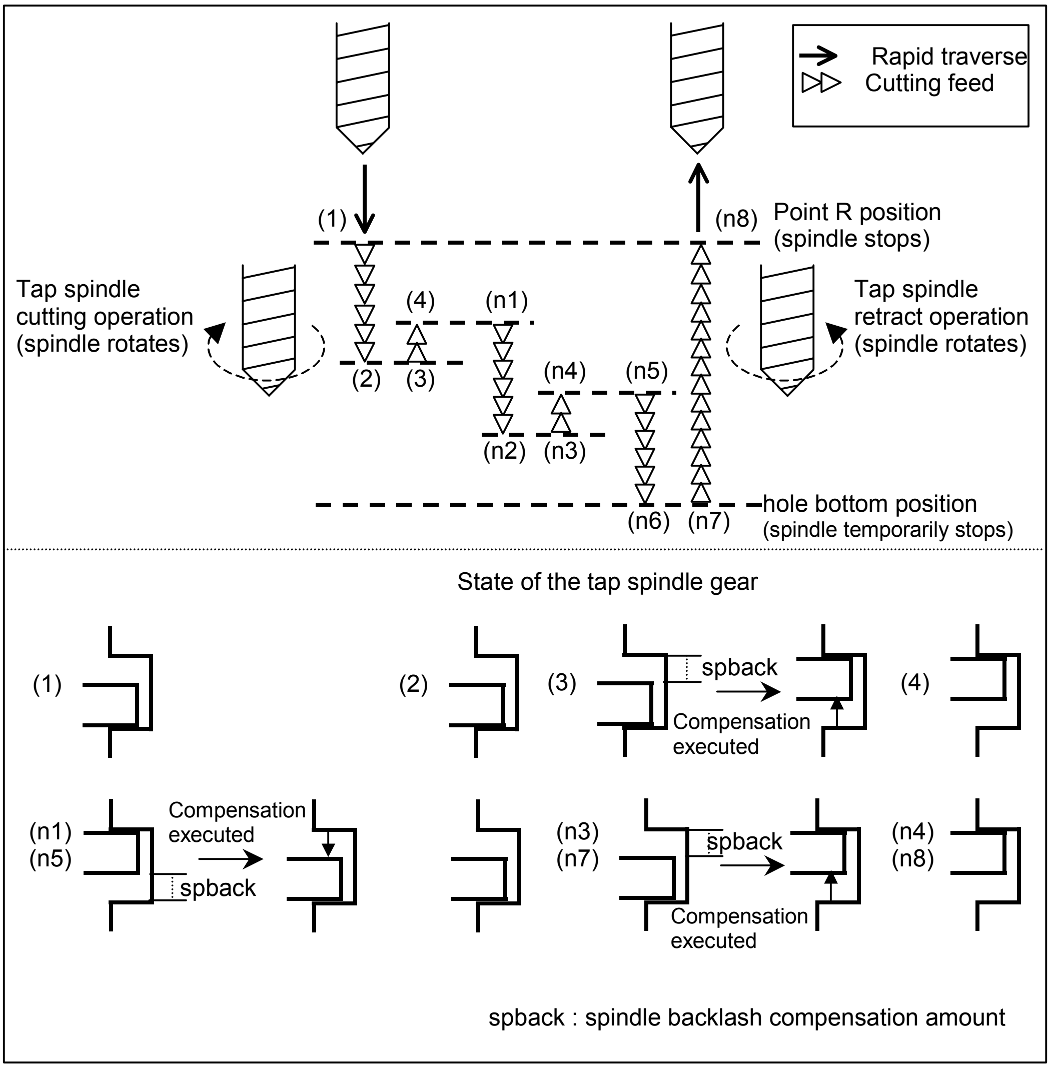
This function compensates for the backlash produced when the spindle is reversed in tapping. This is effective in improving the accuracy of tapping.
The amount of spindle backlash compensation is set by parameter.

(1) Synchronous tapping cycle
Backlash compensation is only performed in the tapping retract operation after the machine has reached the hole bottom position ((3) in the figure below).



(2) Pecking tapping cycle / deep-hole tapping cycle
Backlash compensation is performed at three times as follows.
(a) In the retract operation after the tap spindle cutting operation ((3) and (n3) in the figure below).
(b) In the tap spindle cutting operation for the second time or later ((n1) and (n5) in the figure below).
(c) In the tapping retract operation after reaching the hole bottom point ((n7) in the figure below).

Compensation frequency at (a) and (b) is changed according to the number of cuttings at each time.



(Note) In the deep-hole tapping cycle, point R position is set at (4), (n1), (n4), and (n5) in the figure above.

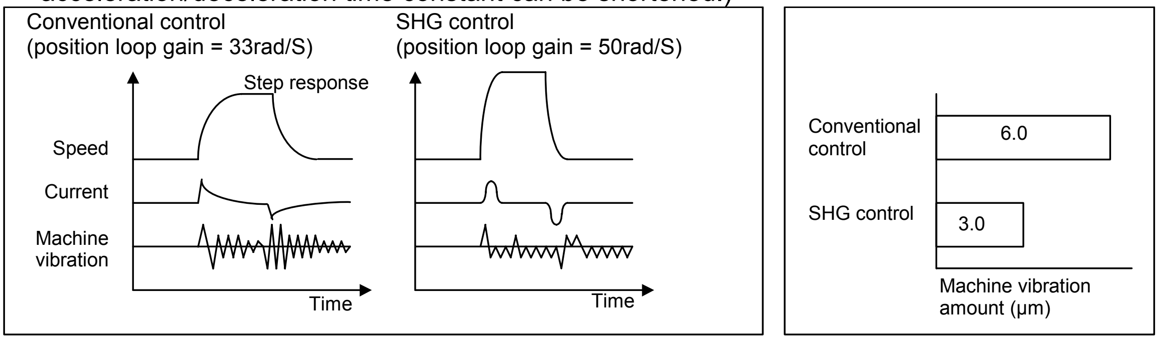
13.2 Dynamic Accuracy Compensation

13.2.1 Smooth High-gain Control (SHG Control)

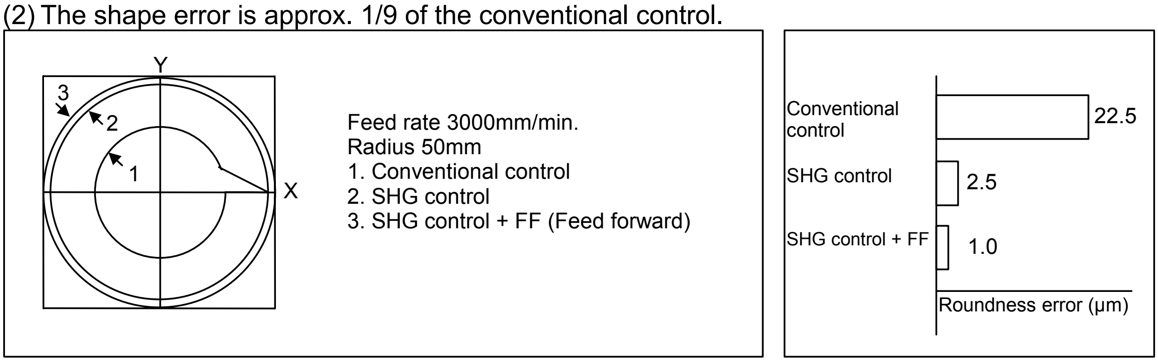
	E60	E68
M system	○	○
L system	○	○

This is a high-response and stable position control method using the servo system (MDS-□-V□/SVJ2). This SHG control realizes an approximately two-fold position loop gain equally compared to the conventional control method.
The features of the SHG control are as follows.

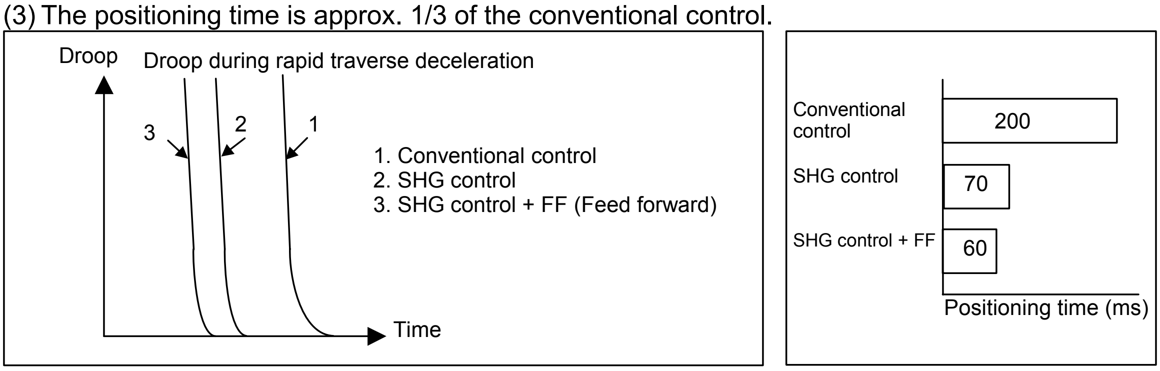
(1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/deceleration time constant can be shortened.)



(2) The shape error is approx. 1/9 of the conventional control.



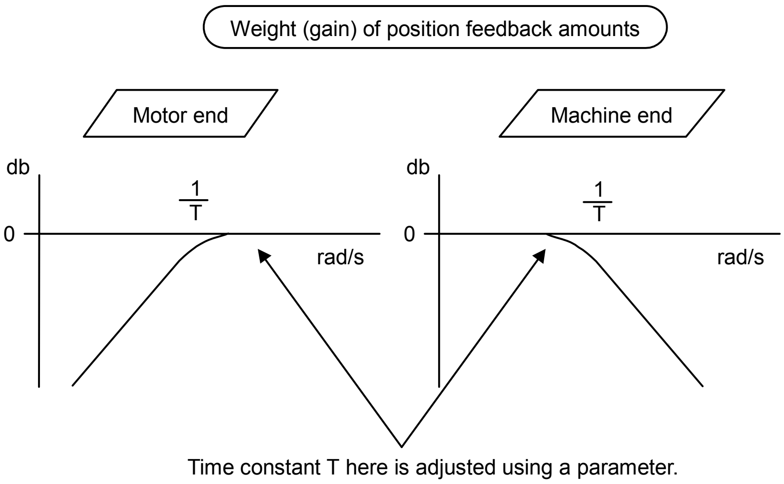
(3) The positioning time is approx. 1/3 of the conventional control.



13.2.2 Dual Feedback

	E60	E68
M system	–	Δ
L system	–	Δ

Depending on the frequency, the weight (gain) of the position feedback amount provided by the motor end detector and position feedback amount provided by the machine end detector stands in the correlation shown in the figure below. Semi-closed control is provided on a transient basis whereas positioning can be controlled by the closed status.
This function is used to select the primary delay filter time constant during dual feedback control as a parameter setting.



13.2.3 Lost Motion Compensation

	E60	E68
M system	○	○
L system	○	○

This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

14. Automation Support Functions

14.1 External Data Input

By using the DDB interface, the following functions can be realized from the PLC ladder.

14.1.1 External Search

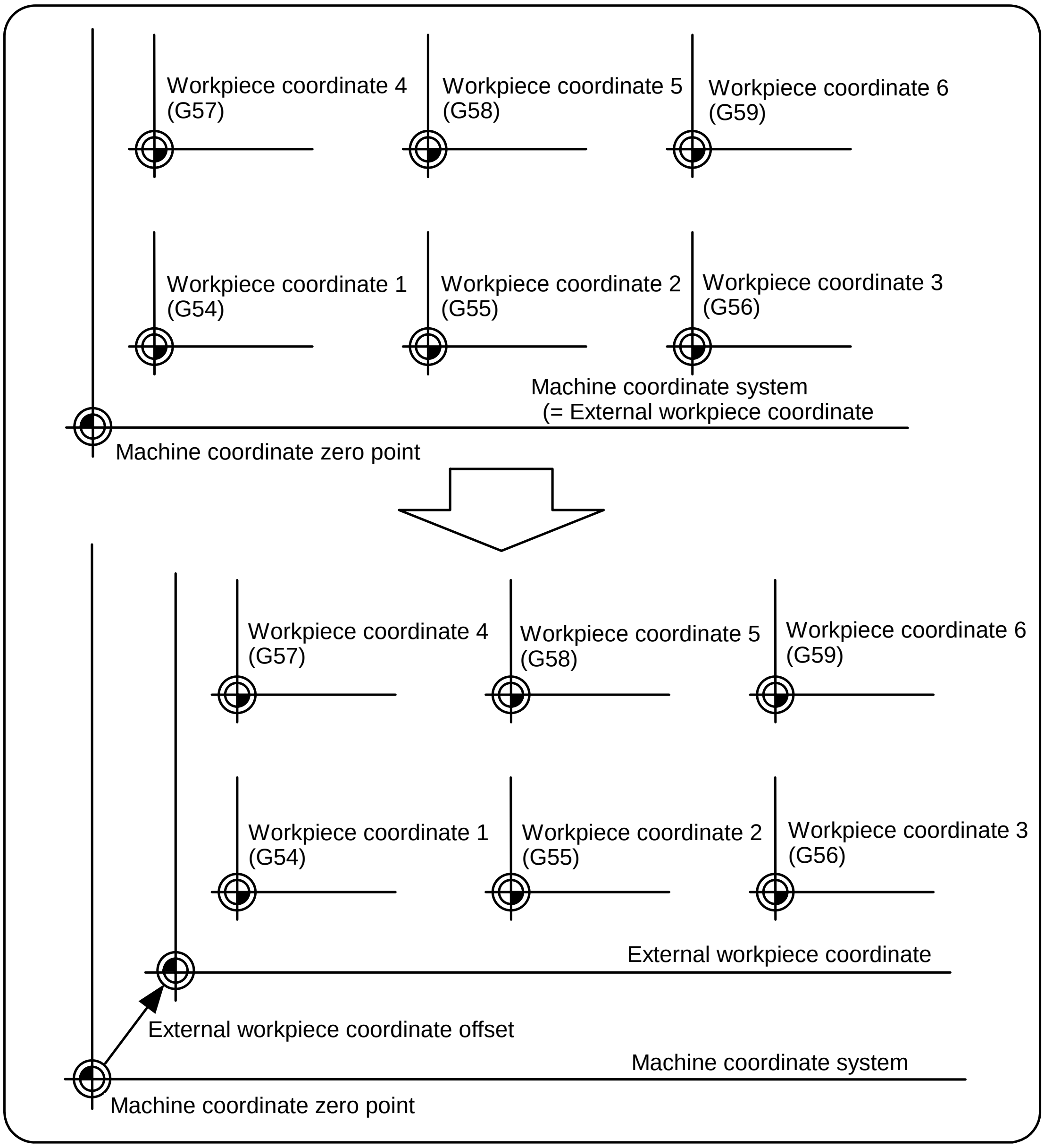
	E60	E68
M system	○	○
L system	○	○

The program No. and sequence No. to be automatically started in the memory or tape mode, ladder can be searched from the PLC ladder. The currently searched details can be read.

14.1.2 External Workpiece Coordinate Offset

	E60	E68
M system	○	○
L system	○	○

External workpiece coordinate offset that serves as the reference for all the workpiece coordinate systems is available outside the workpiece coordinates.
By setting the external workpiece coordinate offset, the external workpiece coordinate system can be shifted from the machine coordinate system, and all the workpiece coordinate systems can be simultaneously shifted by an amount equivalent to the offset.
When the external workpiece coordinate offset is zero, the external workpiece coordinate systems coincide with the machine coordinate system.
It is not possible to assign movement commands by selecting the external workpiece coordinates.



14.1.3 External Tool Offset

	E60	E68
M system	○	○
L system	○	○

The tool offset amount can be referred and updated from the PLC ladder.

14.2 Measurement

14.2.1 Skip

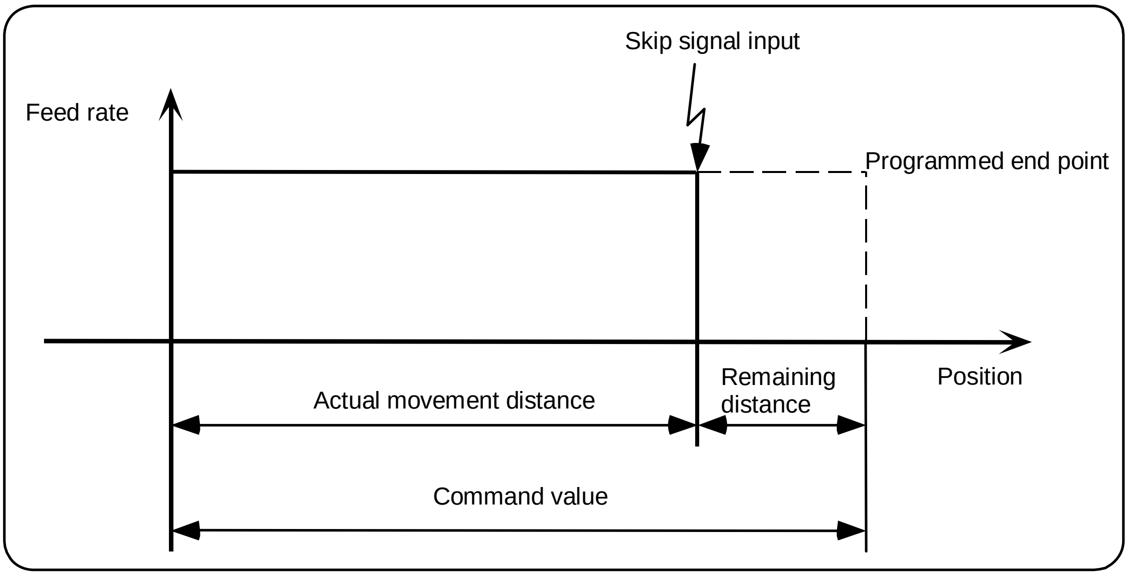
14.2.1.1 Skip

	E60	E68
M system	○	○
L system	○	○

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

G31 Xx1 Yy1 Zz1 Ff1 ;

M31	: Measurement command
Xx1, Yy1, Zz1	: Command values
Ff1	: Feed rate



When the G31 command is issued, acceleration/deceleration is accomplished in steps (time constant = 0).

There are two types of skip feed rate.
1. Feed rate based on program command when F command is present in program
2. Feed rate based on parameter setting when F command is not present in program

(Note 1) The approximate coasting distance up to feed stop based on the detection delay in the skip signal input is calculated as below.

$$\delta \fallingdotseq \frac{F}{60} \times (Tp + t)$$

δ : Coasting distance (mm)
F : G31 rate (mm/min)
Tp : Position loop time constant (s) = (position loop gain)$^{-1}$
T : Response delay time of 0.0035 (s)

(Note 2) Skipping during machine lock is not valid.

14.2.1.2 Multiple-step Skip

	E60	E68
M system	○	○
L system	○	○

(1) G31.n method

This function realizes skipping by designating a combination of skip signals for each skip command (G31.1, G31.2, G31.3).
The combination of the skip signals 1, 2 and 3 are designated with parameters for each G code (G31.1, 31.2, 31.3), and the skip operation is executed when all signals in the combination are input.

G31.n Xx1 Yy1 Zz1 Ff1 ;

M31.n	: Skip command (n=1, 2, 3)
Xx1, Yy1, Zz1	: Command format axis coordinate word and target coordinates
Ff1	: Feed rate (mm/min)

(2) G31Pn method

As with the G31.n method, the valid skip signal is designated and skip is executed. However, the method of designating the valid skip signal differs.
The skip signals that can be used are 1 to 4. Which is to be used is designated with P in the program. Refer to Table 1 for the relation of the P values and valid signals.
Skip can be executed on dwell, allowing the remaining dwell time to be canceled and the next block executed under the skip conditions (to distinguish external skip signals 1 to 4) set with the parameters during the dwell command (G04).

G31 Xx1 Yy1 Zz1 Pp1 Ff1 ;

G31	: Skip command
Xx1, Yy1, Zz1	: Command format axis coordinate word and target coordinates
Pp1	: Skip signal command
Ff1	: Feed rate (mm/min)

(a) Specify the skip rate in command feed rate F. However, F modal is not updated.
(b) Specify skip signal command in skip signal command P. Specify the P value in the range of 1 to 15. If it exceeds the specified range, a program error occurs.

Table 1 Valid skip signals

Skip signal command P	Valid skip signal			
	4	3	2	1
1				○
2			○	
3			○	○
4		○		
5		○		○
6		○	○	
7		○	○	○
8	○			
:	:	:	:	:
13	○	○		○
14	○	○	○	
15	○	○	○	○

14.2.1.4 PLC Skip

	E60	E68
M system	–	○
L system	–	○

This function enables skip operations to be performed by signals that are input from the user PLC.

14.2.5 Automatic Tool Length Measurement

	E60	E68
M system	○	○
L system	○	○

(1) Automatic Tool Length Measurement; G37 (M system)

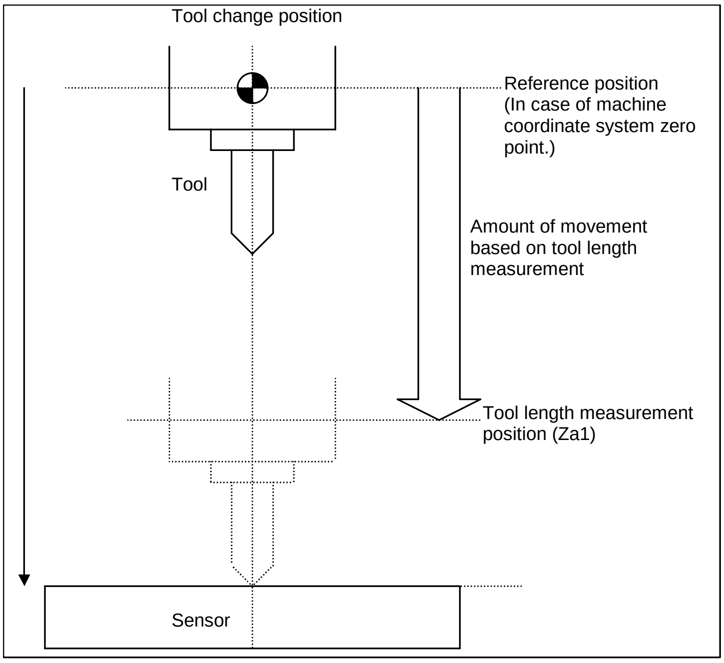
This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset amount for that tool.

If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current compensation amount is further corrected.
If the compensation amount at this time is one type, the compensation amount is automatically corrected; if there is a distinction between the tool length compensation amount and wear compensation amount, the wear amount is automatically corrected.

G37 Z_R_D_F_ ;

Z	:	Measurement axis address and measurement position coordinate. X, Z ...(NC axis)
R	:	The distance between the point at which tool movement is to start at the measurement speed and the measurement position.
D	:	The range in which the tool is to stop.
F	:	The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.

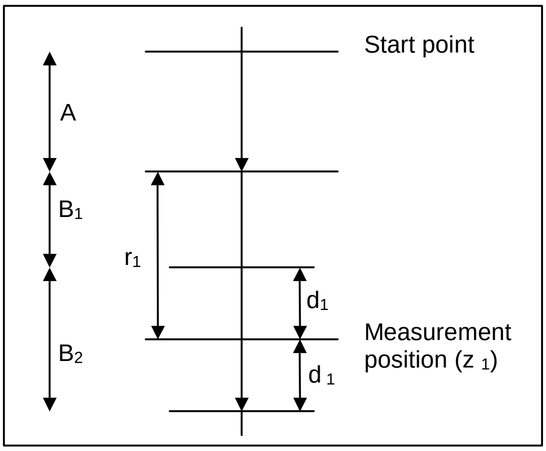


At this time, the tool length offset amount has a minus ("–") value.

Example of program

```
G28  Z0 ;
T01 ;
M06  T02 ;
G43  G00  Z0  H01 ;
G37  Z–300.  R10.D2.F10 ;
  .
  .
```

In this case, the distance (H01 = Za1 – z0) from the tool T01 tip to the top of the measurement sensor is calculated as the tool length offset amount that is then registered in the tool offset table.



Area A : Moves with rapid traverse feed rate.
Areas B_1, B_2 : Moves with the measurement speed (f_1 or parameter setting)

If a sensor signal is input in area B_1, an error will occur.
If a sensor signal is not input in the area B_2, an error will occur.

(2) Automatic tool length measurement (L series)

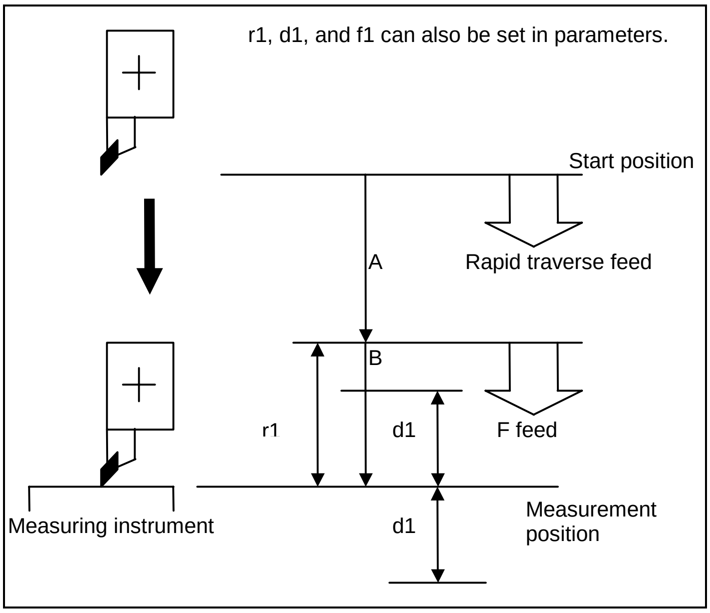
This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset amount for that tool.

If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current wear compensation amount is further corrected.

G37 α_R_D_F_ ;

α	:	Measurement axis address and measurement position coordinate. ... X, Z
R	:	The distance between the point at which tool movement is to start at the measurement speed and the measurement position. (Always a radial value: incremental value)
D	:	The range in which the tool is to stop. (Always a radial value: incremental value)
F	:	The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.



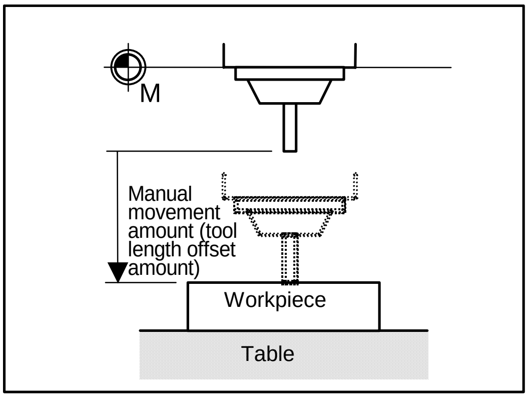
When the tool moves from the start position to the measurement position specified in G37 x1 (z1), it passes through the A area at rapid traverse. Then, it moves at the measurement rate set in F command or parameter from the position specified in r1. If the measurement position arrival signal turns ON during the tool is moving in the B area, an error occurs. If the measurement position arrival signal does not turn ON although the tool passes through the measurement position x1 (z1) and moves d1, an error occurs.

14.2.6 Manual Tool Length Measurement 1

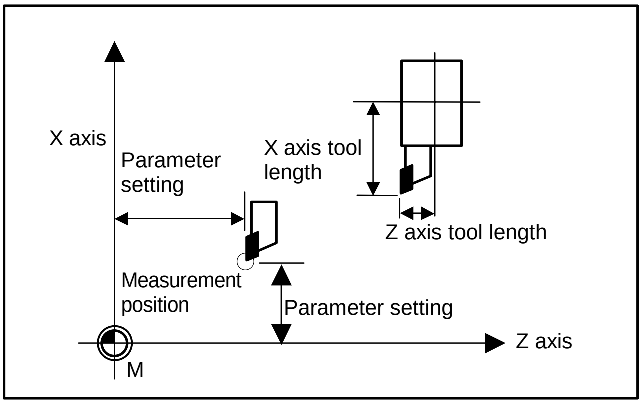
	E60	E68
M system	○	○
L system	○	○

Simple measurement of the tool length is done without a sensor.

(1) Manual tool length measurement 1 [M system]
When the tool is at the reference position, this function enables the distance from the tool tip to the measurement position (top of workpiece) to be measured and registered as the tool length offset amount.



(2) Manual tool length 1 measurement [L system]
A measurement position (machine coordinates) to match the tool nose on the machine is preset and the tool nose is set to the measurement position by manual feed, then the operation key is pressed, thereby automatically calculating the tool offset amount and setting it as the tool length offset amount.



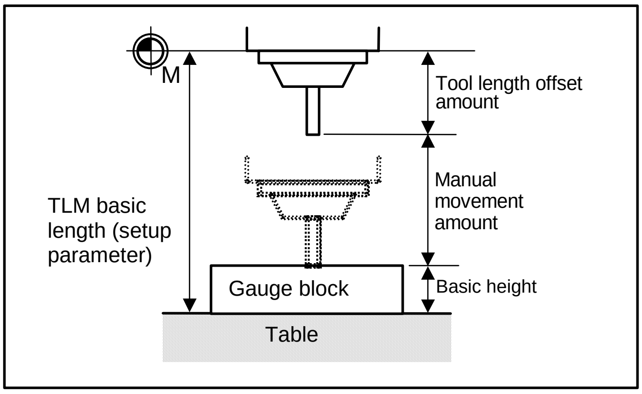
Measurement method

(a) Preset the machine coordinates of the measurement position in a given parameter as the measurement basic value.
(b) Select a tool whose tool length offset amount is to be measured.
(c) Set the tool nose to the measurement position by manual feed.
(d) Press the input key. The tool length offset amount is calculated and displayed on the setting area.
Tool length offset amount = machine coordinates – measurement basic value
(e) Again press the input key to store the value in the memory as the tool length offset amount of the tool.

14.2.7 Manual Tool Length Measurement 2

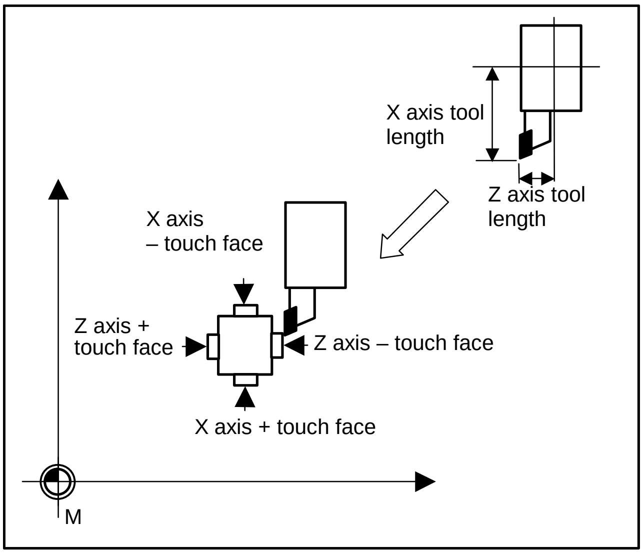
	E60	E68
M system	–	O
L system	O	O

(1) Manual tool length measurement 2 [M system]
When the tool is positioned at the reference position, this function enables the distance from the reference position to the tool tip to be measured and registered as the tool length offset amount. In this case, the position of the gauge block used as a reference must be set as the basic height.



If the height axis designation parameter is ON, the axis designated for plane selection basic axis K is the axis targeted for measurement as the height axis.
Furthermore, if the tool length measurement check parameter is ON, an input OK/cancel confirmation message appears after input key has been pressed.

(2) Manual tool length 2 measurement II [L system]
A device in which a touch sensor is built is used. Simply by causing the tool nose to touch the touch sensor in manual feed, the tool offset amount can be calculated and stored in tool offset amount memory.



Measurement method

(a) Preset the machine coordinates of the touch sensor touch face in parameter as the measurement basic value.
(b) Select the tool whose tool length offset amount is to be measured.
(c) Cause the tool nose to touch the touch sensor in manual feed.
The tool length offset amount is automatically calculated from the machine coordinates when the tool nose touches the touch sensor and the measurement basic value. It is stored in memory as tool length offset amount.

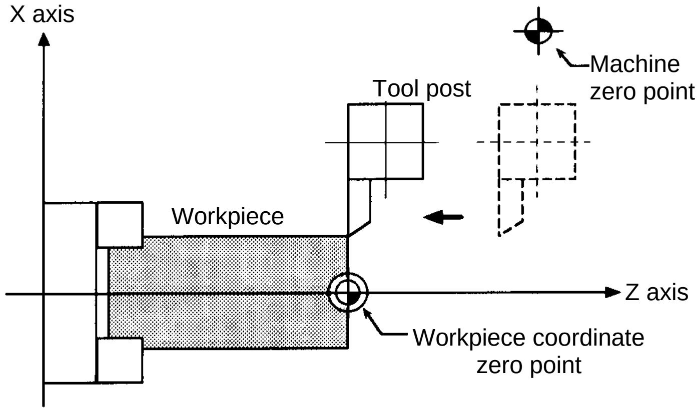
Tool length offset amount
= machine coordinates – measurement basic value (sensor position)

14.2.8 Workpiece Coordinate Offset Measurement

	E60	E68
M system	–	○(*)
L system	–	○

The external workpiece coordinate offset data for the Z axis can be set by cutting the workpiece face by means of manual operations and inputting the workpiece measurement signal.

Measurement of external workpiece coordinate offset data for Z axis



Setting method

(1) Select the tool, and cut the workpiece face.
(2) When the workpiece measurement signal is input, the external workpiece coordinate offset data for the Z axis is calculated from the machine coordinate values, length of the tool used and tool nose wear offset amount, and stored in the memory.

(Note) The (*) mark is the simple workpiece coordinate offset input.
With this function, the workpiece coordinate system offset data automatic calculation value and machine position for corresponding axes are displayed in the setting part on the workpiece coordinate offset screen.

14.2.9 Workpiece Position Measurement

	E60	E68
M system	–	○
L system	–	–

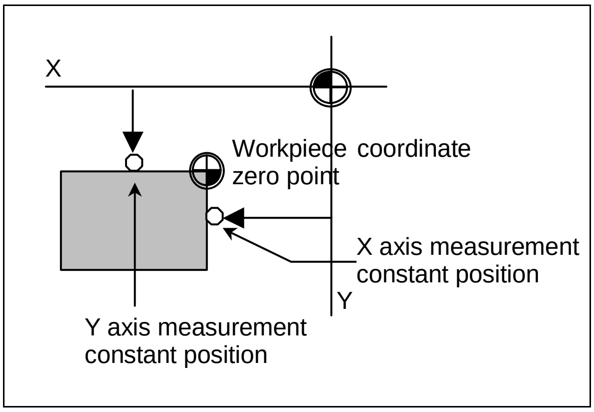
The workpiece position measurement function is used to measure each axis' coordinate point by installing a sensor on the spindle and the sensor contacting the workpiece with the manual feed or handle feed.
The surface, hole center and width center coordinates are calculated from the measured coordinates, and those calculated results are set in the workpiece coordinate offset.
Only 1st part system is available for the workpiece position measurement.

(1) Surface workpiece offset measurement

The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.
Measurement position coordinate X = X axis' skip machine position + Sensor diameter/2
+ Center compensation amount (Horizontal)
Measurement position coordinate Y = Y axis' skip machine position + Sensor diameter/2
+ Center compensation amount (Vertical)
Measurement position coordinate Z = Z axis' skip machine position - Sensor length

The sensor diameter/2 changes between +/- with the last tool movement direction during the measurement.
The sensor diameter and center compensation amount are applied to the X axis or Y axis.
The sensor length is applied to the Z axis.

The measurement position coordinate of the X axis, Y axis or Z axis is set in the specified workpiece coordinate offset.



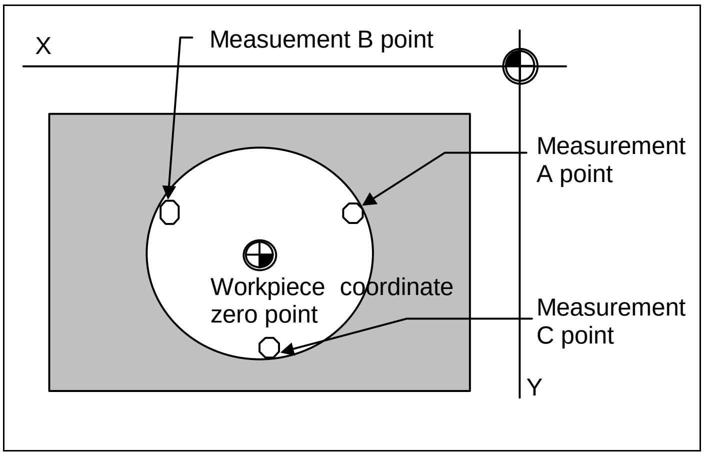
To set the workpiece coordinate offset, the X axis is measured and the X axis' offset coordinate is set. Then, the Y axis' offset is measured and set. Finally, the Z axis' offset is measured and set.

(2) Hole center workpiece offset measurement

The measurement position coordinates of two axes (X, Y) are measured at three points, and the hole center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.

Measurement position coordinate X = X axis' skip machine position + Center compensation amount (Horizontal)

Measurement position coordinate Y = Y axis' skip machine position + Center compensation amount (Vertical)



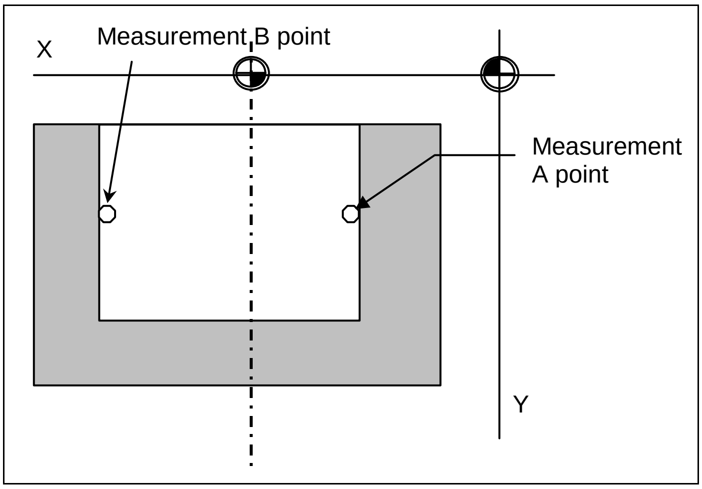
To set the workpiece coordinate offset, the position X and Y of the measurement A point are measured, and the measured values are set in the measurement A point. In the same manner as the measurement A point, the measurement B point and then C point are measured and set.
The hole center coordinate is calculated by setting the workpiece coordinate system after setting three points, and the calculated result is set in the workpiece coordinate offset.

(3) Width center workpiece offset measurement

The measurement position coordinate of the X axis or Y axis is measured at two points, and each axis' groove center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.

Measurement position coordinate X = X axis' skip machine position + Center compensation amount (Horizontal)

Measurement position coordinate Y = Y axis' skip machine position + Center compensation amount (Vertical)



To set the workpiece coordinate offset, the position X (position Y) of the measurement A point is measured, and the measured value is set in the measurement A point. In the same manner as the measurement A point, the measurement B point is measured and set.
The groove width center coordinate of the X axis (Y axis) is calculated by setting the workpiece coordinate system after setting two points, and the calculated result is set in the workpiece coordinate offset.

14.3 Monitoring

14.3.1 Tool Life Management

14.3.1.1 Tool Life Management I

	E60	E68
M system	○	○
L system	○	○

(1) M series

The time (0 to 4000 hours) and frequency (0 to 65000 times) of use of the user PLC specified tool are accumulated. Tool data including the time and frequency of use of the PLC specified tool is output.

(2) L series

Tool life management is performed using the time and frequency of use of a tool.

(a) Management by the time of use

The cutting time after specification of a tool selection (T) command (G01, G02, and G33) is added to the tool use time for the specified tool.
If the use time reaches the life time when a tool selection command is specified, an alarm is given.

(b) Management by the frequency of use

The tool use counter corresponding to the number of the specified tool is incremented each time a tool selection (T) command is specified for the tool.
If the counter reaches the life time when a tool selection command is specified, an alarm is given.

14.3.1.2 Tool Life Management II

	E60	E68
M system	○	○
L system	○	○

(1) M series

A spare tool change function is added to tool life management I. This function selects a usable tool out of the spare tools of the group determined by the value specified by the user PLC, then outputs data of such usable spare tool. The spare tool can be selected in two ways: the tools are selected in order they were registered in the group or the tool whose remaining life is the longest of all in the group is selected.

(2) L series

The life of each tool (time and frequency) is controlled, and when the life is reached, a spare tool that is the same type is selected from the group where the tool belongs and used.
• No. of groups: 80 sets
• No. of tools in group: Max. 16 tools

14.3.2 Number of Tool Life Management Sets

20/40/80 sets

	E60	E68
M system	–	–
L system	○80	○80

100/200 sets

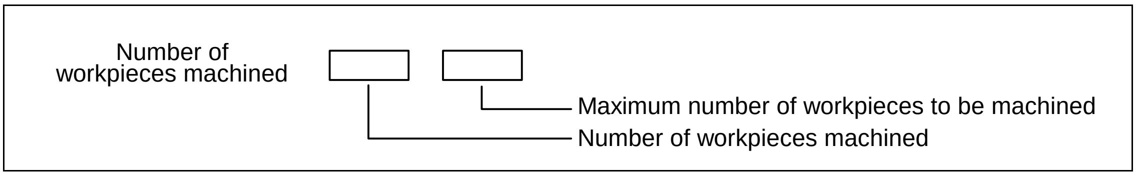
	E60	E68
M system	○100	○200
L system	–	–

14.3.3 Number of Parts

	E60	E68
M system	○	○
L system	○	○

Part count display

The number of machined parts is counted up each time a part is machined, and displayed .



14.3.4 Load Meter

	E60	E68
M system	○	○
L system	○	○

Using the user PLC, this function displays the spindle load, Z-axis load, etc. in the form of bar graphs.

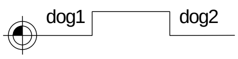
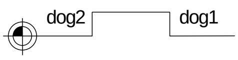
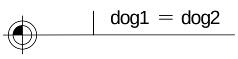
14.3.5 Position Switch

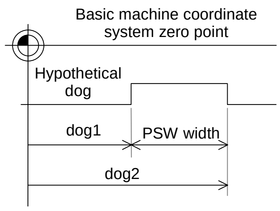
	E60	E68
M system	○24	○24
L system	○24	○24

The position switch (PSW) function provides hypothetical dog switches in place of the dog switches provided on the machine axes by setting the axis names and coordinates indicating the hypothetical dog positions as parameters beforehand so that signals are output to the PLC interface when the machine has reached these hypothetical dog positions. The hypothetical dog switches are known as position switches (PSW).

The coordinates indicating the hypothetical dog positions (dog1, dog2) on the coordinate axes whose names were set by parameters ahead of time in place of the dog switches provided on the machine axes are set using position switches. When the machine has reached the hypothetical dog positions, a signal is output to the device supported by the PLC interface.

Example of dog1, dog2 settings and execution

<table>
<tr><th>dog1, dog2 settings</th><th>dog1, dog2 positions</th><th>Description</th></tr>
<tr><td>dog1 < dog2</td><td>
</td><td>Signal is output between dog1 and dog2</td></tr>
<tr><td>dog1 > dog2</td><td>
</td><td>Signal is output between dog2 and dog1</td></tr>
<tr><td>dog1 = dog2</td><td>
</td><td>Signal is output at the dog1 (dog2) position</td></tr>
</table>



14.3.12 Synchronous Error Observation

	E60	E68
M system	–	○
L system	–	○

With this function, synchronous tapping error (screw pitch error) generated during synchronous tapping cycle operation is monitored and a warning is output when the error exceeds a certain value.

14.5 Others

14.5.1 Programmable Current Limitation

	E60	E68
M system	－	O
L system	－	O

This function allows the current limit value of the servo axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc.
The commanded current limit value is designated with a ratio of the limit current to the rated current.
The current limit value can also be set from the D.D.B. function and setting and display unit.
The validity of the current limit can be selected with the external signal input.
However, the current limit value of the PLC axis cannot be rewritten.

G10 L14 X dn ;

L14	: Current limit value setting (+ side/– side)
X	: Axis address
dn	: Current limit value 1% to 300%

(1) If the current limit is reached when the current limit is valid, the current limit reached signal is output.
(2) The following two modes can be used with external signals as the operation after the current limit is reached.
- Normal mode
  The movement command is executed in the current state.
  During automatic operation, the movement command is executed to the end, and then the next block is moved to with the droops still accumulated.
- Interlock mode
  The movement command is blocked (internal interlock).
  During automatic operation, the operation stops at the corresponding block, and the next block is not moved to.
  During manual operation, the following same direction commands are ignored.

(3) During the current limit, the droop generated by the current limit can be canceled with external signals.
(Note that the axis must not be moving.)
(4) The setting range of the current limit value is 1% to 300%. Commands that exceed this range will cause a program error.
"P35 CMD VALUE OVER" will be displayed.
(5) If a decimal point is designated with the G10 command, only the integer will be valid.
(Example) G10 L14 X10.123 ; The current limit value will be set to 10%.
(6) For the axis name "C", the current limit value cannot be set from the program (G10 command).
To set from the program, set the axis address with an incremental axis name, or set the axis name to one other than "C".

15. Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

	E60	E68
M system	○	○
L system	○	○

All operations are stopped by the emergency stop signal input and, at the same time, the drive section is stopped using the dynamic brake and the movement of the machine is stopped.
At this time, the READY lamp goes OFF and the servo ready signal is turned OFF.

15.1.2 Data Protection Key

	E60	E68
M system	○	○
L system	○	○

With the input from the user PLC, it is possible to prohibit the setting and deletion of parameters and the editing of programs from the screen.
Data protection is divided into the following groups.

Group 1: For protecting the tool data and protecting the coordinate system presettings as based on origin setting (zero)
Group 2: For protecting the user parameters and common variables
Group 3: For protecting the machining programs

15.2 Display for Ensuring Safety

15.2.1 NC Warning

	E60	E68
M system	○	○
L system	○	○

The warnings that are output by the NC system are listed below.
When one of these warnings has occurred, a warning number is output to the PLC and a description of the warning appears on the screen. Operation can be continued without taking further action.

Type of warning	Description
Servo warning	The servo warning is displayed.
Spindle warning	The spindle warning is displayed.
System warning	The system warning is displayed. (State such as temperature rise, battery voltage low, etc.)
Absolute position warning	A warning in the absolute position detection system is displayed.
Auxiliary axis warning	The auxiliary axis warning is displayed.

15.2.2 NC Alarm

	E60	E68
M system	○	○
L system	○	○

The alarms that are output by the NC system are listed below. When one of these alarms has occurred, an alarm number is output to the PLC, and a description of the alarm appears on the screen. Operation cannot be continued without taking remedial action.

Type of warning	Description
Operation alarm	This alarm occurring due to incorrect operation by the operator during NC operation and that by machine trouble are displayed.
Servo alarm	This alarm describes errors in the servo system such as the servo drive unit, motor and encoder.
Spindle alarm	This alarm describes errors in the spindle system such as the spindle drive unit, motor and encoder.
MCP alarm	An error has occurred in the drive unit and other interfaces.
System alarm	This alarm is displayed with the register at the time when the error occurred on the screen if the system stops due to a system error.
Absolute position detection system alarm	An alarm in the absolute position detection system is displayed.
Auxiliary axis alarm	The auxiliary axis alarm is displayed.
Computer link error	The computer link alarm is displayed.
User PLC alarm	The user PLC alarm is displayed.
Program error	This alarm occur during automatic operation, and the cause of this alarm is mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

15.2.3 Operation Stop Cause

	E60	E68
M system	○	○
L system	○	○

The stop cause of automatic operation is displayed on the screen.

15.2.4 Emergency Stop Cause

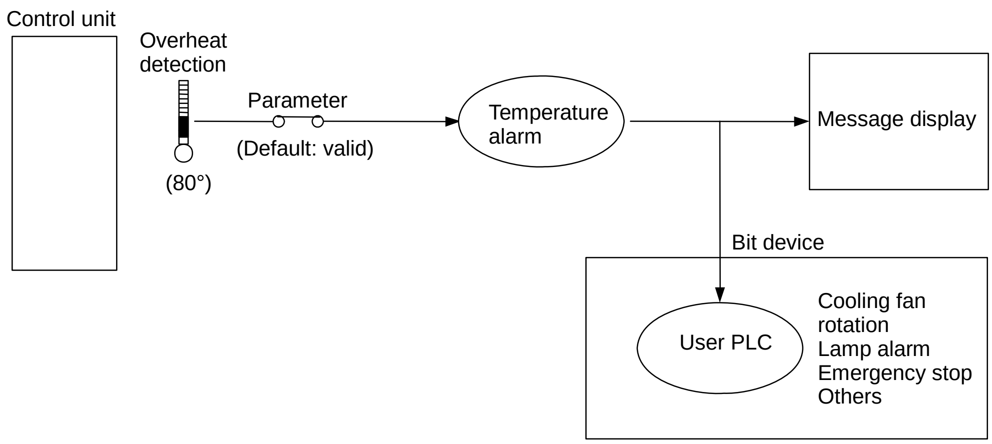
	E60	E68
M system	○	○
L system	○	○

When "EMG" (emergency stop) message is displayed in the operation status display area of the screen, the emergency stop cause can be confirmed.

15.2.5 Temperature Detection

	E60	E68
M system	○	○
L system	○	○

When overheating is detected in the control unit, an overheat signal is output at the same time as the alarm is displayed. If the system is in automatic run at the time, run is continued, but it cannot be started after reset or completion by M02/M30. (It can be started after block stop or feed hold.)
When the temperature falls below the specified temperature, the alarm is released and the overheat signal is turned OFF.
The overheat alarm occurs at 80°C or more in the control unit.



(Note 1) If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

15.3 Protection

15.3.1 Stroke End (Over travel)

	E60	E68
M system	○	○
L system	○	○

When limit switches and dogs have been attached to the machine and a limit switch has kicked a dog, the movement of the machine is stopped by the signal input from the limit switch.
At the same time, the alarm output is sent to the machine.
The stroke end state is maintained and the alarm state is released by feeding the machine in the reverse direction in the manual mode to disengage the dog.

15.3.2 Stored Stroke Limit

The stored stroke limits I, II, IIB, IB and IC are handled as follows.

<table>
<tr><th>Type</th><th>Prohibited range</th><th>Explanation</th></tr>
<tr><td>I</td><td>Outside</td><td>•Set by the machine maker.
•When used with II, the narrow range designated by the two types becomes the movement valid range.</td></tr>
<tr><td>II</td><td>Outside</td><td rowspan="2">•Set by the user.
•The change or function of parameter can be turned OFF/ON with the program command.
•Select II or IIB with the parameters.</td></tr>
<tr><td>IIB</td><td>Inside</td></tr>
<tr><td>IB</td><td>Inside</td><td>•Set by the machine maker.</td></tr>
<tr><td>IC</td><td>Outside</td><td>•Set by the machine maker.
•Can be rewritten with DDB.</td></tr>
</table>

15.3.2.1 Stored Stroke Limit I/II

	E60	E68
M system	○	○
L system	○	○

(1) Stored Stroke Limit I

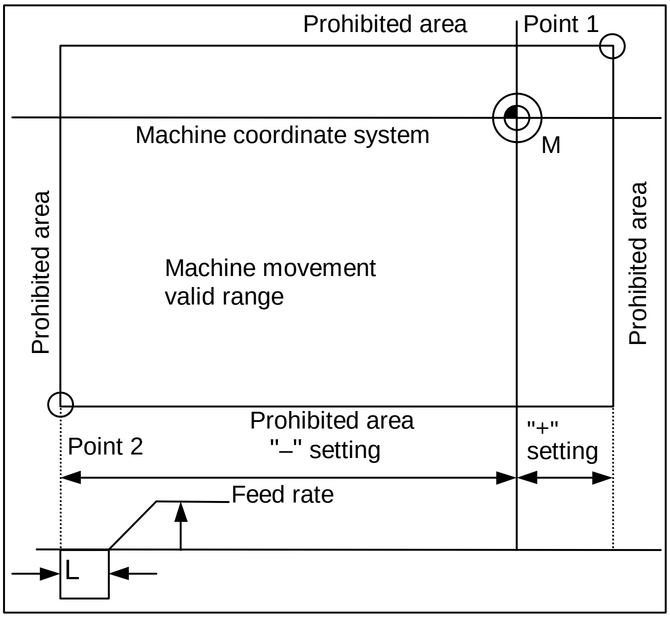
This is the stroke limit function used by the machine maker, and the area outside the set limits is the entrance prohibited area.

The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit II function described in the following section, and the tolerable area of both functions is the movement valid range.

The setting range is −99999.999 to +99999.999mm.

The stored stroke limit I function is made valid not immediately after the controller power is turned ON but after reference position return.

The stored stroke limit I function will be invalidated if the maximum and minimum values are set to the same data.



The values of points 1 and 2 are set using the coordinate values in the machine coordinate system.

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.

The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

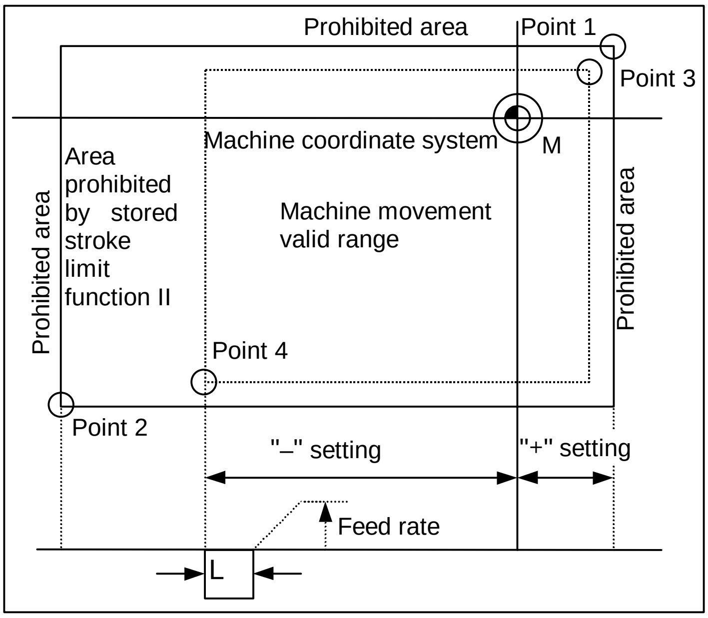
(2) Stored Stroke Limit II

This is the stroke limit function that can be set by the user, and the area outside the set limits is the prohibited area.

The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit I function described in the foregoing section, and the tolerable area of both functions is the movement valid range.

The setting range is –99999.999 to +99999.999mm.

The stored stroke limit II function will be invalidated if the maximum and minimum parameter values are set to the same data.



The values of points 3 and 4 are set with the coordinate values in the machine coordinate system.

The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.

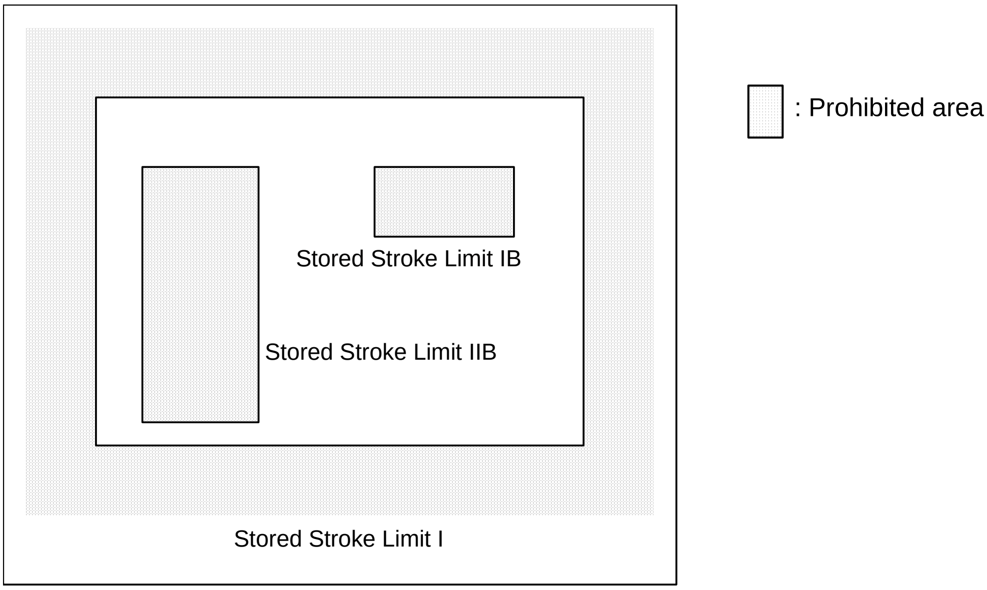
The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

The stored stroke limit II function can also be invalidated with the parameter settings.

15.3.2.2 Stored Stroke Limit IB

	E60	E68
M system	–	○
L system	–	○

Three areas where tool entry is prohibited can be set using the stored stroke limit I, stored stroke limit II, IIB and stored stroke limit IB functions.



When an attempt is made to move the tool beyond the set range, an alarm is displayed, and the tool decelerates and stops. If the tool has entered into the prohibited area and an alarm has occurred, it is possible to move the tool only in the opposite direction to the direction in which the tool has just moved.

(Note 1) Bear in mind that the following will occur if the same data is set for the maximum and minimum value of the tool entry prohibited area:

(1) When zero has been set for the maximum and minimum values, tool entry will be prohibited in the whole area.
(2) If a value other than zero has been set for both the maximum and minimum values, it will be possible for the tool to move in the whole area.

15.3.2.3 Stored Stroke Limit IIB

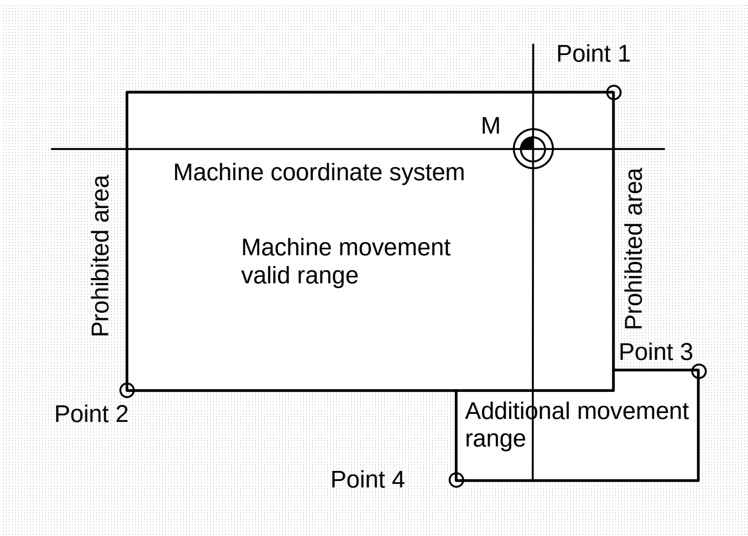
	E60	E68
M system	–	○
L system	–	○

A parameter is used to switch between this function and stored stroke limit II. With stored stroke limit IIB, the range inside the boundaries which have been set serves as the tool entry prohibited area.

15.3.2.4 Stored Stroke Limit IC

	E60	E68
M system	–	○
L system	–	○

The boundary is set for each axis with the parameters. The inside of the set boundary is the additional movement range.
This cannot be used with stored stroke limit IB.



The position of points 3 and 4 are set with the machine coordinate.

The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

▭ : Prohibited area

15.3.4 Chuck/Tailstock Barrier Check

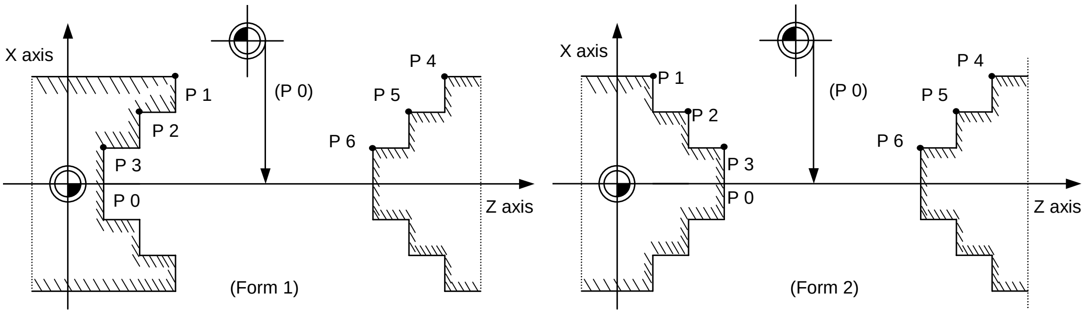
	E60	E68
M system	–	–
L system	○	○

By limiting the tool nose point move range, this function prevents the tool from colliding with the chuck or tail stock because of a programming error.
When a move command exceeding the area set in a given parameter is programmed, the tool is stopped at the barrier boundaries.

Program format

G22 ; ..... Barrier ON
G23 ; ..... Barrier OFF (cancel)

(1) When the machine is about to exceed the area, the machine is stopped and an alarm is displayed. To cancel the alarm, execute reset.
(2) The function is also effective when the machine is locked.
(3) This function is valid when all axes for which a barrier has been set have completed reference position return.
(4) Chuck barrier/tail stock barrier setting



The chuck barrier and tail stock barrier are both set with the machine coordinate by inputting one set of three-point data in the parameter. Points P1, P2 and P3 are the chuck barrier, and points P4, P5 and P6 are the tail stock barrier. The X axis is set with the coordinate value (radius value) from the workpiece center, and the Z axis is set with the basic machine coordinate system coordinate.
Point P0 is the chuck barrier and tail stock barrier's basic X coordinates, and the workpiece center coordinate in the basic machine coordinate system is set.
The barrier area is assumed to be symmetrical for the Z axis, and if the X axis coordinate of barrier point P_ is minus, the sign is inverted to plus and the coordinate is converted for a check.
Set the absolute values of the X axis coordinates of the barrier points as shown below:

P1 >= P2 >= P3, P4 >= P5 >= P6

(However, this need not apply to the Z axis coordinates.)

15.3.5 Interlock

	E60	E68
M system	○	○
L system	○	○

The machine movement will decelerate and stop as soon as the interlock signal, serving as the external input, is turned ON.
When the interlock signal is turned OFF, the machine starts moving again.

(1) In the manual mode, only that axis for which the interlock signal is input will stop.
(2) In the automatic mode, all axes will stop when the interlock signal is input to even one axis which coincides with the moving axis.
(3) Block start interlock
While the block start interlock signal (*BSL) is OFF (valid), the execution of the next block during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

(Note 1) This signal is valid for all blocks including internal operation blocks such as fixed cycles.
(Note 2) This signal (*BSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

(4) Cutting start interlock
While the cutting start interlock signal (*CSL) is OFF (valid), the execution of all movement command blocks except positioning during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.

(Note 1) The signal is valid for all blocks including internal operation block such as fixed cycles.
(Note 2) This signal (*CSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

15.3.6 External Deceleration

	E60	E68
M system	○	○
L system	○	○

This function reduces the feed rate to the deceleration speed set by the parameter when the external deceleration input signal, which is the external input from the user PLC, has been set to ON. External deceleration input signals are provided for each axis and for each movement direction ("+" and "-"), and a signal is valid when the signal in the direction coinciding with the direction of the current movement has been input. When an axis is to be returned in the opposite direction, its speed is returned immediately to the regular speed assigned by the command.
When non-interpolation positioning is performed during manual operation or automatic operation, only the axis for which the signal that coincides with the direction of the current movement has been input will decelerate.
However, with interpolation during automatic operation, the feed rate of the axis will be reduced to the deceleration rate if there is even one axis for which the signal that coincides with the direction of current movement has been input.

The external deceleration input signal can be canceled using a parameter for the cutting feed only.

15.3.8 Door Interlock

15.3.8.1 Door Interlock I

	E60	E68
M system	O	O
L system	O	O

Outline of function

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.
When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.
When the door open signal has been input during automatic operation, the suspended machining can be resumed by first closing the door concerned and then initiating cycle start again.

Description of operation

When a door is open

The NC system operates as follows when the door open signal is input:

(1) It stops operations.
 1. When automatic operation was underway
 The machine is set to the feed hold mode, and all the axes decelerate and stop.
 The spindle also stops.
 2. When manual operation was underway
 All the axes decelerate and stop immediately.
 The spindle also stops.

(2) The complete standby status is established.

(3) After all the servo axes and the spindle have stopped, the ready OFF status is established.

(4) The door open enable signal is output.
 Release the door lock using this signals at the PLC.

When a door is closed

After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the door open signal is set to OFF.

(5) All the axes are set to ready ON.

(6) The door open enable signal is set to OFF.

Resuming operation

(7) When automatic operation was underway
 Press the AUTO START button.
 Operation now resumes from the block in which machining was suspended when the door open signal was input.

(8) When manual operation was underway
 Axis movement is commenced when the axis movement signals are input again.

(9) Spindle rotation
 Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again: this can be done either by operations performed by the operator or by using the user PLC.

15.3.8.2 Door Interlock II

	E60	E68
M system	○	○
L system	○	○

Outline of function

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.

When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.

With the door interlock function established by the door open II signal, automatic start can be enabled even when the door open signal has been input. However, the axes will be set to the interlock status.

Description of operation

When a door is open

The NC system operates as follows when the door open II signal is input:

(1) It stops operations.
All the axes decelerate and stop.
The spindle also stops.

(2) The complete standby status is established.

(3) After all the servo axes and the spindle have stopped, the ready OFF status is established. However, the servo ready finish signal (SA) is not set to OFF.

When a door is closed

After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the door open signal is set to OFF.

(4) All the axes are set to ready ON.

(5) The door open enable signal is set to OFF.

Resuming operation

(6) When automatic operation was underway
The door open signal is set to OFF, and after the ready ON status has been established for all the axes, operation is resumed.

(7) When manual operation was underway
Axis movement is commenced when the axis movement signals are input again.

(8) Spindle rotation
Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again: this can be done either by operations performed by the operator or by using the user PLC.

(Note) Concerning the handling of an analog spindle

The signals described in this section are valid in a system with bus connections for the NC control unit and drive units. When an analog spindle is connected, the NC system cannot verify that the spindle has come to a complete stop. This means that the door should be opened after the PLC has verified that the spindle has come to a complete stop. Since the spindle may resume its rotation immediately after the door has been closed, set the forward and reverse rotation signals to OFF when opening the door so as to ensure safety.

Appendix 1. Differences from door interlock I

(1) The method used to stop the machine during automatic operation is the same as with the axis interlock function.
(2) The servo ready finish signal (SE) is not set to OFF.
(3) Automatic start is valid during door interlock. However, the interlock takes effect for the axis movements.
(4) When this door interlock function (door open signal ON) is initiated during axis movement, the axes decelerate and stop.
(5) When this door interlock function (door open signal) is set to OFF, the axis movement resumes.

15.3.9 Parameter Lock

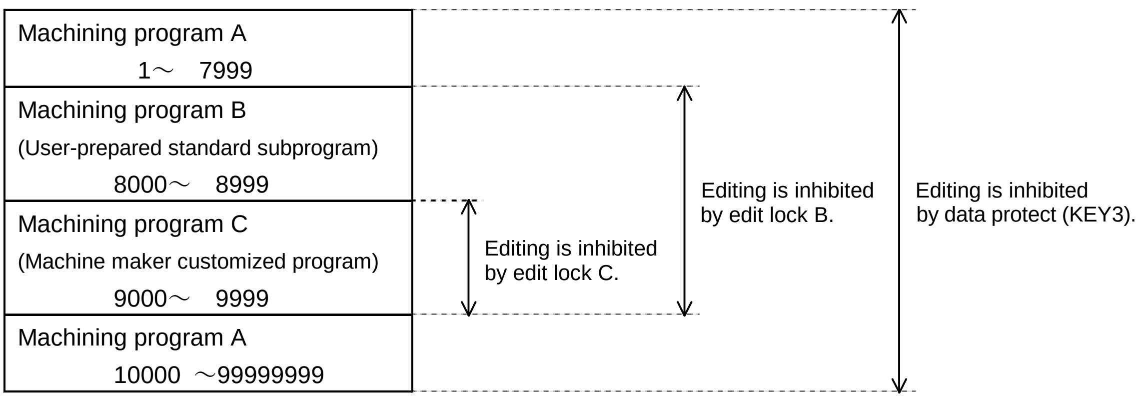
	E60	E68
M system	○	○
L system	○	○

This function is used to prohibit changing the setup parameter.

15.3.10 Program Protect (Edit Lock B, C)

	E60	E68
M system	○	○
L system	○	○

The edit lock function B or C inhibits machining program B or C (group with numbers) from being edited or erased when these programs require to be protected.



15.3.11 Program Display Lock

	E60	E68
M system	○	○
L system	○	○

This function allows the display of only a target program (label address 9000) to be invalidated for the program display in the monitor screen, etc.
The operation search of a target program can also be invalidated.
The validity of the display is selected with the parameters. The setting will be handled as follows according to the value.

0: Display and search are possible.
1: Display of the program details is prohibited.
2: Display and operation search of the program details are prohibited.

The program details are not displayed in the prohibited state, but the program number and sequence number will be displayed.

15.4 Maintenance and Troubleshooting

15.4.1 History Diagnosis

	E60	E68
M system	○	○
L system	○	○

This is a maintenance function which is useful for tracing down the history and NC operation information and analyzing trouble, etc. This information can be output as screen displays or as files.

(1) Screen display showing operation history and event occurrence times
The times/dates (year/month/day and hour/minute/second) and messages are displayed as the operation history data. The key histories, alarm histories and input/output signal change histories are displayed as the messages.

(2) Outputting the data in the operation history memory
Information on the alarms occurring during NC operation and stop codes, signal information on the changes in the PLC interface input signals and the key histories can be output through the RS-232C interface.

15.4.2 Setup / Monitor for Servo and Spindle

	E60	E68
M system	○	○
L system	○	○

The information on the servos (NC axes), spindles, PLC axes and power supplies appears on the screen.
Main information displayed on the monitor:
Position loop tracking deviation, motor speeds, load current, detector feedback, absolute position detection information, drive unit alarm histories, operation times, drive unit software versions, etc.

15.4.3 Data Sampling

	E60	E68
M system	○	○
L system	○	○

- Sampling of the servo and spindle data for which an alarm occurrence is a stop condition is performed all the time. By using the waveform display function, this sampling data can be displayed in the waveforms.
- The data currently displayed can be stored on a memory card, and read out when required.

15.4.4 Waveform Display

	E60	E68
M system	○	○
L system	○	○

The following servo data and spindle data can be displayed as waveforms. Data can be displayed for two channels simultaneously in 1-hour increments on a continuous basis or on a one-shot basis.

Servo data	Spindle data
Current feedback Current command Position deviation Position command Speed feedback	Motor load Motor rotation speed (speed command value) Position deviation Position command Speed feedback (r/min) Synchronous tap error width (µm) Synchronous tap error angle (0.001 degree)

15.4.5 Machine Operation History Monitor

	E60	E68
M system	○	○
L system	○	○

Up to 256 past key inputs on the operation board and changes in the input signals are recorded. The history contents can be viewed on the history screen, and the data is retained even after the power has been turned OFF.

15.4.6 NC Data Backup

With this function, the parameters and other data of the NC control unit can be backed up in the memory cassette.
The data can also be restored.

(1) RS-232C

	E60	E68
M system	○	○
L system	○	○

[Backup target]

Machining programs, parameters, workpiece offset data, common variables, tool compensation data, tool life control data
Ladders (ladder, message)
SRAM data

(2) Cassette memory

	E60	E68
M system	○	○
L system	○	○

The memory cassette for maintenance is used to back up and restore the NC data.

Model	Compatible memory cassette for maintenance
E60/E68	HR410/HR450

[Backup targets ... The following data is backed up in a batch.**]**

Ladders (ladder, message)
SRAM data

(3) IC card

	E60	E68
M system	–	○
L system	–	○

[Backup target]

Machining programs, parameters, common variables, tool compensation data, tool life control data
Ladders (ladder, message)
SRAM data

15.4.7 PLC I/F Diagnosis

	E60	E68
M system	○	○
L system	○	○

When the **I/F DIAGN** menu key is pressed, the PLC interface diagnosis screen appears.
The input and output signals for PLC control can be displayed and set on this screen.
This function can be used to check the machine sequence operations for PLC development, check the input/output data between the control unit and PLC when trouble occurs in operation, initiate forced definitions, and so on.

15.4.13 Signal Trace Function

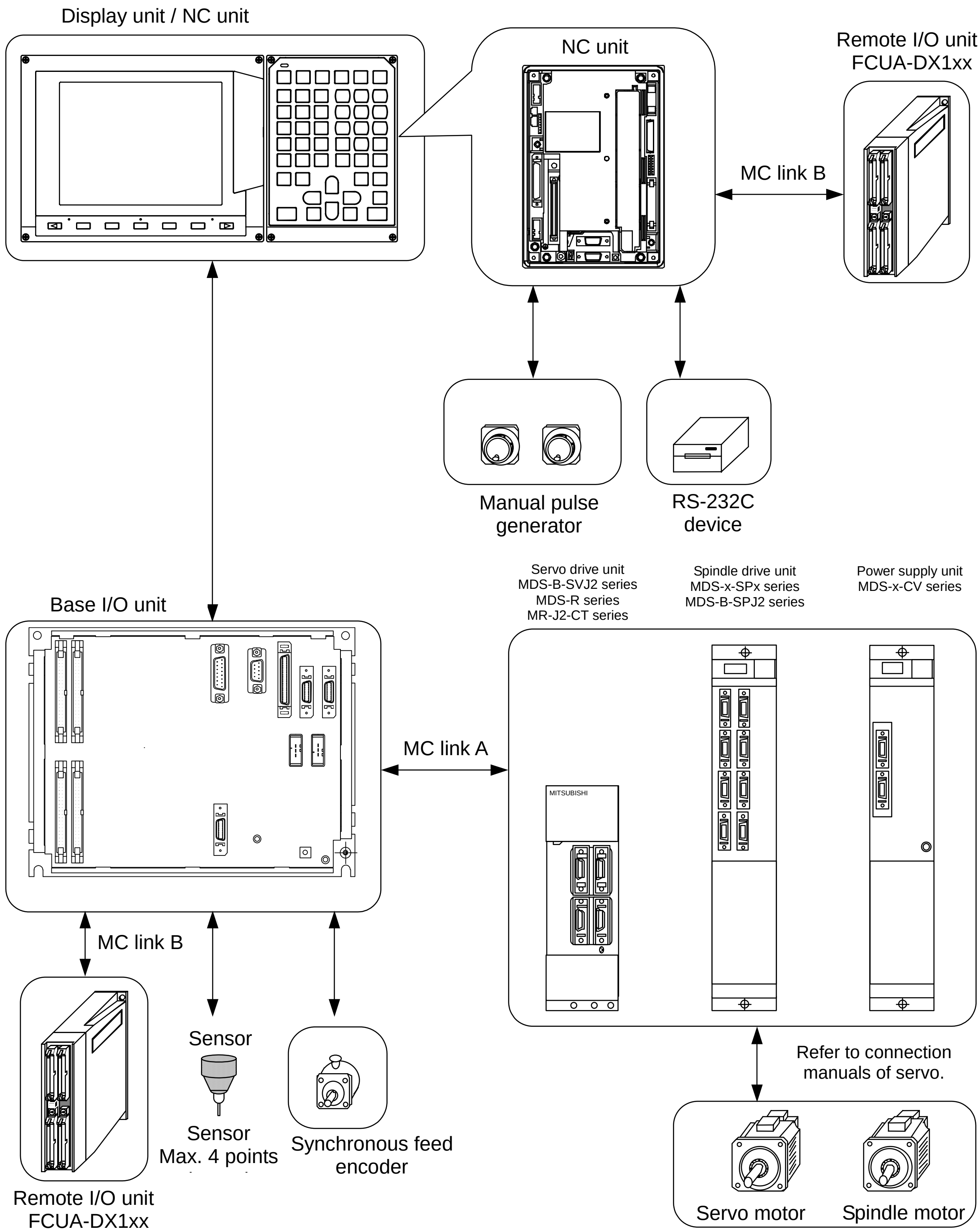
	E60	E68
M system	–	○
L system	–	○

With this function, status of various devices to be used for the external signals and user ladders can be traced, and ladder program can be monitored.
Set the <BITDEVICE> (max. 8 points) and <WORDDEVICE> (max. 2 points) to be traced on the "DEVICE SET" screen and also set the <MONI SELECT>. Once trace is started, tracing status of the set device can be monitored in the "DEVICE MONITOR" screen. The latest 256 sampling data can be monitored for each device.
Trace can be stopped at the specific conditions by setting the trigger.
Trace is continued until it is stopped manually or by trigger. So, trace is automatically started when the power is turned OFF and ON during trace execution.

16. Cabinet and Installation

16.1 Cabinet Construction

The configuration of the unit used by the EZMotion-NC E60/E68 series is shown below.
Refer to the Connection / Maintenance Manual for details.



List of configuration units

(1) NC unit

Model	**Configuration module model**	**Function**
FCU6-MU071 (E60)	HR761/HR763 card	Main card
	HR741 card	Memory card
	Q6-BAT	Battery
	Base plate	
	Cover	
FCU6-MU072 (E68)	HR761/HR763 card	Main card
	HR742 card	Memory card
	Q6-BAT	Battery
	Base plate	
	Cover	

(2) Option for NC unit

Configuration module model	**Function**
HR753 card (E60)	E60 spindle connection option
HR751 card (E68)	PLC accelerator card

(3) Option for front IC card I/F unit

Model	Configuration module model	Function
FCU6-EP105-1 (E68)	HR253	Front IC card
	HR551	Bus extension card
	F161 cable	Bus cable
	Installation plate	

(4) Display unit

Model	Configuration module model	Function
FCU6-DUE71 /FCU6-DUE71-1 (Note 1) (E60)	MDT962B-4A	9-type monochrome CRT
	FCUA-R100 cable	CRT power supply cable
	F590 cable	Between HR761 and CRT
	Menu key	
	Escutcheon	
	Base plate	
FCU6-DUT11 /FCU6-DUT11-1 (Note 1) (E60)	LTBLDT168G6C	7.2-type monochrome LCD
	HR721 card	Power supply for backlight
	F090 cable	Between HR761 and HR721
	NZ24-4 cable	Between HR721 and LCD
	Menu key	
	Escutcheon	
	Base plate	
FCU6-DUN24 (Note 2) (E68) FCU6-DUN26 (E60) (Note 3)	AA084VC06	8.4-type color LCD
	HR722 card	LCD relay card
	CXA-L0605-VJL	Backlight inverter
	F090 L0.1M cable	Between HR761 and HR722
	F098 cable	Between HR722 and LCD
	F484 L0.25M cable	Inverter cable
	Menu key	
	Escutcheon	
	Base plate	

(Note 1) The units with the name FCU6-xxxxx-1 are provided with an adapter for mounting on the front of the units.

(Note 2) With E68 standard, the display unit is mounted from the front.

(Note 3) Escutcheon of FCU6-DUN26 differs from that of FCU6-DUN24.

(5) Keyboard unit

Model	Configuration module model	Function
FCU6-KB071 /FCU6-KB071-1 (Note 1) (E60)	Housing	Sheet attached for machining center
	KS-6MB911A-P	Keyboard switch
	F053 cable	Between HR761 and keyboard
	Base plate	
FCU6-KB024 (Note 2) (E60/E68)	Housing	Sheet attached for machining center
	KS-6MB911B-P	Keyboard switch
	F054 cable	Between HR761 and keyboard
	Base plate	

(6) Base I/O unit

Model		Configuration module model	Function
FCU6-HR341	Sink/Source input 64 points Sink output 48 points Analog output 1 point I/O share line (Note 5)	HR341 card	DI64/DO48/AO1
		FCUA-R-TM	Terminal resistor
		Aluminum die-cast	
FCU6-HR351	Sink/Source input 64 points Sink output 48 points Analog output 1 point I/O share line (Note 5)	HR351 card	DI64/DO48/AO1
		FCUA-R-TM	Terminal resistor
		Aluminum die-cast	
FCU6-DX220 (E68)	Sink/Source input 64 points Sink output 64 points I/O share line (Note 3)(Note 4)	HR327 card	DI64/DO64
		FCUA-R-TM	Terminal resistor
		Aluminum die-cast	
FCU6-DX221 (E68)	Sink/Source input 64 points Source output 64 points I/O share line (Note 3)(Note 4)	HR337 card	DI64/DO64
		FCUA-R-TM	Terminal resistor
		Aluminum die-cast	

(Note 1) The units with the name FCU6-xxxxx-1 are provided with an adapter for mounting on the front of the units.

(Note 2) With E68 standard, the keyboard unit is mounted from the front.

(Note 3) DI/O is a cable with no strain relief.

(Note 4) The 5th to 8th channels of SKIP and 2nd channel of RIO cannot be used.

(Note 5) I/O share line is the interface with servo drive unit, remote I/O, skip signal input and synchronous feed encoder.

(7) Remote I/O unit

Model		Component module model	Function
FCUA-DX100	Sink/Source input 32 points Sink output 32 points	RX311 card	DI32/DO32
		Case	
FCUA-DX101	Sink/Source input 32 points Sink output 32 points	RX312 card	DI32/DO32
		Case	
FUCA-DX110	Sink/Source input 64 points Sink output 48 points	RX311 card	DI32/DO32
		RX321-1 card	DI32/DO16
		Case	
FCUA-DX111	Sink/Source input 64 points Sink output 48 points	RX312 card	DI32/DO32
		RX322-1 card	DI32/DO16
		Case	
FUCA-DX120	Sink/Source input 64 points Sink output 48 points Analog output 1 point	RX311 card	DI32/DO32
		RX321 card	DI32/DO16/AO1
		Case	
FUCA-DX121	Sink/Source input 64 points Sink output 48 points Analog output 1 point	RX312 card	DI32/DO32
		RX322 card	DI32/DO16/AO1
		Case	
FCUA-DX140	Sink/Source input 32 points Sink output 32 points Analog input 4 points Analog output 1 point	RX311 card	DI32/DO32
		RX341 card	AI4/AO1
		Case	
FCUA-DX141	Sink/Source input 32 points Sink output 32 points Analog input 4 points Analog output 1 point	RX312 card	DI32/DO32
		RX341 card	AI4/AO1
		Case	

(8)Peripheral devices

Model		Component module model	Function
PD25	Input: 200VAC Output: 24VDC		With the power supply ON/OFF function
HD60	Manual pulse generator		Manual pulse generator for 12VDC (25pulse/rev) Use F320/F321 cable. Without MELDAS logo.
HD60-1	Manual pulse generator		Manual pulse generator for 12VDC (25pulse/rev) Use F320/F321 cable. With MELDAS logo.
UFO-01-2Z9	Manual pulse generator		Manual pulse generator for 5VDC (100pulse/rev) Use F023/F024 cable. Without MELDAS logo.
FCUA-R-TM	Terminal connector		Terminal for remote I/O communication
FCUA-A-TM	Terminal connector		Terminal for drive part communication
Ground plate D		A complete set of ground plate D	Appendix 1.9
Ground plate E		A complete set of ground plate E	Appendix 1.9
OSE-1024-3-15-68	Synchronous feed encoder		

16.2 Power Supply

⚠ **Caution**

⚠ Follow the power supply specifications (input voltage range, frequency range, momentary power failure time range) described in this manual.

⚠ Follow the environment conditions (ambient temperature, humidity, vibration, ambient atmosphere) described in this manual.

(1) Environment conditions in control part

(a) E60: Color display

Unit name			NC unit	Display unit	Keyboard unit
Model			**FCU6-MU071**	**FCU6-DUN26**	**FCU6-KB024**
General specification	Ambient temperature	During operation	0 to 55°C		
		During storage	-20 to 60°C		
	Ambient humidity	Long term	10 to 75% RH (With no dew condensation)		
		Short term	10 to 95% RH (With no dew condensation) **(Note 1)**		
	Vibration resistance		$4.9m/s^2$ or less		
	Shock resistance		$29.4m/s^2$ or less		
	Working atmosphere		No corrosive gas, dust or oil mist		
Power supply specification	Power supply voltage		24VDC ±5% Ripple 200mV max.		
	Instantaneous stop tolerance time		Depends on the specifications of 24VDC power supply unit used. (Use more than 20ms)		
	Current consumption (max.)		2A (NC unit + display unit + keyboard unit)		
Heating value (max.)			50W		
Mass			3.2kg (with FUC6-DUN26)		1.0kg
Outline dimension			Refer to Appendix 3.1		

(Note 1) Short term refers to within one month.

(b) E60: Monochrome display

Unit name			NC unit	Display unit	
Model			**FCU6-MU071**	**FCU6-DUE71**	**FCU6-DUT11**
General specification	Ambient temperature	During operation	0 to 55°C		Surface temperature of LCD display unit: 0 to 50°C **(Note 1)**
		During storage	-20 to 60°C		
	Ambient humidity	Long term	10 to 75% RH (With no dew condensation)		
		Short term	10 to 95% RH (With no dew condensation) **(Note 2)**		
	Vibration resistance		$4.9m/s^2$ or less		
	Shock resistance		$29.4m/s^2$ or less		
	Working atmosphere		No corrosive gas, dust or oil mist		
Power supply specification	Power supply voltage		24VDC ±5% Ripple 200mV max.	Single phase 100VAC -15% to +10% 50/60Hz	—
	Instantaneous stop tolerance time		20ms (with an external power supply unit PD25)		
	Current consumption (max.)		2A (NC unit + display unit)	100VAC 0.4A	—
Heating value (max.)			80W (with FCU6-DUE71), 50W (with FCU6-DUT11)		
Mass			5.5kg (with FCU6-DUE71), 2.5 kg (with FCU6-DUT11)		
Outline dimension			Refer to Appendix 3.1		

(Note 1) If it is hotter than 45°C, the quality of the LCD (contrast ratio) deteriorates.
(Note 2) Short term refers to within one month.

(c) E68

<table>
<tr><td colspan="3">Unit name</td><td>NC unit</td><td>Display unit</td><td>Keyboard unit</td></tr>
<tr><td colspan="3">Model</td><td>FCU6-MU072</td><td>FCU6-DUN24</td><td>FCU6-KB024</td></tr>
<tr><td rowspan="7">General specification</td><td rowspan="2">Ambient temperature</td><td>During operation</td><td colspan="3">0 to 55°C</td></tr>
<tr><td>During storage</td><td colspan="3">-20 to 60°C</td></tr>
<tr><td rowspan="2">Ambient humidity</td><td>Long term</td><td colspan="3">10 to 75% RH (With no dew condensation)</td></tr>
<tr><td>Short term</td><td colspan="3">10 to 95% RH (With no dew condensation) (Note 1)</td></tr>
<tr><td colspan="2">Vibration resistance</td><td colspan="3">4.9m/s^2 or less</td></tr>
<tr><td colspan="2">Shock resistance</td><td colspan="3">29.4m/s^2 or less</td></tr>
<tr><td colspan="2">Working atmosphere</td><td colspan="3">No corrosive gas, dust or oil mist</td></tr>
<tr><td rowspan="3">Power supply specification</td><td colspan="2">Power supply voltage</td><td colspan="3">24VDC ±5%
Ripple 200mV max.</td></tr>
<tr><td colspan="2">Instantaneous stop tolerance time</td><td colspan="3">Depends on the specifications of 24VDC power supply unit used. (Use more than 20ms)</td></tr>
<tr><td colspan="2">Current consumption (max.)</td><td colspan="3">2A
(NC unit + display unit + keyboard unit)</td></tr>
<tr><td colspan="3">Heating value (max.)</td><td colspan="3">50W</td></tr>
<tr><td colspan="3">Mass</td><td colspan="2">3.2kg (with FCU6-DUN24)</td><td>1.0kg</td></tr>
<tr><td colspan="3">Outline dimension</td><td colspan="3">Refer to Appendix 3.2</td></tr>
</table>

(Note 1) Short term refers to within one month.

(2) Environment conditions in electric cabinet

Unit name			**Base I/O unit**			
Model			**FCU6-HR341**	**FCU6-HR351**	**FCU6-DX220**	**FCU6-DX221**
General specification	Ambient temperature	During operation	0 to 55°C			
		During storage	-20 to 60°C			
	Ambient humidity	Long term	10 to 75% RH (With no dew condensation)			
		Short term	10 to 95% RH (With no dew condensation) **(Note 1)**			
	Vibration resistance		$4.9m/s^2$ or less			
	Shock resistance		$29.4m/s^2$ or less			
	Working atmosphere		No corrosive gas, dust or oil mist			
Power supply specification	Power supply voltage		24VDC ±5% Ripple 200mV max.			
	Instantaneous stop tolerance time		Depends on the external power supply used. (Use more than 20ms)			
	Current consumption (max.)		0.8A **(Note 2)**			
Heating value (max.)			40W **(Note 3)**			
Mass			0.6kg			
Outline dimension			Refer to Appendix 3.4.			

(Note 1) Short term refers to within one month.
(Note 2) This value is only of the control circuit part (DCIN connector). For the current value of the I/O circuit, calculate with the number of points used and load.
(Note 3) When all DI/DO points are ON.

(3) Remote I/O unit

Unit name			**Remote I/O unit**			
Model			**FCUA-DX10x**	**FCUA-DX11x**	**FCUA-DX12x**	**FCUA-DX14x**
General specification	Ambient temperature	During operation	0 to 55°C			
		During storage	-20 to 60°C			
	Ambient humidity	Long term	10 to 75% RH (With no dew condensation)			
		Short term	10 to 95% RH (With no dew condensation) **(Note 1)**			
	Vibration resistance		$4.9m/s^2$ or less			
	Shock resistance		$29.4m/s^2$ or less			
	Working atmosphere		No corrosive gas, dust or oil mist			
Power supply specification	Power supply voltage		24VDC ±5% Ripple 200mV max.			
	Instantaneous stop tolerance time		Depends on the external power supply used. (Use more than 20ms)			
	Current consumption (max.)		0.4A **(Note 2)**	0.7A **(Note 2)**	0.8A **(Note 2)**	0.7A **(Note 2)**
Heating value (max.)			25W **(Note 3)**	40W **(Note 3)**	40W **(Note 3)**	30W **(Note 3)**
Mass			470g	570g	590g	550g
Outline dimension			Refer to Appendix 3.5.			

(Note 1) Short term refers to within one month.
(Note 2) This value is only of the control circuit part (DCIN connector). For the current value of the I/O circuit, calculate with the number of points used and load.
(Note 3) When all DI/DO points are ON.

(4) Servo / Spindle

Refer to the following manuals for details on the servo and spindle system.

MDS-C1 Series SPECIFICATIONS MANUAL (BNP-C3040)
MDS-CH Series SPECIFICATIONS MANUAL (BNP-C3016)
MDS-R Series SPECIFICATIONS AND INSTRUCTION MANUAL (BNP-C3045)
MDS-B-SVJ2 Series SPECIFICATIONS AND INSTRUCTION MANUAL (BNP-B3937)
MDS-B-SPJ2 Series SPECIFICATIONS MANUAL (BNP-B2164)
MR-J2-CT Series SPECIFICATIONS AND INSTRUCTION MANUAL (BNP-B3944)

17. Servo / Spindle System

Refer to the following manuals for details on the servo and spindle system.

MDS-C1 Series	SPECIFICATIONS MANUAL	(BNP-C3040)
MDS-CH Series	SPECIFICATIONS MANUAL	(BNP-C3016)
MDS-R Series	SPECIFICATIONS AND INSTRUCTION MANUAL	(BNP-C3045)
MDS-B-SVJ2 Series	SPECIFICATIONS AND INSTRUCTION MANUAL	(BNP-B3937)
MDS-B-SPJ2 Series	SPECIFICATIONS MANUAL	(BNP-B2164)
MR-J2-CT Series	SPECIFICATIONS AND INSTRUCTION MANUAL	(BNP-B3944)

17.1 Feed Axis

17.1.1 MDS-C1-V1/C1-V2 (200V)

(1) Servo motor: HCxx-A51/E51(1000kp/rev)

	E60	E68
M system	–	☐
L system	–	☐

(2) Servo motor: HCxx-A42/E42 (100kp/rev)

	E60	E68
M system	–	☐
L system	–	☐

17.1.3 MDS-CH-V1/CH-V2 (400V)

(1) Servo motor: HCxx-A51/E51 (1000kp/rev)

	E60	E68
M system	–	☐
L system	–	☐

(2) Servo motor: HCxx-A42/E42 (100kp/rev)

	E60	E68
M system	–	☐
L system	–	☐

17.1.4 MDS-B-SVJ2 (Compact and small capacity)

(1) Servo motor: HCxx-A42/E42 (100kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

(2) Servo motor: HCxx-A47 (100kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

(3) Servo motor: HCxx-A33/E33(25kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

(4) Servo motor: HC-SF/HC-RF (16kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

(5) Servo motor: HC-MF (8kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

17.1.6 MDS-R-V1/R-V2 (200V Compact and small capacity)

(1) Servo motor: HFxx-A48 (260kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

(2) Servo motor: HFxx-A47 (130kp/rev)

	E60	E68
M system	☐	☐
L system	☐	☐

17.2 Spindle

17.2.1 MDS-C1-SP/C1-SPH/C1-SPM/B-SP (200V)

(1) Spindle motor: SJ/SJ-V

	E60	E68
M system	–	□
L system	–	□

(2) IPM spindle motor: SJ-PMF

	E60	E68
M system	–	□
L system	–	□

17.2.2 MDS-CH-SP/CH-SPH (400V)

	E60	E68
M system	–	□
L system	–	□

17.2.3 MDS-B-SPJ2 (Compact and small capacity)

(1) Spindle motor: SJ-P/SJ-PF

	E60	E68
M system	Δ	□
L system	Δ	□

17.3 Auxiliary Axis

17.3.1 Index/Positioning Servo: MR-J2-CT

(1) Servo motor: HC-SF/HC-RF(16kp/rev)

	E60	E68
M system	□	□
L system	□	□

(2) Servo motor: HC-MF(8kp/rev)

	E60	E68
M system	□	□
L system	□	□

17.4 Power Supply

17.4.1 Power Supply: MDS-C1-CV/B-CVE

	E60	E68
M system	–	□
L system	–	□

17.4.2 AC Reactor for Power Supply

	E60	E68
M system	–	□
L system	–	□

17.4.3 Ground Plate

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

18. Machine Support Functions

18.1 PLC

18.1.1 PLC Basic Function

18.1.1.1 Built-in PLC Basic Function

	E60	E68
M system	○	○
L system	○	○

(1) Ladder commands

Basic commands (bit processing commands)

20 commands including LD, LDI, OR, ORI, AND, ANI, OUT, PLS, etc.

Function commands

76 commands including data transfer, 4 basic arithmetic operations, logic arithmetic operations, large/small identification, binary/BCD conversion, branching, conditional branching, decoding, encoding, etc.

Exclusive commands

ATC control commands, and 14 others

Tool life management

Processing speed 2µs/step

(2) Devices

The table below lists the devices that can be used by the PLC. (GX Developer)

Device	Device range		Unit	Details	Remarks
X	X0 to X4BF	(1216 points)	1 bit	Input signal to PLC Machine input, etc.	
Y	Y0 to Y53F	(1344 points)	1 bit	Output signal from PLC Machine output, etc.	
M	M0 to M8191	(8192 points)	1 bit	Temporary memory	
F	F0 to F127	(128 points)	1 bit	Temporary memory, Alarm message interface	
L	L0 to L255	(256 points)	1 bit	Latch relay (backup memory)	
SM	SM0 to SM127	(128 points)	1 bit	Special relay	
T	T0 to T15	(16 points)	1 bit/16 bits	10 ms unit timer	
	T16 to T55	(40 points)	1 bit/16 bits	10 ms unit timer (fixed timer)	
	T56 to T135	(80 points)	1 bit/16 bits	100 ms unit timer	
	T136 to T231	(96 points)	1 bit/16 bits	100 ms unit timer (fixed timer)	
	T232 to T239	(8 points)	1 bit/16 bits	100 ms integral timer	
	T240 to T255	(16 points)	1 bit/16 bits	100 ms integral timer (fixed timer)	
C	C0 to C23	(24 points)	1 bit/16 bits	Counter	
	C24 to C127	(104 points)	1 bit/16 bits	Counter (fixed counter)	
D	D0 to D1023	(1024 points)	16 bits/ 32 bits	Data register Register for arithmetic operations	
R	R0 to R8191	(8192 points)	16 bits/ 32 bits	File register. R500 to R549 and R1900 to R2799 are released to the user for interface between the PLC and controller. R1900 to R2799 are backed up by the battery.	
Z	Z0 to Z1	(2 point)	16 bits	For D or R address indexing (for ±n)	
N	N0 to N7	(8 points)	–––	Master controller nesting level	
P	P0 to P255	(256 points)	–––	Label for conditional jump and subroutine call commands	
K	K-32768 to K32767		–––	Decimal constant for 16-bit command	
	K-2147483648 to K2147483647		–––	Decimal constant for 32-bit command	
H	H0 to HFFFF		–––	Hexadecimal constant for 16-bit command	
	H0 to HFFFFFFFF		–––	Hexadecimal constant for 32-bit command	

(Note) The maximum number of part systems for E60/E68 is 1. Therefore, 2nd part system cannot be used.

(3) External alarm messages

The contents of the alarms that have occurred during sequence (user PLC) processing can be displayed on the screen.

Up to four alarm message displays can be displayed simultaneously on the alarm diagnosis screen. The maximum length of one message is 32 characters.

(4) External operator messages

When a condition has arisen in which a message is to be relayed to the operator, an operator message can be displayed separately from the alarm message.

The maximum length of an operator message on the alarm diagnosis screen is 60 characters.

The number of messages displayed at the same time is one.

(5) PLC switches

32 points of PLC switches can be set on the screen, and the ON/OFF control executed. The switches can be used as part of the machine operation switches. The switch applications can be freely determined with the sequence program, and each switch name can be created with the PLC and displayed on the screen.

(6) Load meter display

A load meter can be displayed on the screen.

Up to two axes designated with the built-in PLC such as the spindle load and Z axis load can be displayed as bar graphs on the screen.

(7) Timer / counter setting display

(a) PLC timer

The setting value of the timer used by the built-in PLC can be set from the screen on the screen.

The timer types include the 10ms, 100ms and 100ms integral types.

Whether to validate the timer in the PLC program or to validate the setting value from the screen can be selected with the parameters.

Whether to hold the integral timer when the power is turned OFF can also be selected.

(b) PLC counter

The setting value of the counter used by the built-in PLC can be set from this screen.

Whether to validate the constants in the PLC program or to validate the setting value from the screen can be selected with the parameters.

Whether to hold the counter value when the power is turned OFF can also be selected.

(8) PLC parameter setting display

The PLC constants set with the data type and the bit selection parameters set with bit types can be set from the screen as parameters used by the built-in PLC.

(a) PLC constants

There are PLC constants that can be set with data types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up. If data is set in the R register corresponding to the PLC constant with sequence program MOV commands, etc., the data will be backed up. However, the display will not change, so enter another screen, and then select this screen again.

Up to 48 items can be set, and the setting range is ±8 digits.

(b) Bit selection parameters

There are bit selection parameters set with bit types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up.

When using bit operation in the sequence program, the details of the R register are transferred to the temporary memory (M) with the MOV command. If the data is set in the R register corresponding to the bit selection with the MOV command, etc., the data will be backed up. However, the display will not change, so enter another screen and then select this screen again.

(9) External key input

By inputting the key data from the built-in PLC, the same operation as when the operator operates the operation board can be done.

(10) Real spindle speed output

The real spindle speed is converted by the signals of the encoder installed on the spindle and is output to the PLC. The output increment is 0.001r/min.

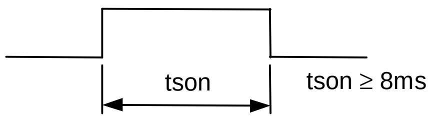
(11) Workpiece counter display

The number of workpieces can be set and displayed when continuously machining workpieces.
The M code to be count, the current number of machined workpieces and the max. machining value is set with parameters.
This data can be read by the user PLC (when built-in PLC specifications are used), and the number of machined workpieces can be controlled. A signal will be output to the PLC when the counted number reaches the set max. value.

(12) High speed input/output signal

There are signals that can be input and output at a 7.1ms cycle for high-speed processing.

(a) Input signal ON time



(b) After the signal output is set in the interface, it can be output to the machine side with a max. 7.1ms delay. The input also appears on the interface with a 7.1ms delay.

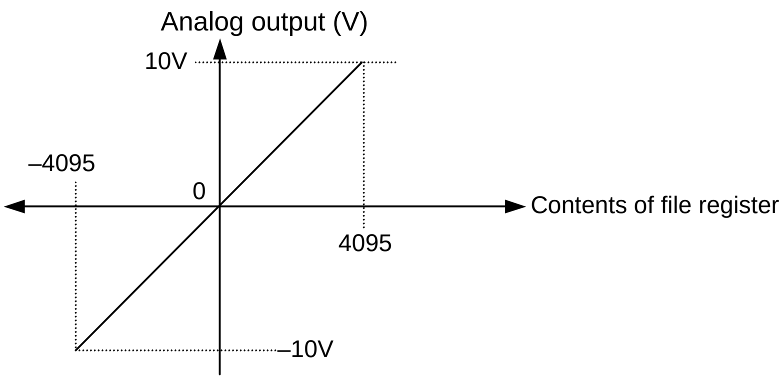
(c) The signals used for high-speed processing are assigned with the parameters. Assignment is possible in a continuous 16-point unit.

(13) PLC analog voltage control

(a) Analog output

When the specified data is put in the file register, the corresponding analog voltage is output from the analog output external connector.

<Relationship between file register contents and analog output voltage>



Output voltage	0 to ±10V (±5%)
Resolution	Full scale (10V)/4095
Load condition	10 kΩ resistance load (standard)
Output impedance	220 Ω

(Note) The remote I/O unit DX12x/DX14x and the base I/O unit HR341/HR351 are required for analog output.

18.1.2 Built-in PLC Processing Mode

An exclusive sequence program that controls the various signals between the controller and machine to realize operation applicable to each machine must be created.
The sequence execution modes include high-speed processing and main processing.

(1) High-speed processing
This mode provides repeated execution at 7.1ms cycles. It is used to process signals requiring high speeds.
The max. number of program steps for high-speed processing (1 period) is 150 steps when using basic commands.

(2) Main processing
This mode provides normal sequence processing. The processing cycle depends on the number of sequence steps.

18.1.2.2 MELSEC Development Tool I/F

	E60	E68
M system	○	○
L system	○	○

This function enables the data of the PLC contained inside the NC system to be developed and debugged using the GX Developer installed in a personal computer (OS: Windows).
Many and varied functions of the GX Developer make it possible to reduce the PLC data development and debugging time.

18.1.3 Built-in PLC Capacity (Number of steps)

4000(PLC Emulation)

	E60	E68
M system	○	–
L system	○	–

10000(PLC Emulation)

	E60	E68
M system	–	○
L system	–	○

32000

	E60	E68
M system	–	Δ
L system	–	Δ

18.1.4 Machine Contact Input/Output I/F

	E60	E68
M system	○	○
L system	○	○

⚠ **Caution**

❗ Follow the remote type machine contact input/output interface described in this manual. (Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)

Refer to "EZMotion-NC E60/E68 Series CONNECTION AND MAINTENANCE MANUAL" for details.

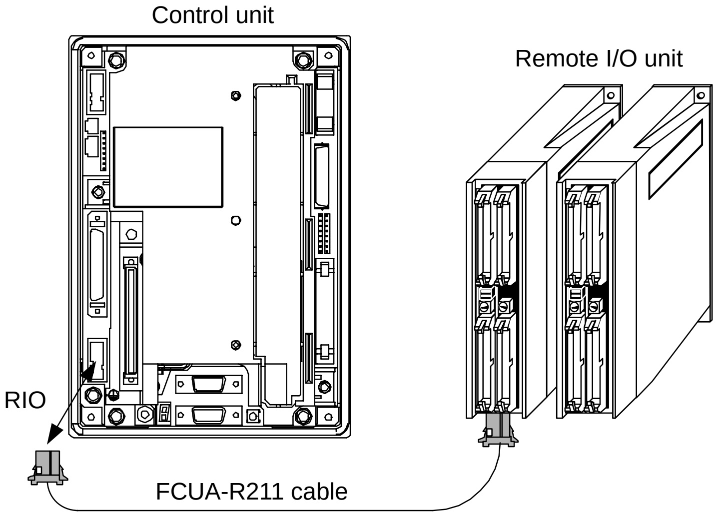
(1) Types of remote I/O units

Remote I/O unit list

Model		Component module model	Function
FCUA-DX100	Sink/Source input 32 points Sink output 32 points	RX311 card	DI32/DO32
		Case	
FCUA-DX101	Sink/Source input 32 points Sink output 32 points	RX312 card	DI32/DO32
		Case	
FUCA-DX110	Sink/Source input 64 points Sink output 48 points	RX311 card	DI32/DO32
		RX321-1 card	DI32/DO16
		Case	
FCUA-DX111	Sink/Source input 64 points Sink output 48 points	RX312 card	DI32/DO32
		RX322-1 card	DI32/DO16
		Case	
FUCA-DX120	Sink/Source input 64 points Sink output 48 points Analog output 1 point	RX311 card	DI32/DO32
		RX321 card	DI32/DO16/AO1
		Case	
FUCA-DX121	Sink/Source input 64 points Sink output 48 points Analog output 1 point	RX312 card	DI32/DO32
		RX322 card	DI32/DO16/AO1
		Case	
FCUA-DX140	Sink/Source input 32 points Sink output 32 points Analog input 4 points Analog output 1 point	RX311 card	DI32/DO32
		RX341 card	AI4/AO1
		Case	
FCUA-DX141	Sink/Source input 32 points Sink output 32 points Analog input 4 points Analog output 1 point	RX312 card	DI32/DO32
		RX341 card	AI4/AO1
		Case	

(2) Operation board remote I/O unit

With this function, the assignment of the operation board remote I/O directly connected with the control unit to the PLC device can be switched by the parameter. Depending on the parameter settings, the operation board remote I/O can be used as the remote I/O channel #2 equivalent.



Assignment of the operation board remote I/O unit to the PLC device includes the following two patterns and can be switched.

(a) Standard (M50 compatible) assignment method (Pattern 1)
PLC devices are assigned by the M50 compatible assignment method. That means that the input device Nos. are assigned to X100 and after, and the output device Nos. are assigned to Y100 and after.
Up to 4 stations can be used for the remote I/O units. Set the rotary switch within the range 0 to 3.

(b) Remote I/O channel #2 equivalent assignment method (Pattern 2)
PLC devices are assigned by the remote I/O channel #2 equivalent method. That means that the input device Nos. are assigned to X640 and after, and the output device Nos. are assigned to Y740 and after.
Up to 8 stations can be used for the remote I/O units. Set the rotary switch within the range 0 to 7.

Remote I/O unit Rotary switch No.	Device No. to be input		Device No. to be output	
	Pattern 1	Pattern 2	Pattern 1	Pattern 2
0	X100 to X11F	X640 to X65F	Y100 to Y11F	Y740 to Y75F
1	X120 to X13F	X660 to X67F	Y120 to Y13F	Y760 to Y77F
2	R80, R81	X680 to X69F	R180, R181	Y780 to Y79F
3	R82, R83	X6A0 to X6BF	R182, R183	Y7A0 to Y7BF
4	-	X6C0 to X6DF	-	Y7C0 to Y7DF
5	-	X6E0 to X6FF	-	Y7E0 to Y7FF
6	-	X700 to X71F	-	Y800 to Y81F
7	-	X720 to X73F	-	Y820 to Y83F

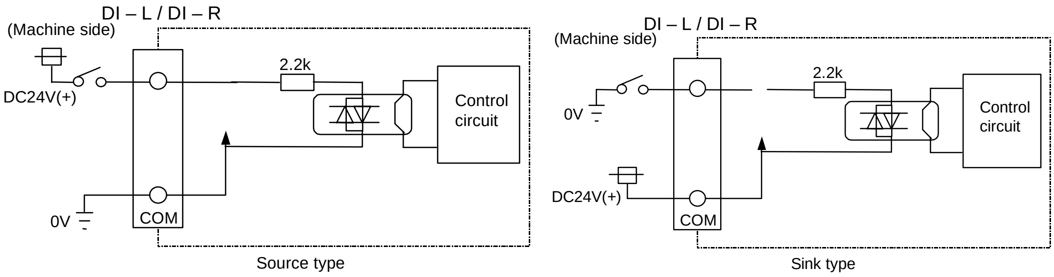
(Note 1) X108 is assigned for an operation board reset signal and cannot be used for the other applications.

(Note 2) Analog output is not possible even if DX12x or DX14x is connected.

(3) Outline of digital signal input circuit

There is a sink type and source type digital signal input circuit. The type is selected with a card unit in each unit.

Input circuit

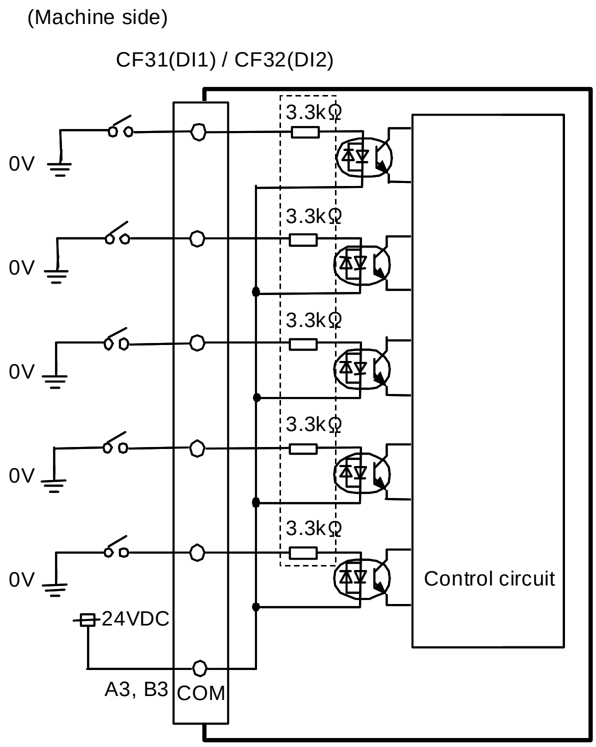


(4) Outline of digital signal input circuit

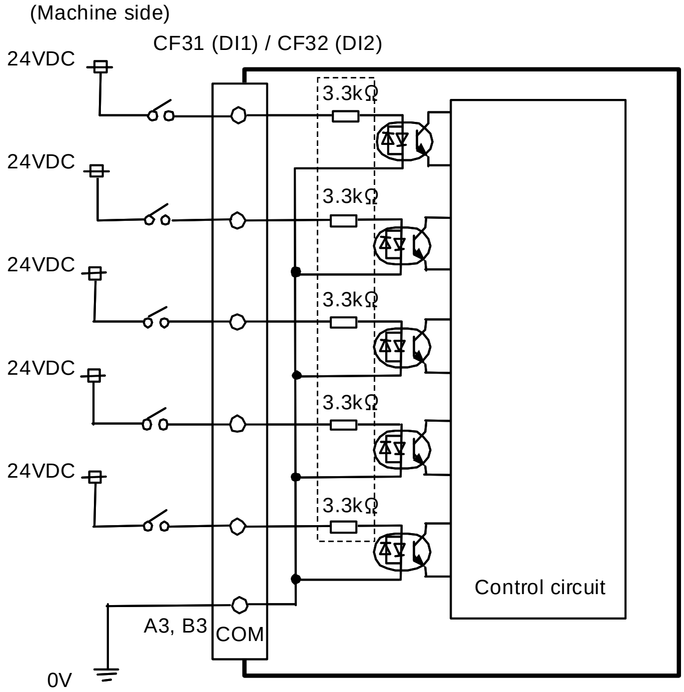
The digital signal input circuit can be selected from the sink type or source type for each connector of each unit.

(a) Input circuit

Source type



Sink type



: 2.2kΩ for FCU6-DX220/DX221

(b) Conditions for input

The input signals must be used within the following condition ranges.

Sink type

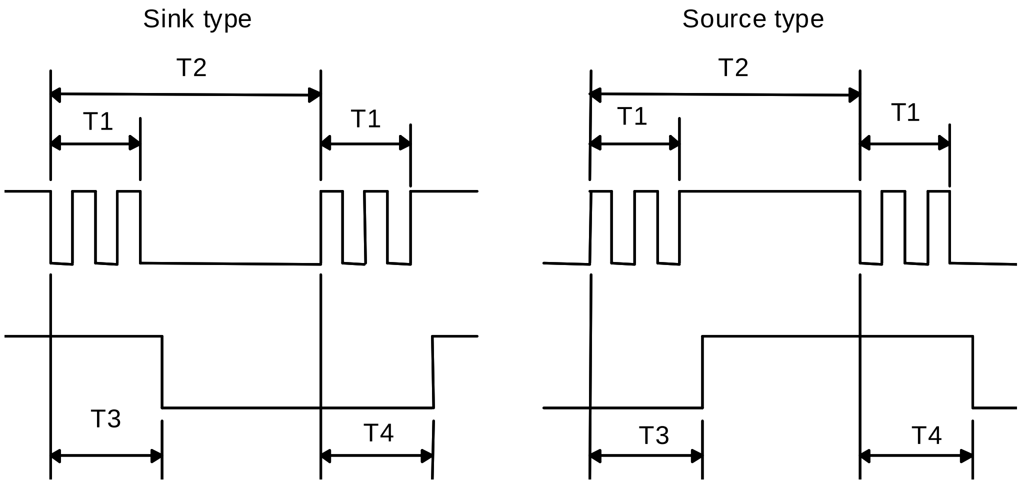
Input voltage at external contact ON	6V or less
Input current at external contact ON	9mA or more
Input voltage at external contact OFF	20V or more, 25.2V or less
Input current at external contact OFF	2mA or less
Tolerable chattering time	3ms or less (Refer to T1 below)
Input signal holding time	40ms or more (Refer to T2 below) **(Note 1)**
Input circuit operation delay time	3ms≦T3≒T4≦16ms
Machine side contact capacity	30V or more, 16mA or more

(Note 1) "40ms or more" is a rough standard. An input signal cannot be recognized unless it lasts longer than the period of the process cycle of the ladder.

Source type

Input voltage at external contact ON	18V to 25.2V
Input current at external contact ON	9mA or more
Input voltage at external contact OFF	4V or less
Input current at external contact OFF	2mA or less
Tolerable chattering time	3ms or less (Refer to T1 below)
Input signal holding time	40ms or more (Refer to T2 below) **(Note 1)**
Input circuit operation delay time	3ms≦T3≒T4≦16ms
Machine side contact capacity	30V or more, 16mA or more

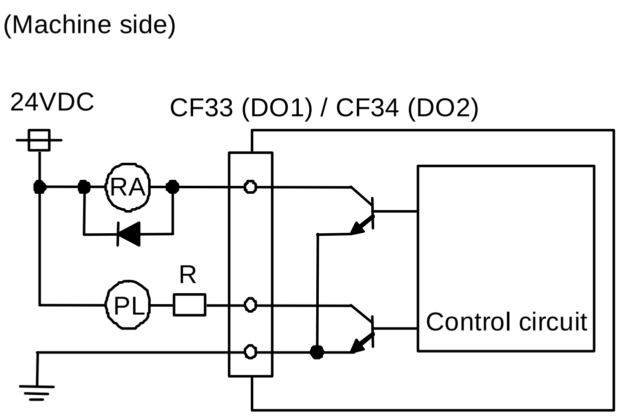
(Note 1) "40ms or more" is a rough standard. An input signal cannot be recognized unless it lasts longer than the period of the process cycle of the ladder.



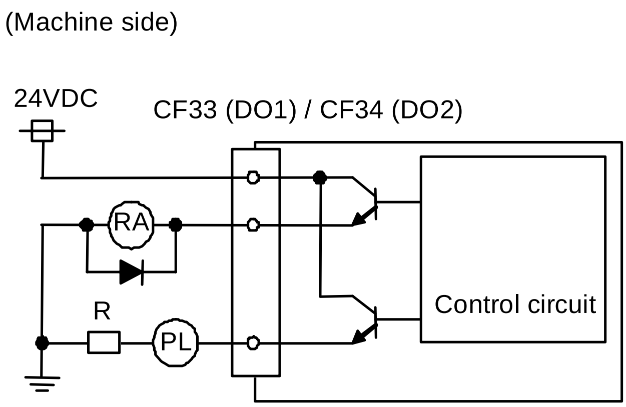
(5) Outline of digital signal output circuit

The digital signal output circuit has the sink type and source type

(a) Output circuit



Sink type



Source type

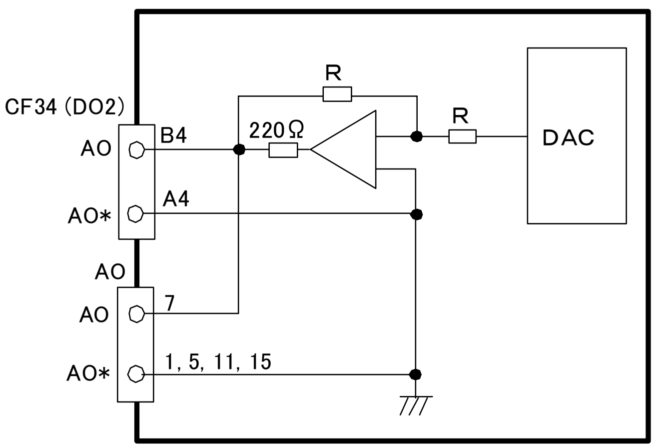
(b) Conditions for output

Insulation method	Non-insulation
Rated load voltage	24VDC
Max. output current	60mA/Point
Output delay time	40µs

(6) Outline of analog signal output circuit (Only FCU6-HR341/HR351)

Analog signals are output to the CF34 connector or AO connector, so connect either of the connectors which is easier to connect.

(a) Output circuit



(b) Conditions for output

Output voltage	0V to ±10V(±5%)
Resolution	12bits (±10V×n/4096) (n=2^0 to 2^{11})
Load condition	10kΩ load resistance
Output impedance	220Ω

18.1.5 Ladder Monitor

	E60	E68
M system	○	○
L system	○	○

This function enables the operating status of the sequence circuit to be checked on the screen.
The monitor functions include the following.

(1) Circuit monitoring
(2) Screen stop by monitor stop trigger point
(3) Entry monitoring
(4) Decimal-hexadecimal conversion present value monitoring

18.1.6 PLC Development

18.1.6.1 On-board Development

	E60	E68
M system	○	○
L system	○	○

PLC ladders can be developed on the control unit.
PLC ladder circuits can be created, edited, etc.

18.1.6.2 MELSEC Development Tool

	E60	E68
M system	○	○
L system	○	○

The GX Developer installed in a personal computer (OS: Windows) can be used.

18.1.9 PLC Password Lock

	E60	E68
M system	○	○
L system	○	○

This function makes it possible to use a code number to prohibit user PLC editing and input/output in order to prevent the illegal writing by the end users of the user PLC data prepared by the machine tool builder.

User PLC protection using code number

On-board PLC file writing, editing operations (write, insert, delete, change) for PLC circuits and PLC file input/output operations are enabled by inputting the code number.

The operations that are prohibited during user PLC protection by the code number are listed in the table below.

List of operations

Operation		**Code number input**	**No code number input**	**Remarks**
Ladder circuits	Readout	Possible	Possible	
	Write	Possible	Impossible	
	Insert	Possible	Impossible	
	Delete	Possible	Impossible	
	Change	Possible	Impossible	
	Monitor	Possible	Possible	
PLC files	RUN/STOP	Possible	Possible	
	Write	Possible	Impossible	
	Input/output	Possible	Impossible	
External alarm messages	Input/output	Possible	Impossible	

18.1.13 PLC Message

Japanese

	E60	E68
M system	○	○
L system	○	○

English

	E60	E68
M system	○	○
L system	○	○

German

	E60	E68
M system	○	○
L system	○	○

Italian

	E60	E68
M system	○	○
L system	○	○

French

	E60	E68
M system	○	○
L system	○	○

Spanish

	E60	E68
M system	○	○
L system	○	○

Chinese (Simplified Chinese characters)

	E60	E68
M system	○	○
L system	○	○

Portuguese

	E60	E68
M system	○	○
L system	○	○

Hungarian

	E60	E68
M system	○	○
L system	○	○

Dutch

	E60	E68
M system	○	○
L system	○	○

Swedish

	E60	E68
M system	○	○
L system	○	○

18.1.14 User PLC version up

	E60	E68
M system	–	○
L system	–	○

With this function, version upgrade is available by using the ROM cassette in which PLC program is written.

18.2 Machine Construction

18.2.1 Servo OFF

	E60	E68
M system	○	○
L system	○	○

When the servo OFF signal (per axis) is input, the corresponding axis is set in the servo OFF state. When the moving axis is mechanically clamped, this function is designed to prevent the servomotor from being overloaded by the clamping force.
Even if the motor shaft should move for some reason or other in the servo OFF state, the movement amount will be compensated in the next servo ON state by one of the following two methods. (You can select the compensation method using a parameter.)

(1) The counter is corrected according to the movement amount (follow up function).
(2) The motor is moved according to the counter and compensated.

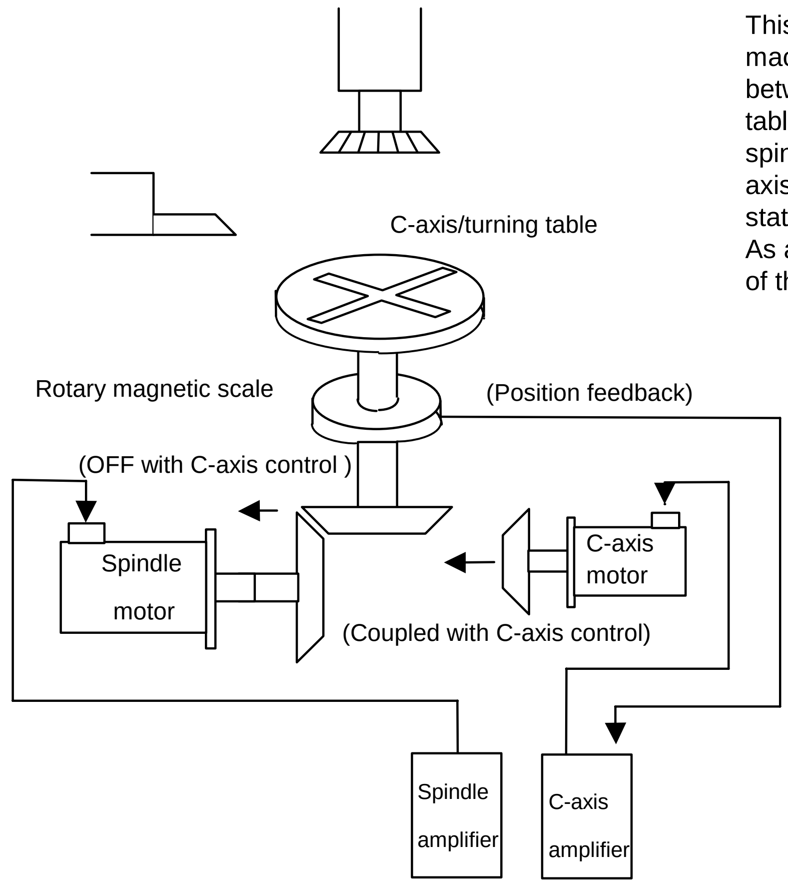
When follow up is designated, the movement amount will be compensated even in the emergency stop state.
The axis is simultaneously set with servo OFF to the interlock state.

Mechanical handle
Even if the servo OFF axis is moved with the mechanical handle with the application of the servo OFF function and follow up function, the position data can be constantly read in and the machine position updated. Thus, even if the axis is moved with the mechanical handle, the coordinate value display will not deviate.

18.2.2 Axis Detach

	E60	E68
M system	○	○
L system	○	○

This function enables the control axis to be freed from control. Conversely, an axis that has been freed from control can be returned to the control status.
This function enables the rotary table or attachments to be removed and replaced.
Automatic operation is disabled until the axis for which the control axis detach command has been released completes its dog-type reference position return.



This shows the configuration of a machine for which switching between the C axis and turning table is performed. When the spindle motor is connected, the C axis is placed in the detached status.
As a result, the position feedback of the detector is ignored.

```
POSITION

   X  123.456
   Z    0.000#1
   C  345.678><
```

The detached status > < is indicated on the right of the current value display on the POSITION screen and at the same time the servo ready for the controller output signal is set to OFF.
The current position counter retains the value applying when detach was assigned.

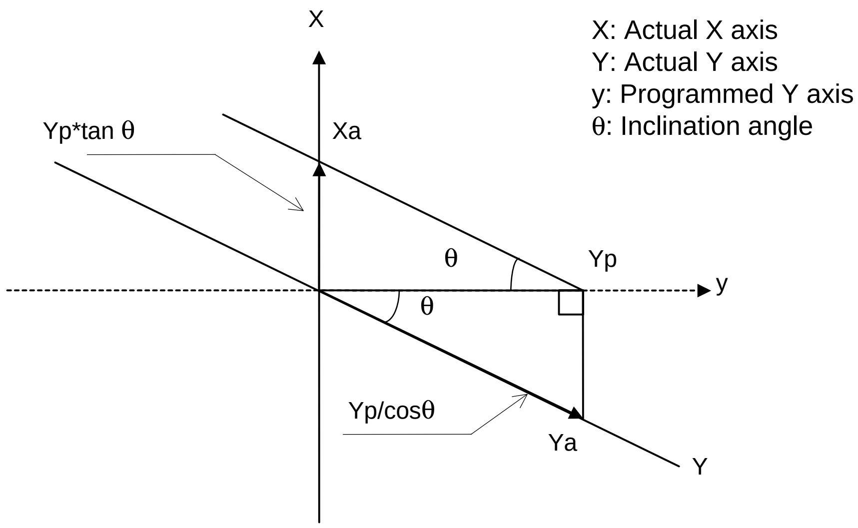
(Note) Axis detach can be executed even for the absolute position detection specifications axis, but when the axis is reinstalled, the zero point must be set.

18.2.4 Inclined Axis Control

	E60	E68
M system	–	–
L system	○	○

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables it to be programmed and controlled in the same way as with an orthogonal axis.
The inclination angle is set using a parameter, and axes are controlled using the movement amounts of the axes that are obtained through conversion and offset using this angle.

<Example of use> When the X axis serves as the basic axis and the Y axis serves as the inclined axis



The Y-axis position and Yp on the programmed coordinates (on the orthogonal coordinates) are respectively the Xa and Ya positions produced by combining the X axis and Y axis along which the machine actually moves.
Therefore, the Y-axis (inclined axis) movement amount is expressed by the following formula:

$Ya = Yp/\cos\theta$ (1)

The X-axis (basic axis) movement amount is offset by the inclined movement of the Y axis, and it is expressed as follows:

$Xa = Xp - Yp \times \tan\theta$ (2)

The Y-axis (inclined axis) speed is as follows:

$Fa = Fp/\cos\theta$

Xa, Ya and Fa are the actual movement amounts and speed. Xp, Yp and Fp are the movement amounts and speed on the program coordinates.

18.2.5 Index Table Indexing

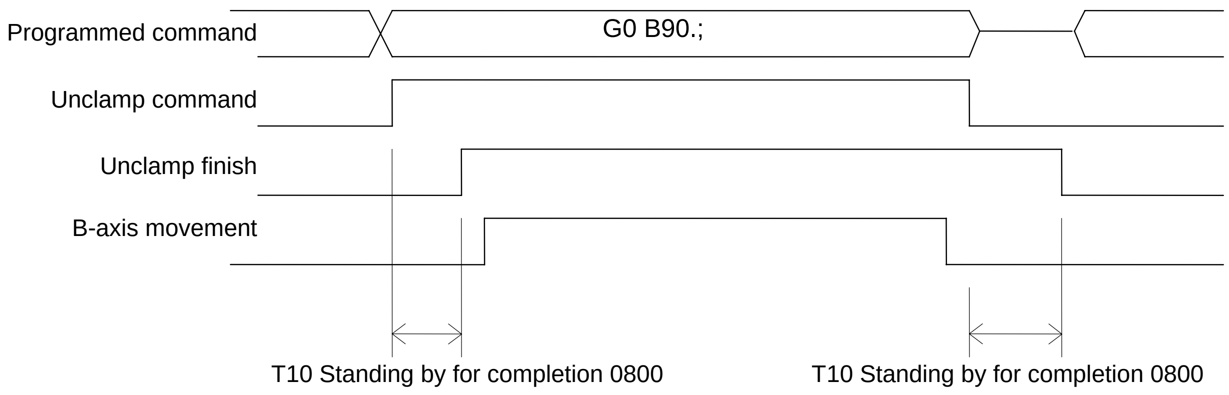
	E60	E68
M system	–	○
L system	–	○

The indexing of the index table can be performed by setting the index axes. Programming is facilitated because, in terms of the index commands, only the indexing angle need to be designated using the address of the programmed axis serving as the index setting axis, and there is no need to designate special M codes for clamping and unclamping the table.

The following operations are performed for the index table indexing function.

(1) Set "1" to the "index axis selection" parameter for the axis along which the indexing table is to be indexed.
(2) Designate the movement commands (absolute or incremental) for the selected axis using a program.
(3) The unclamp command signal is now output prior to the axis movement.
(4) When the axes are unclamped, the unclamp finish signal is set (ladder used for processing).
(5) After checking the unclamp finish signal, the designated axis starts moving.
(6) Upon completion of the movement, the unclamp command signal is set to OFF.
(7) Clamp the axes and set the unclamp finish signal to OFF (ladder used for processing).
(8) After checking that the unclamp finish signal is OFF, processing of the next block is initiated.

Operation timing chart



18.2.6 NSK Table Connection Control

	E60	E68
M system	–	O
L system	–	O

By assigning commands to the control unit from the 2nd miscellaneous function and the PLC and pulse train input, this function transmits commands by serial communication (RS-232C specification) to the dedicated drive unit of the NSK mega-torque motor.

Command designation method

By setting the command for the prescribed R register and the numerical value command following address B designated by the 2nd miscellaneous function as signed binary numbers using the PLC, the control unit prepares the positioning commands from this command and numerical value command, and it sends them to the drive unit by serial communication (RS-232C specification).
Furthermore, by setting ON the handle mode of the prescribed R register using the PLC, the pulse train input based on the handle is sent as the movement command to the drive unit.
Designate the commands with the number of mega-torque motor axes connected already set in the parameter and with the settings for the input/output basic parameters used for communication already set. Up to 16 axes can be connected as the mega-torque motor axes.

The numerical value command in angle increments is prepared by setting the numerical value following address B of the 2nd miscellaneous function as a signed binary number in the prescribed R register using the PLC.

18.2.7 Auxiliary Axis Control (J2-CT)

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

The MR-J2-CT drive unit for positioning and indexing can be connected for auxiliary axis control. The drive unit is a single-axis control unit, and the control is performed from the PLC. It comes with the following functions, and is suited to controlling a peripheral device of the machine.

(1) Feed functions
- (a) Four different feed rates can be set and selected using parameter settings.
- (b) Constant inclination acceleration/deceleration, linear acceleration/deceleration or soft acceleration/deceleration can be selected.
- (c) When rotary axis is used, automatic short-cut discrimination and rotary direction can be assigned by commands.

(2) Command methods
- (a) Station method
  Any point (station) obtained when the rotary axis has been divided into equal parts can be selected by a command, and the axis can be positioned at that point. The maximum number of divisions is 360.
- (b) Arbitrary coordinate designation method
  The arbitrary coordinates (absolute position as referenced to the zero point) can be commanded from the PLC and the axis can be positioned at these coordinates.

(3) Operation functions
- (a) JOG mode
  In this mode, the axis is rotated at a constant speed in the designated direction while the start signal is ON.
- (b) Automatic mode
  In this mode, the axis is positioned at the designated station number by the start signal.
- (c) Manual mode
  In this mode, the axis is rotated at a constant speed in the designated direction while the start signal is ON. When the start signal is set to OFF, the axis is positioned at the nearest station position.
- (d) Arbitrary coordinate mode
  In this mode, the axis is positioned at the arbitrary coordinates designated with the PLC by the start signal. When the start signal is set to OFF prior to the completion of the positioning, the axis immediately decelerates and stops.
- (e) Manual handle mode
  In this mode, axis travel is carried out by the pulse command (manual handle command) sent from the PLC.
- (f) Reference position return mode
  In this mode, the axis is positioned at the coordinate reference position. Two methods are used: one method is based on a dog switch and the other method is to carry out positioning to the reference position that is stored in the memory.
- (g) Press-fit-and-positioning mode
  In this mode, the axis is positioned while it is pressed against the machine end, etc.

18.3 PLC Operation

18.3.1 Arbitrary Feed In Manual Mode

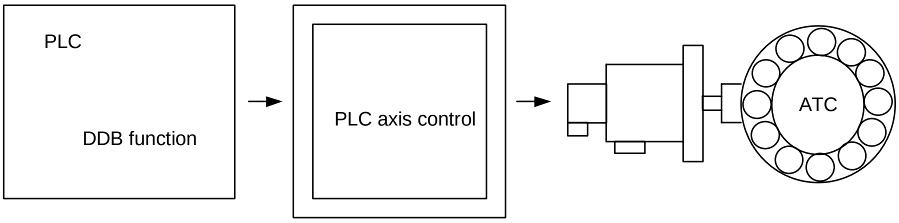
	E60	E68
M system	○	○
L system	○	○

This function enables the feed directions and feed rates of the control axes to be controlled using commands from the user PLC.
The arbitrary feed function controls the movement of the axes at the specified rates while the start signal is output from the PLC to the NC system.
PLC operations can be performed even during manual operation or automatic operation, but they cannot be performed when an axis for which arbitrary feed has been assigned is executing a command from the NC system (that is, while the axis is moving).

18.3.3 PLC Axis Control

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.



Item	Description	
	E60	E68
Number of control axes	Max. 1 axes	Max. 2 axes
Simultaneously controlled axes	PLC control axis is controlled independently from NC control axes. A multiple number of PLC axes can be started simultaneously.	
Least command increment	Least command increment 0.001mm (0.0001 inch)	Least command increment 0.001mm (0.0001 inch) 0.0001mm (0.00001 inch)
Feed rate	(Least command increment: 0.001mm) Rapid traverse 0 to 240000 mm/min (0 to 9448.8 inch/min) Cutting feed 0 to 240000 mm/min (0 to 9448.8 inch/min)	(Least command increment: 0.001mm) Rapid traverse 0 to 1000000 mm/min (0 to 100000 inch/min) Cutting feed 0 to 1000000 mm/min (0 to 100000 inch/min) (Least command increment: 0.0001mm) Rapid traverse 0 to 100000 mm/min (0 to 10000 inch/min) Cutting feed 0 to 100000 mm/min (0 to 10000 inch/min)
Movement commands	Incremental commands from current position Absolute commands for machine coordinate system	
	0 to ±99999999 (0.001mm/0.0001 inch)	0 to ±99999999 (0.001mm/0.0001 inch) 0 to ±9999999 (0.0001mm/0.00001 inch)
Operation modes	Rapid traverse, cutting feed, jog feed (+) (–), reference position return feed (+) (–), handle feed	
Acceleration/deceleration	Rapid traverse, jog feed, reference position return feed ..... Linear acceleration/deceleration Cutting feed ..... Exponential function acceleration/deceleration Handle feed ....Step	
Backlash compensation	Available	
Stroke end	None	
Soft limit	Available	
Rotary axis command	Available For absolute commands: amount within 1 rotation (rotation by amount remaining after division into 360) For incremental commands: rotation by assigned amount	
Inch/mm changeover	None Set to the command that corresponds to the feedback unit.	
Position detector	Encoder (Absolute position can also be detected.)	

(Note) Least input setting increment "C" (0.0001(°)mm/0.00001inch) is a specification for E68 system. This cannot be used with E60

18.4 PLC Interface

18.4.1 CNC Control Signal

	E60	E68
M system	○	○
L system	○	○

Control commands to the CNC system are assigned from the PLC. Input signals with an A/D conversion function and skip inputs that respond at high speed can also be used.

(1) Control signals

- Control signals for operations in automatic operation mode
- Control signals for operations in manual operation mode
- Control signals for program execution
- Control signals for interrupt operations
- Control signals for servo
- Control signals for spindle
- Control signals for mode selection
- Control signals for axis selection
- Control signals for feed rates

(2) Analog voltage control [M system]

When an analog voltage is input to an external connector used to connect CNC analog inputs, the data corresponding to the input voltage can be read out in the prescribed file register. This data can be used for load meter displays, thermal deformation compensation, etc. (Maximum 8 points)

(3) Skip signals

When signals are input to the skip input interface, they are processed by interrupt processing. This enables functions requiring a high response speed to be implemented. (Maximum 4 points)

For further details, refer to the PLC Interface Manual.

18.4.2 CNC Status Signal

	E60	E68
M system	○	○
L system	○	○

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC.
These signals can also be output as analog data by setting the data from the PLC in the R register.

Status output functions

(1) Controller operation ready

When the controller power is turned ON and the controller enters the operation ready status, the "Ready" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the controller power is supplied to when the controller ready status is entered.

(2) Servo operation ready

When the controller power is turned ON and the servo system enters the operation ready status, the "Servo ready" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the power is supplied to when the "Servo ready" signal is turned ON.

(3) In automatic operation

Generally, if the "cycle start" switch is turned ON in the automatic operation mode (memory, MDI), this signal is output until the reset state or emergency stop state is entered by the M02, M30 execution or the reset & rewind input to the controller using the reset button.

(4) In automatic start

The signal that denotes that the controller is operating in the automatic mode is output from the time when the cycle start button is pressed in the memory or MDI mode and the automatic start status has been entered until the time when the automatic operation is terminated in the automatic operation pause status entered by the "feed hold" function, block completion stop entered by the block stop function or resetting.

(5) In automatic pause

An automatic operation pause occurs and this signal is output during automatic operation from when the automatic pause switch is pressed ON until the automatic start switch is pressed ON, or during automatic operation when the mode select switch is changed from the automatic mode to the manual mode.

(6) In rapid traverse

The "In rapid traverse" signal is output when the command now being executed is moving an axis by rapid traverse during automatic operation.

(7) In cutting feed

The "In cutting feed" signal is output when the command now being executed is moving an axis by cutting feed during automatic operation.

(8) In tapping

The "In tapping" signal is output when the command now being executed is in a tap modal which means that one of the statuses below is entered during automatic operation.

(a) G84 (fixed cycle: tapping cycle)
(b) G74 (fixed cycle: reverse tapping cycle)
(c) G63 (tapping mode)

(9) In thread cutting

The "In thread cutting" signal is output when the command now being executed is moving an axis by thread cutting feed during automatic operation.

(10) In rewinding

The "In rewinding" signal is output when the reset & rewind signal is input by M02/M30, etc., during memory operation and the program currently being executed is being indexed.
The rewinding time is short, so there may be cases when it cannot be confirmed with the sequence program (ladder).

(11) Axis selection output

The "Axis selection output" signal for each axis is output to the machine during machine axis movement.

(a) Automatic mode
The signal is output in the movement command of each axis. It is output until the machine stops during stop based on feed hold or block stop.

(b) Manual mode (including incremental feed)
The signal is output while the axis is moving from the time when the jog feed signal is turned ON until the time when it is turned OFF and the machine feed stops.

(c) Handle feed mode
The signal is output at all times when the axis selection input is on.

(12) Axis movement direction

This output signal denotes the direction of the axis now moving, and for each axis a "+" (plus) signal and a "–" (minus) signal are output respectively.

(13) Alarm

This signal indicates the various alarm statuses that arise during controller operation. It is divided into the following types and output.

(a) System errors
(b) Servo alarms
(c) Program errors
(d) Operation errors

(14) In resetting

The "Reset" signal is output during the reset process when the reset & rewind command is input to the controller with the "reset" button on the setting and display unit (including the control unit) is pressed or when the "Reset" signal is input from the machine operation panel, etc.
This signal will also be output when the controller READY status is OFF, when the Emergency stop signal is input or when a servo alarm is occurring, etc.

(15) Movement command finish

In the memory or MDI automatic operation, the "Movement command finish" signal is output when the command block in the machining program features a movement command and when that block command has been completed.
When the movement command and M, S, T or B command have been assigned in the same block, then the movement command signal can be used as a sync signal for either executing the processing of the M, S, T or B command at the same time as the command or executing it upon completion of the movement command.

18.4.5 DDB

	E60	E68
M system	○	○
L system	○	○

The DDB (direct data bus) provides the function for PLC to directly read/write controller data. PLC can read the specified data into a buffer and set (write) the specified data into the controller by setting information required for read/write in the buffer and calling the DDB function. Generally, data is read/written for each data piece, but data related to control axes is processed in batch for as many axes as the specified number of axes.
The feature of the DDB function is the capabilities of referencing read data or write data in the next step just after a DDBA instruction is executed.

18.5 Machine Contact I / O

DI:64/DO:64

	E60	E68
M system	–	□
L system	–	□

DI:64/DO:48/A0:1

	E60	E68
M system	○	□
L system	○	□

DI:96/DO:80

	E60	E68
M system	–	□
L system	–	□

DI:96/DO:80/A0:1

	E60	E68
M system	–	□
L system	–	□

Additional DI/DO (DI:64/DO:48)

	E60	E68
M system	□	□
L system	□	□

Additional DI/DO(DI:32/DO:32)

	E60	E68
M system	□	□
L system	□	□

Operation board IO (DI:32/DO:32)

	E60	E68
M system	□	□
L system	□	□

Operation board IO DI:64/DO:48

	E60	E68
M system	□	□
L system	□	□

Remote IO 32/32

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

Remote IO 64/48

	E60	E68
M system	Δ	Δ
L system	Δ	Δ

18.7 Installing S/W for Machine Tools

18.7.3 Simple Customization

	E60	E68
M system	○	○
L system	○	○

Some messages can be displayed on the initial screen that appears when NC power is turned ON. The total messages can include up to 40 characters in a line and up to 10 lines.

Appendix 1. List of Specifications

Function code		Control unit recognition	Subject to parity V count	CRT display	Setting and display unit key-in	Stored in memory	Punch-out output		Internal NC system function
EIA	ISO						EIA	ISO	
0~9	0~9	Yes	Counted	Displayed	Key-in	Stored	0~9	0~9	Numerical data
A~Z	A~Z	Yes	Counted	Displayed	Key-in	Stored	A~Z	A~Z	Addresses
+	+	Yes	Counted	Displayed	Key-in	Stored	+	+	Sign, variable operator (+)
–	–	Yes	Counted	Displayed	Key-in	Stored	–	–	Sign, variable operator (–)
.	.	Yes	Counted	Displayed	Key-in	Stored	.	.	Decimal point
,	,	Yes	Counted	Displayed	Key-in	Stored	,	,	
/	/	Yes	Counted	Displayed	Key-in	Stored	/	/	Block delete (optional block skip), variable operator (÷)
EOR	%	Yes	Counted	Displayed (%)	No key-in (automatically inserted)	Stored	EOR	%	End of record (tape storage end), rewind start & stop during tape search
EOB/CR	LF/NL	Yes	Counted	Displayed (;)	Key-in, ;/EOB	Stored	EOB	LF	End of block
2+4+5	(	Yes	Counted	Displayed	Key-in, ;/EOB	Stored	2+4+5	(	Control out (comment start)
2+4+7	)	Yes	Counted	Displayed	Key-in, ;/EOB	Stored	2+4+7	)	Control in (comment end)
★	:	Yes	Counted	Displayed	No key-in	Stored	★	:	Program number address (instead of O, ISO only)
★	#	Yes	Counted	Displayed	Key-in	Stored	★	#	Variable number
★	*	Yes	Counted	Displayed	Key-in	Stored	★	*	Variable operator (×)
★	=	Yes	Counted	Displayed	Key-in	Stored	★	=	Variable definition
★	[	Yes	Counted	Displayed	Key-in	Stored	★	[	Variable operator
★	]	Yes	Counted	Displayed	Key-in	Stored	★	]	Variable operator
BS	BS	No	Counted	Blank	No key-in	Stored			
TAB	HT	No	Counted	Blank	No key-in	Stored			
SP	SP	No	Counted	Blank	Key-in	Stored	SP (T-V automatic adjustment)	SP (T-V automatic adjustment)	SPs starting with EOB and ending when first character or number code appears are not subject to parity V count.
	CR	No	Counted	Blank	No key-in	Stored			
DEL	DEL	No	Not counted	Not displayed	No key-in	Not stored			
All space	NULL	No	Not counted	Not displayed	No key-in	Not stored			
All mark	(DEL)	No	Not counted	Not displayed	No key-in	Not stored			
Any other	Any other	No	Counted	(Note 3)	No key-in	Stored			

(Note 1) ★ indicates that corresponding code pattern can be set by parameter.
(Note 2) Codes not listed above are stored on tape but an error will result during operation if they are not comments.
(Note 3) This denotes characters (including blanks) which are stored inside the controller and which correspond to the command codes. @ is not displayed.

Appendix 2. Format Details

		M system			
		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44 α+44	X+53 Y+53 Z+53 α+53	X+53 Y+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Y+44 Z+44 α+44	X+35 Y+35 Z+35 α+35	X+44 Y+44 Z+44 α+44	X+44 Y+44 Z+44 α+44
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35	I+44 J+44 K+44 R+44	I+35 J+35 K+35 R+35 (Note 5)
Dwell	0.001(°) mm/ 0.0001 inch	X+53 P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution)	F63(Feed per minute) F43(Feed per revolution)	F44(Feed per minute) F34(Feed per revolution) (Note 6)
	0.0001 (°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
Tool offset		H3 D3	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←

		L system			
		Metric command	Inch command	Rotary axis (Metric command)	Rotary axis (Inch command)
Program number		08	←	←	←
Sequence number		N5	←	←	←
Preparatory function		G3/G21	←	←	←
Movement axis	0.001(°) mm/ 0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44 α+44	X+53 Y+53 Z+53 α+53	X+53 Y+53 Z+53 α+53
	0.0001(°) mm/ 0.00001 inch	X+44 Z+44 α+44	X+35 Z+35 α+35	X+44 Z+44 α+44	X+44 Z+44 α+44
Arc and cutter radius	0.001(°) mm/ 0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44 K+44 R+44	I+35 K+35 R+35	I+44 K+44 R+44	I+35 K+35 R+35 (Note 5)
Dwell	0.001(°) mm/ 0.0001 inch	X+53 P+8	←	←	←
	0.0001(°) mm/ 0.00001 inch	X+53/P+8	←	←	←
Feed function	0.001(°) mm/ 0.0001 inch	F63(Feed per minute) F35(Feed per revolution)	F44(Feed per minute) F26(Feed per revolution)	F63(Feed per minute) F34(Feed per revolution)	F44(Feed per minute) F26(Feed per revolution) (Note 6)
	0.0001(°) mm/ 0.00001 inch	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution)	F54(Feed per minute) F34(Feed per revolution)	F35(Feed per minute) F25(Feed per revolution) (Note 6)
Tool offset		T1/T2	←	←	←
Miscellaneous function (M)		M8	←	←	←
Spindle function (S)		S8	←	←	←
Tool function (T)		T8	←	←	←
2nd miscellaneous function		A8/B8/C8	←	←	←
Subprogram		P8 H5 L4	←	←	←
Fixed cycle	0.001(°) mm/ 0.0001 inch	R+53 Q53 P8 L4	←	←	←
	0.0001(°) mm/ 0.00001 inch	R+44 Q44 P8 L4	←	←	←

(Note 1) α indicates the additional axis address, such as A, B or C.

(Note 2) The no. of digits check for a word is carried out with the maximum number of digits of that address.

(Note 3) Numerals can be used without the leading zeros.

(Note 4) The meanings of the details are as follows:

Example 1 : 08 : 8-digit program number

Example 2 : G21 : Dimension G is 2 digits to the left of the decimal point, and 1 digit to the right.

Example 3 : X+53 : Dimension X uses + or - sign and represents 5 digits to the left of the decimal point and 3 digits to the right.

For example, the case for when the X axis is positioned (G00) to the 45.123 mm position in the absolute value (G90) mode is as follows:

G00 X45.123 ;

- 123: 3 digits below the decimal point
- X45: 5 digits above the decimal point, so it's 00045, but the leading zeros and the mark (+) have been omitted.
- G00: G0 is possible.

(Note 5) If an arc is commanded using a rotary axis and linear axis while inch commands are being used, the degrees will be converted into 0.1 inches for interpolation.

(Note 6) While inch commands are being used, the rotary axis speed will be in increments of 10 degrees.

Example: With the F1. (per-minute-feed) command, this will become the 10 degrees/minute command.

(Note 7) The decimal places below the decimal point are ignored when a command, such as an S command, with an invalid decimal point has been assigned with a decimal point.

(Note 8) This format is the same for the value input from the memory, MDI or setting and display unit.

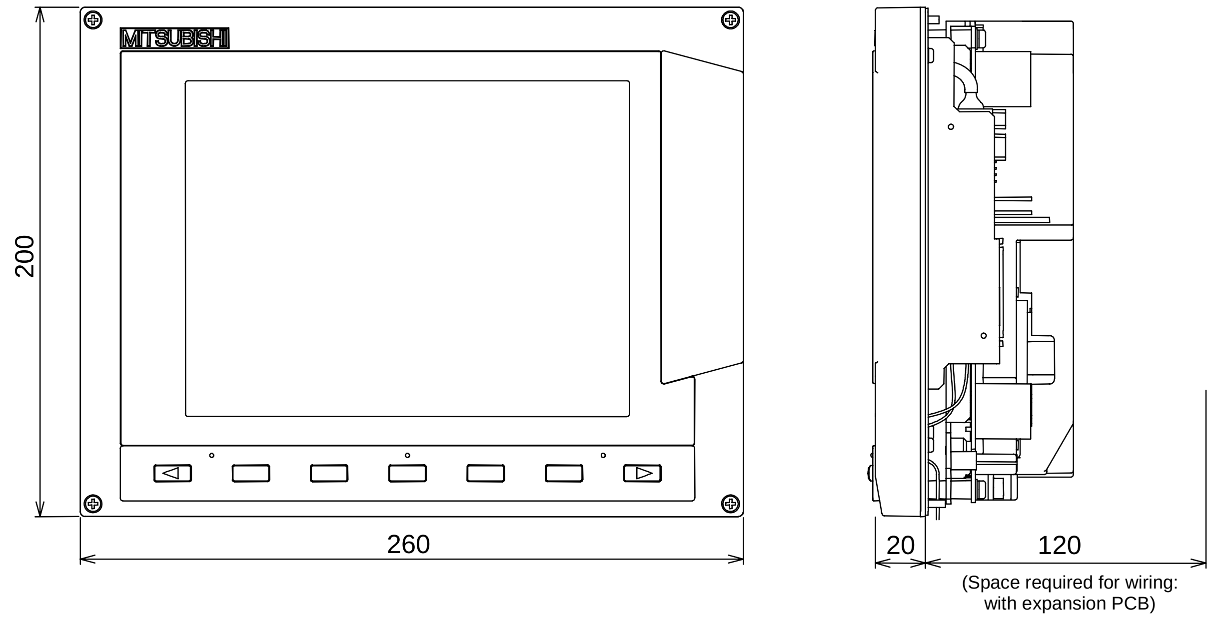
(Note 9) Command the program No. in an independent block. Command the program NO. in the head block of the program.

(Note 10) Least input setting increment "C" (0.0001(°)mm/0.00001inch) and dwell's U address are specifications for E68 system. These cannot be used with E60.

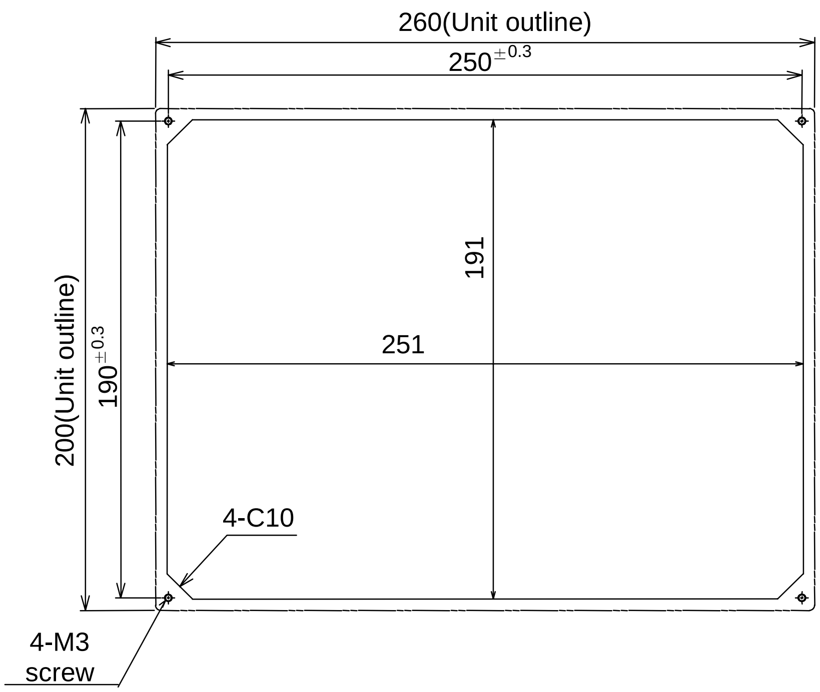
Appendix 3. Outline and Installation Dimension Drawings of Units

Appendix 3.1 E60 Control Unit, Display Unit, Keyboard Unit Outline Drawing

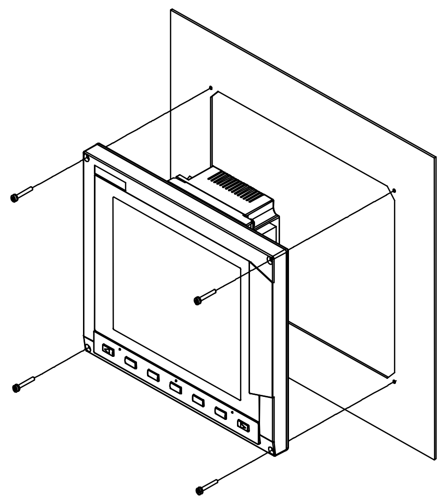
Appendix 3.1.1 Control unit, display unit (FCU6-MU071, FCU6-DUN26) outline drawing



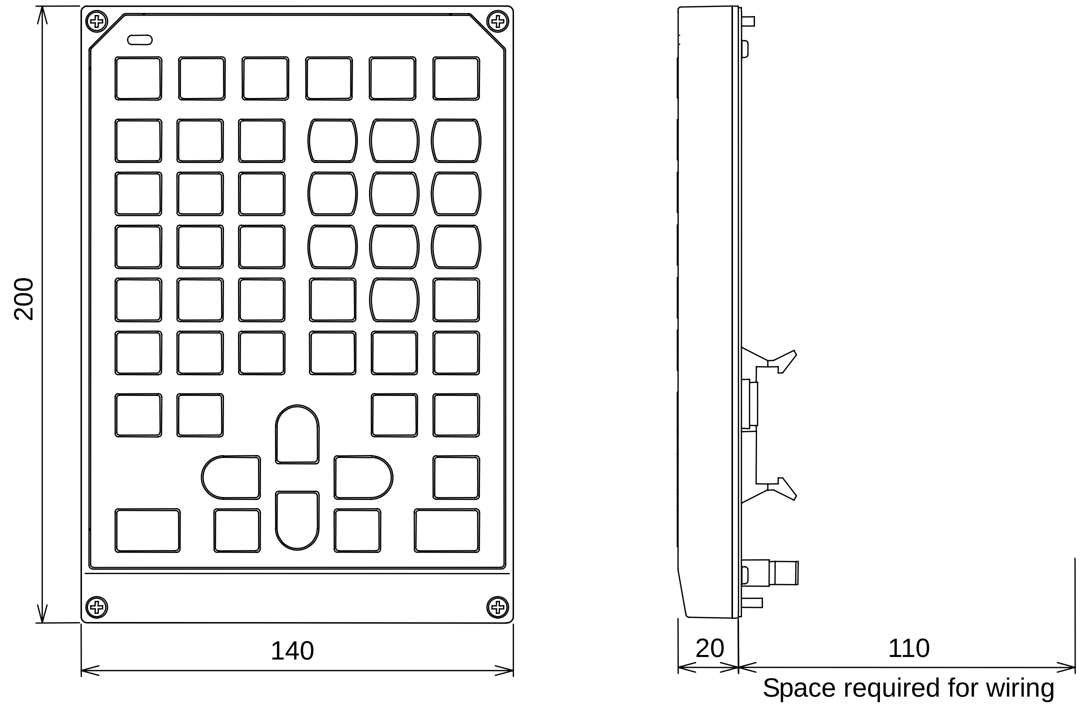
Panel-cut drawing



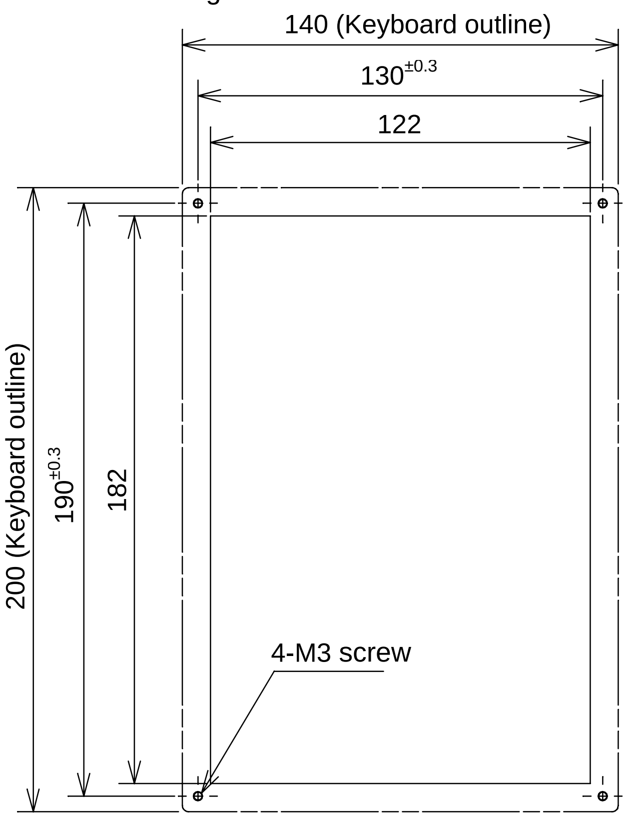
How to fix with screws
from the front of the unit (Reference)



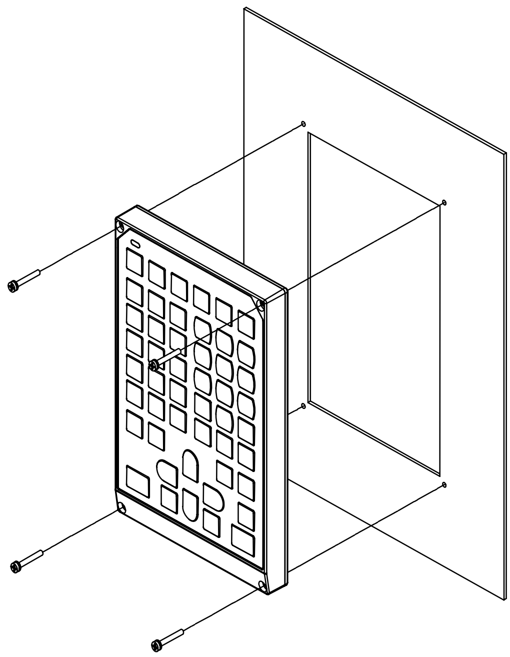
Appendix 3.1.2 Keyboard unit (FCU6-KB024) outline drawing



Panel-cut drawing

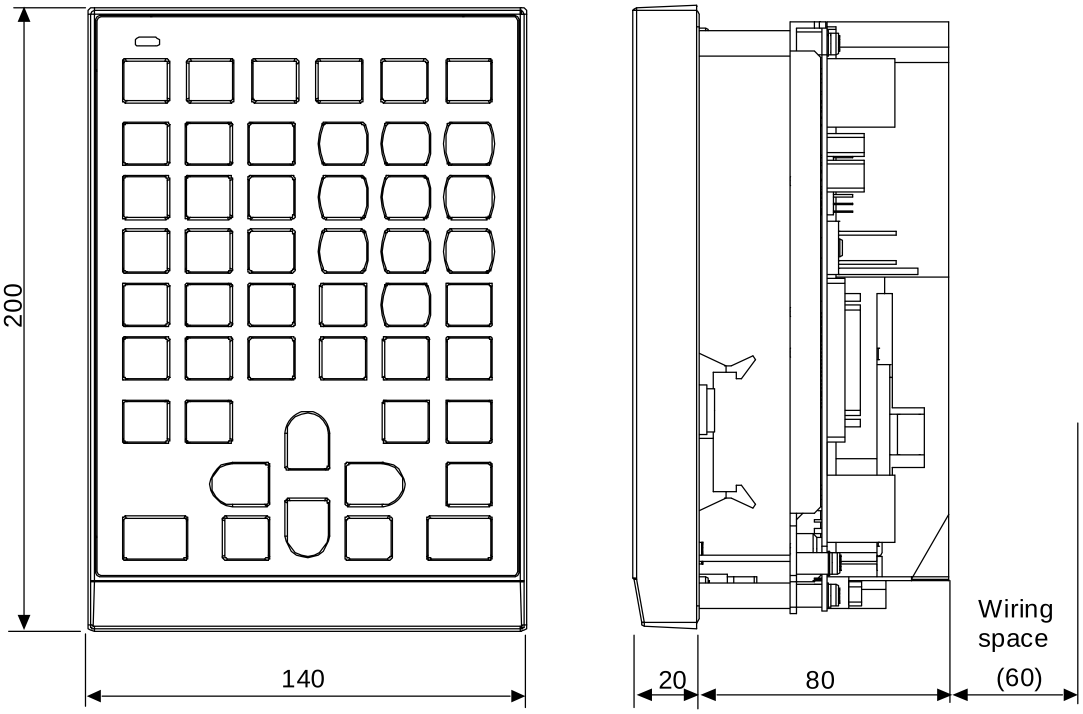


How to fix with screws from the front of the unit (Reference)

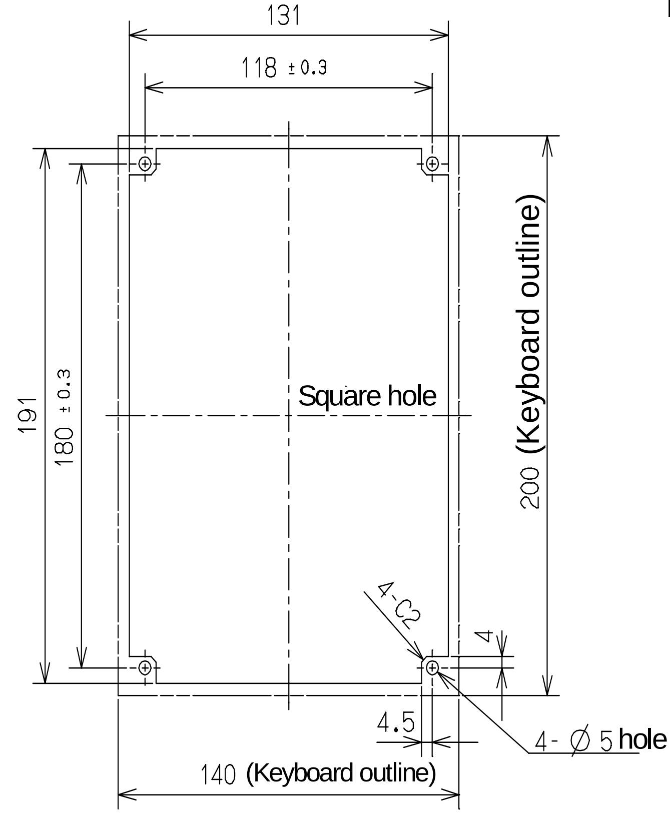


Appendix 3.1.3 Control unit (FCU6-MU071, FCU6-KB071) outline drawing

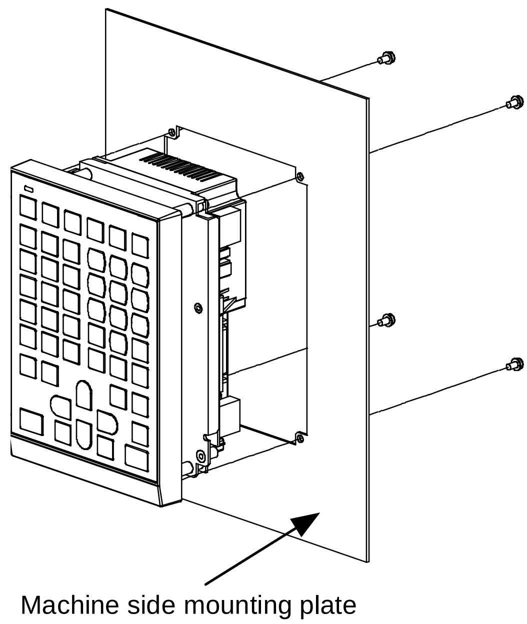
(1) In the case of FCU6-KB071 (Fix with screws from the back of the unit)



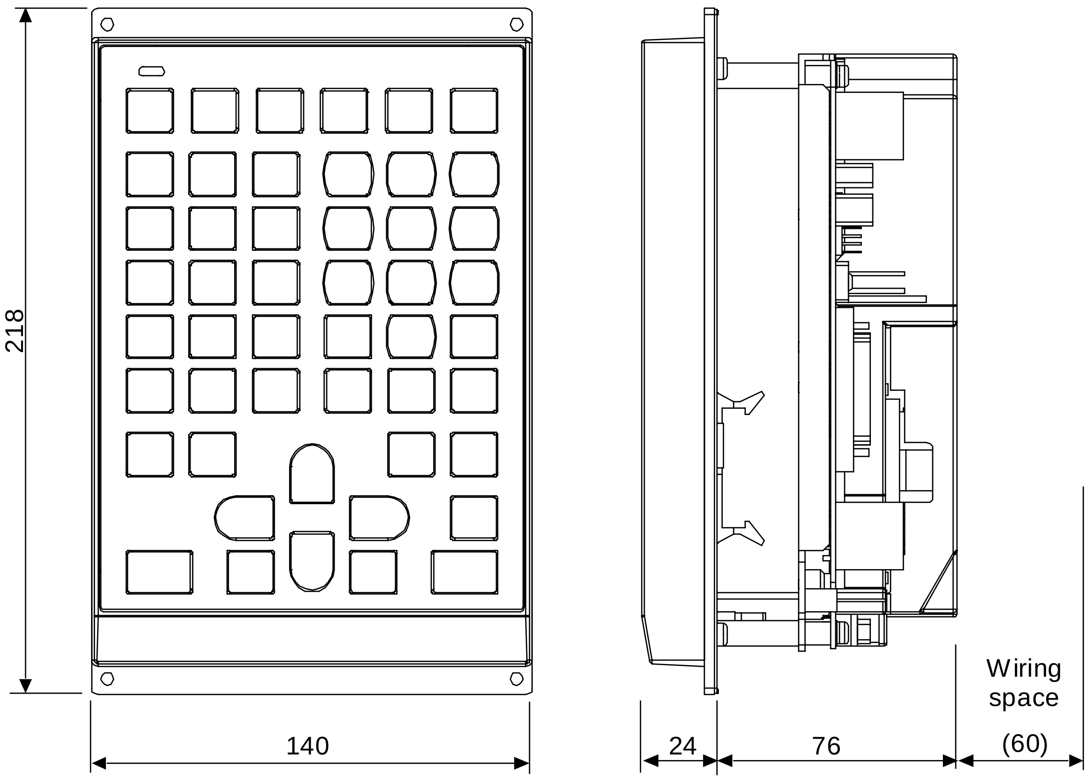
Panel-cut drawing



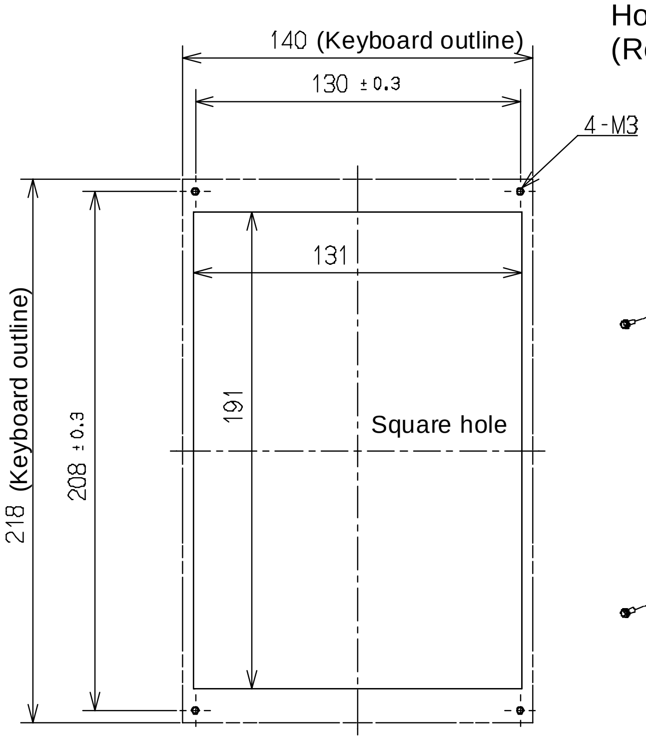
How to fix with screws from the back of the unit (Reference)



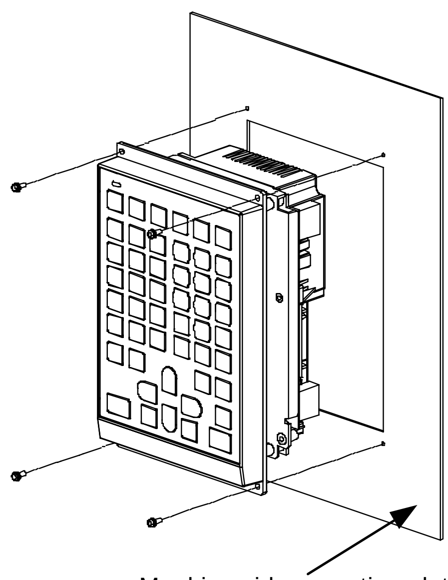
(2) In the case of FCU6-KB071-1 (Fix with screws from the front of the unit)



Panel-cut drawing

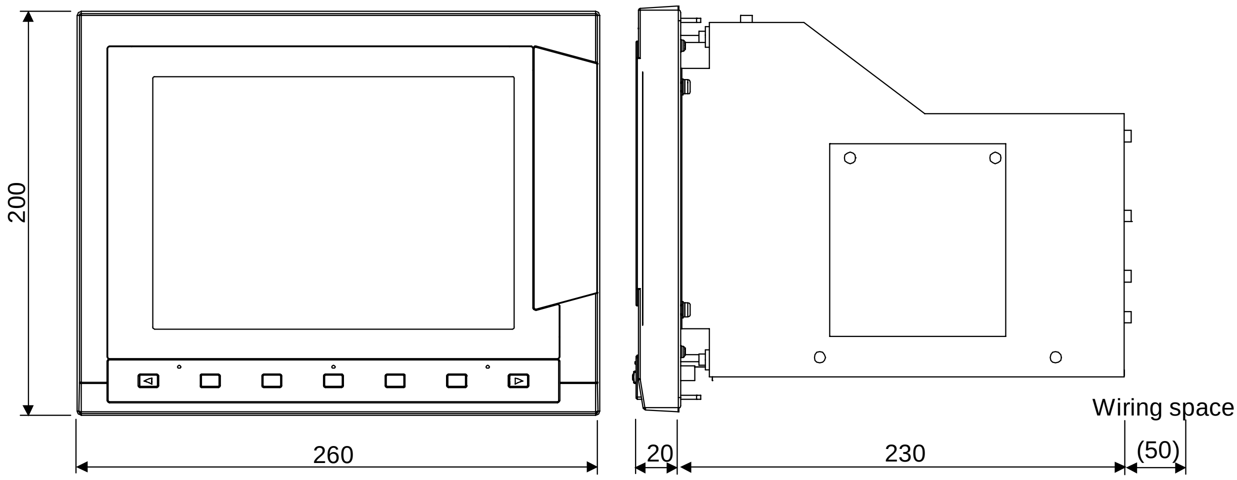


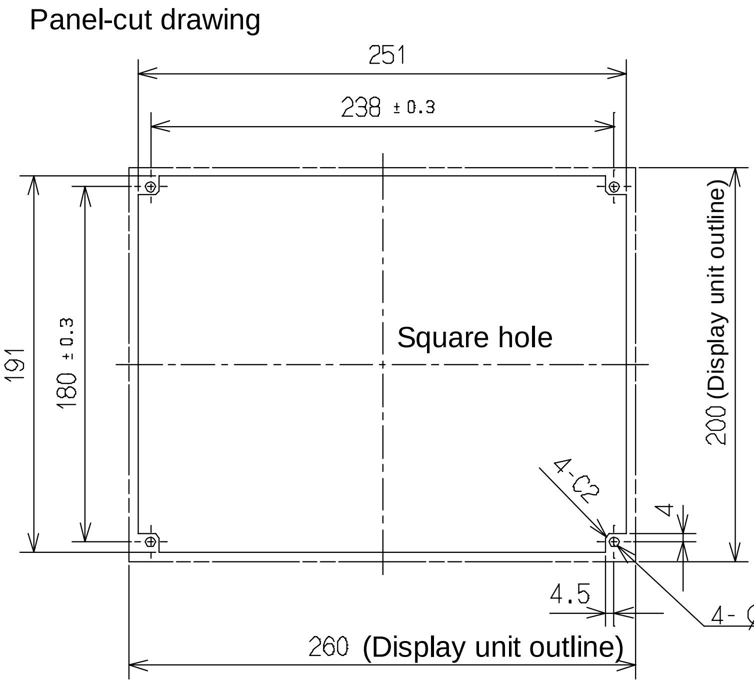
How to fix with screws from the front of the unit (Reference)

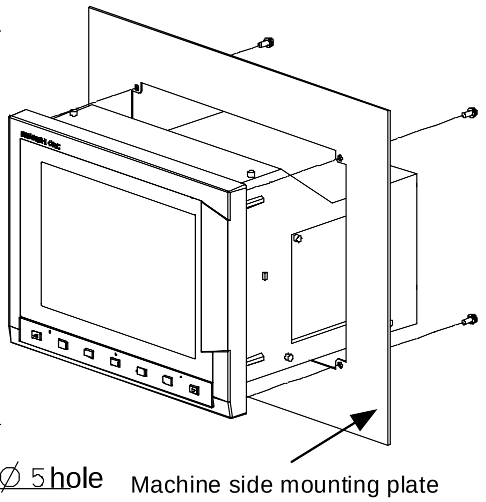


Appendix 3.1.4 Display unit (FCU6-DUE71) outline drawing

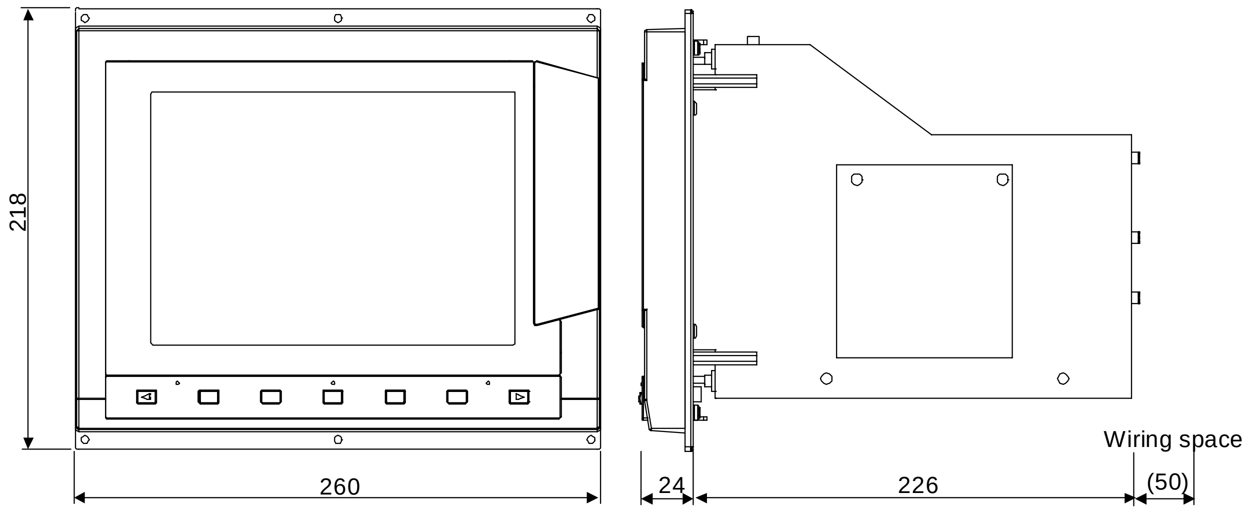
(1) In the case of FCU6-DUE71 (Fix with screws from the back of the unit)



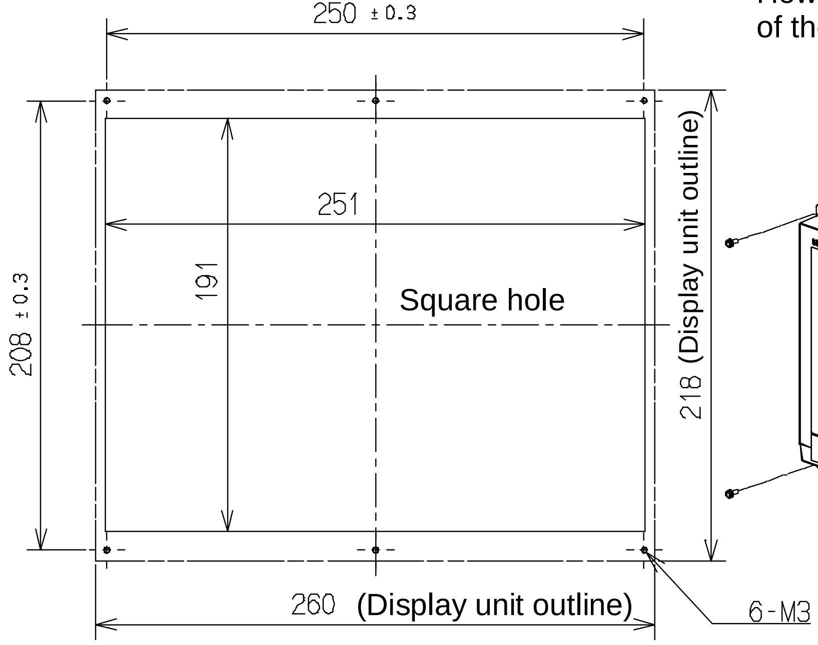




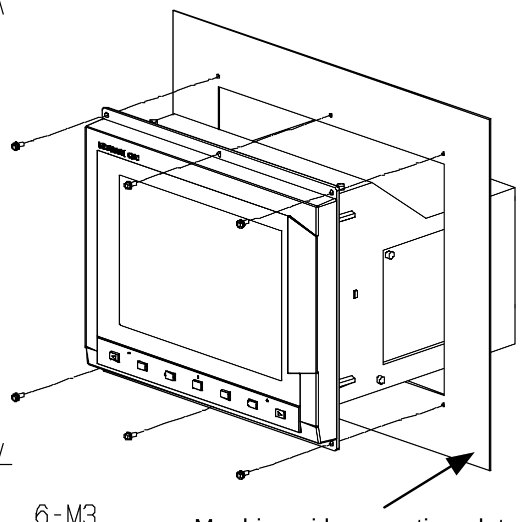
(2) In the case of FCU6-DUE071-1 (Fix with screws from the front of the unit)



Panel-cut drawing

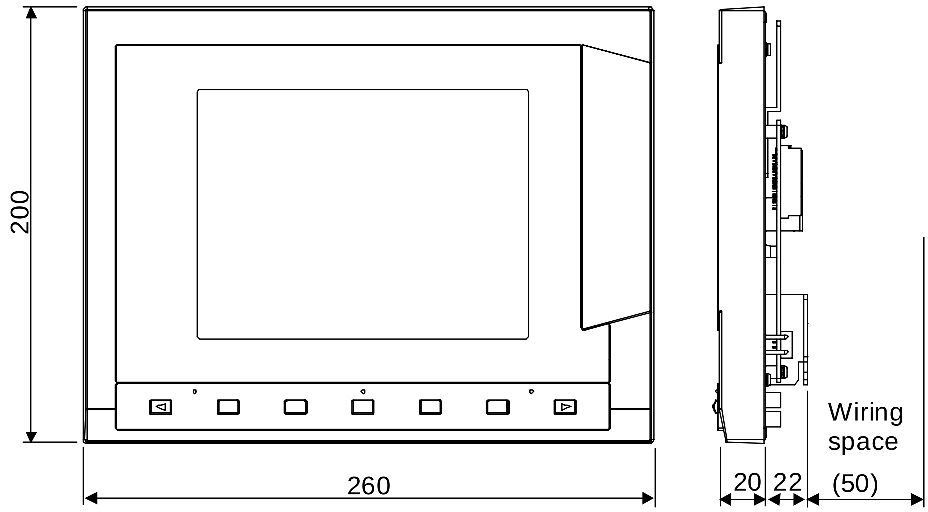


How to fix with screws form the front of the unit (Reference)

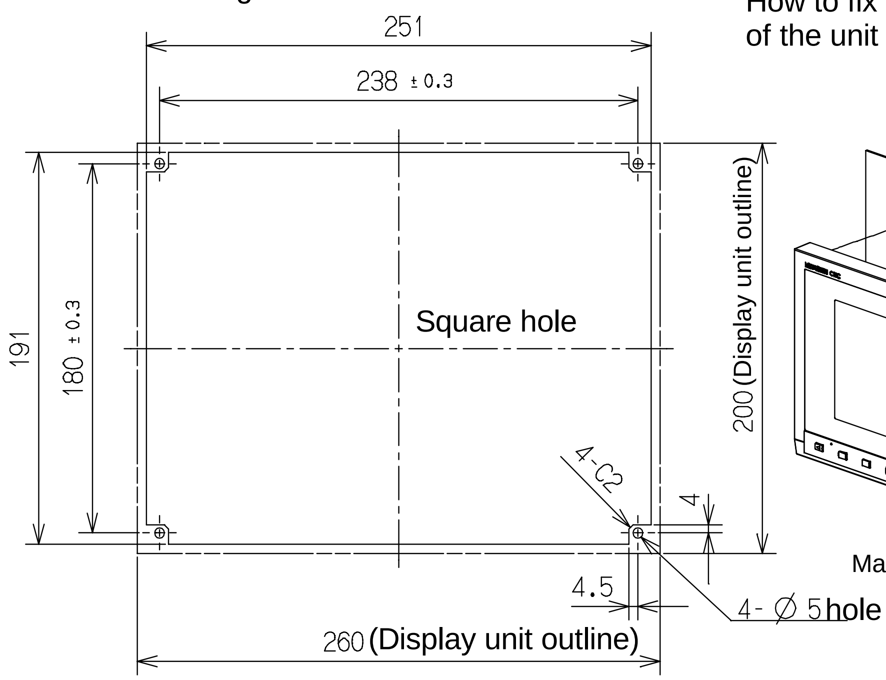


Appendix 3.1.5 Display unit (FCU6-DUT11) outline drawing

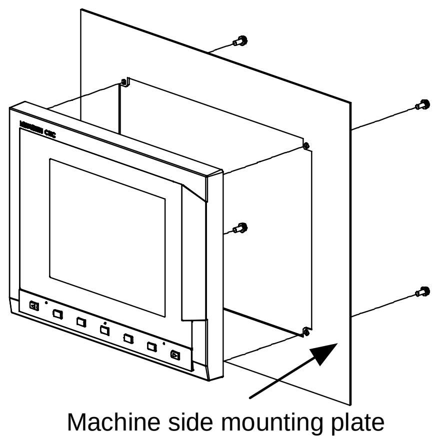
(1) In the case of FCU6-DUT11 (Fix with screws from the back of the unit)



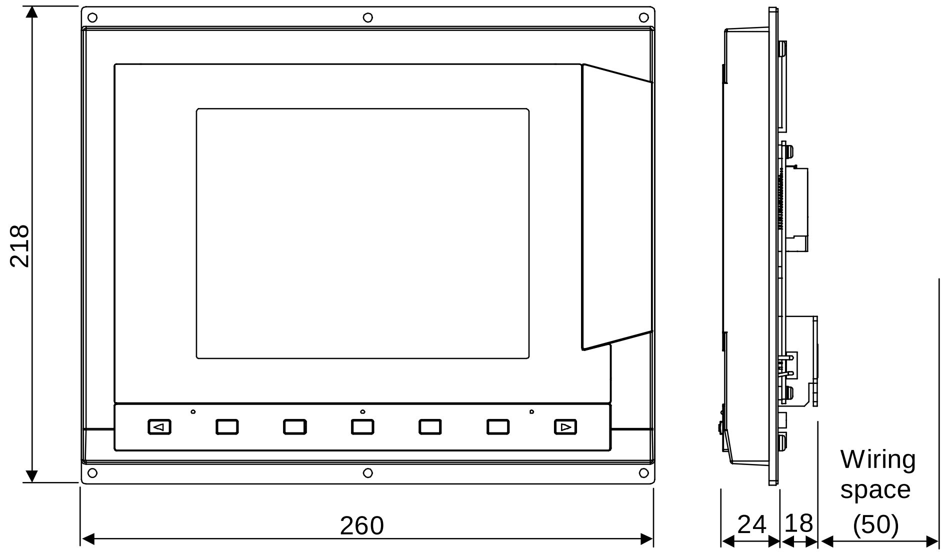
Panel-cut drawing



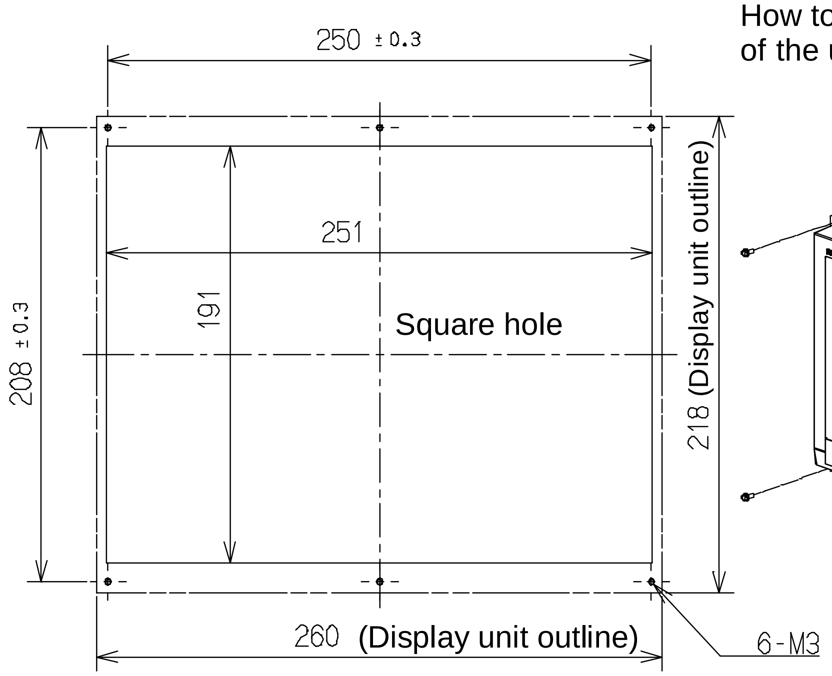
How to fix with screws from the back of the unit (Reference)



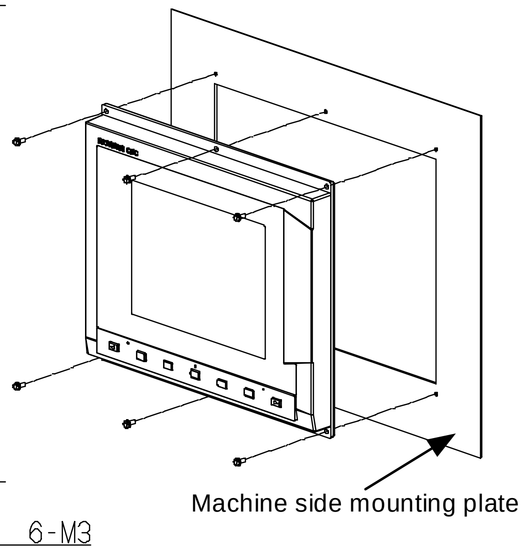
(2) In the case of FCU6-DUT11-1 (Fix with screws from the back of the unit)



Panel-cut drawing

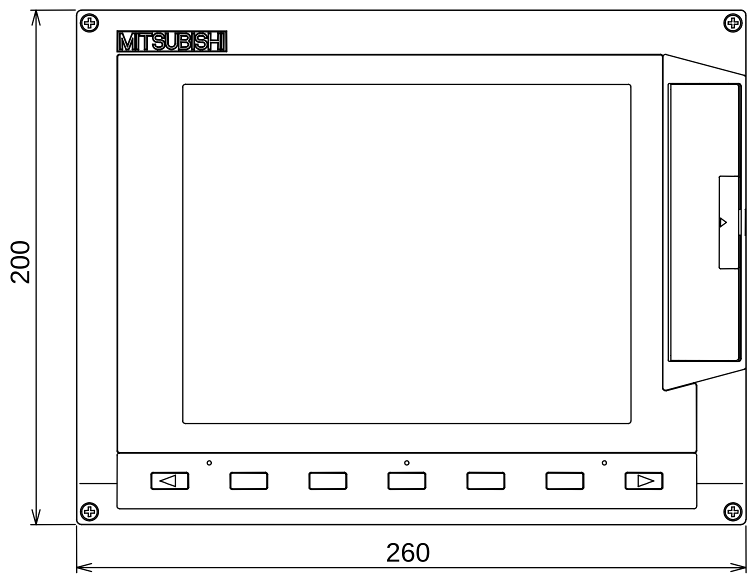


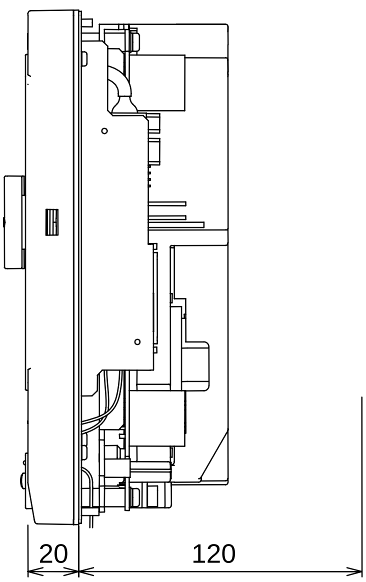
How to fix with screws from the front of the unit (Reference)



Appendix 3.2 E68 Control Unit, Display Unit, Keyboard Unit Outline Drawing

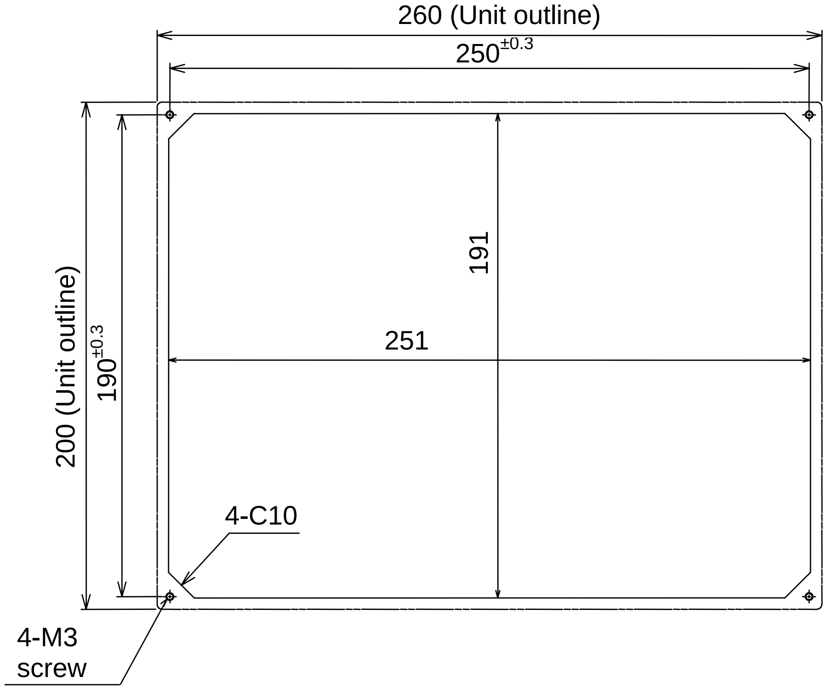
Appendix 3.2.1 Control unit, display unit (FCU6-MU072,FCU6-DUN24) outline drawing



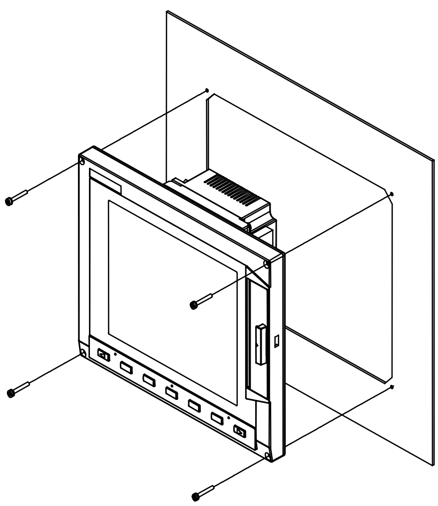


(Space required for wiring:
with expansion PCB)

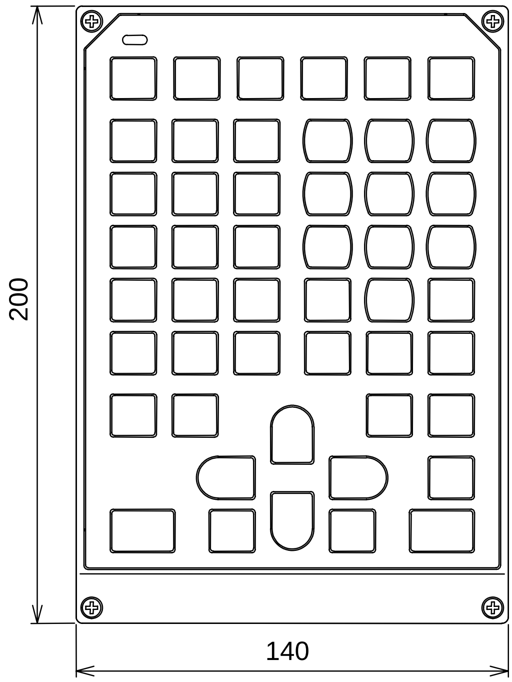
Panel-cut drawing

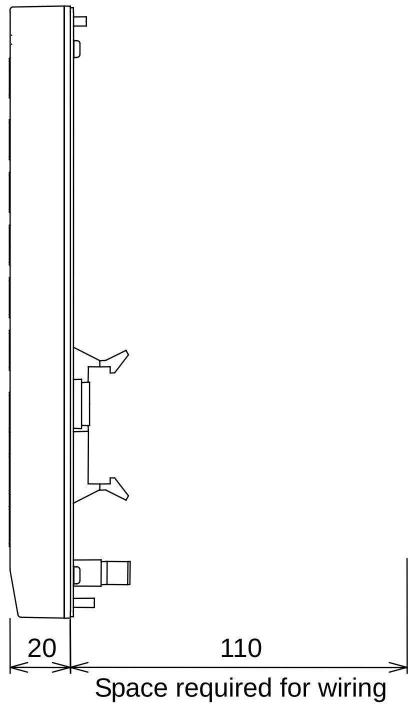


How to fix with screws from the front of the unit (Reference)

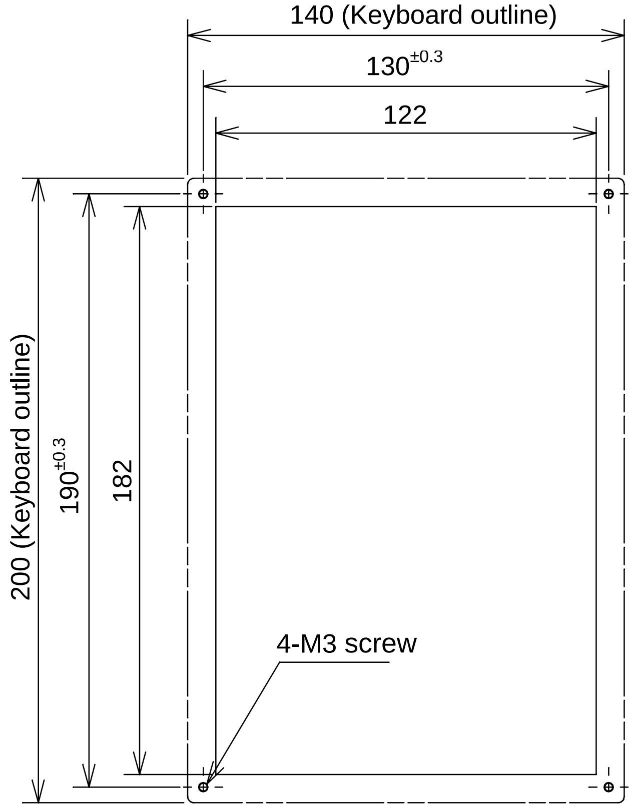


Appendix 3.2.2 Keyboard unit (FCU6-KB024) outline drawing

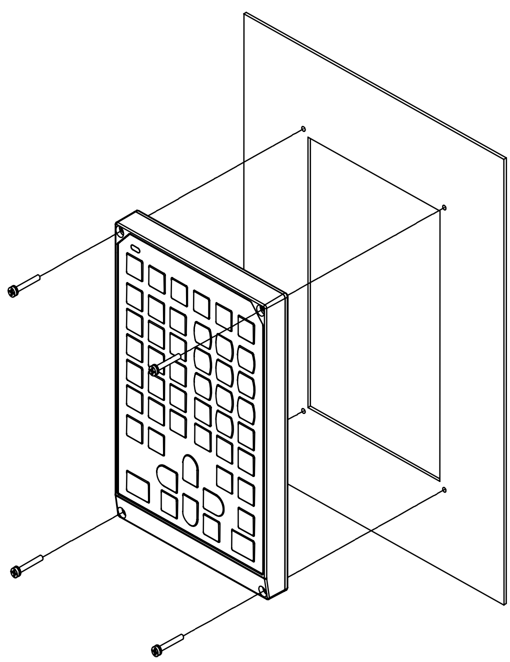




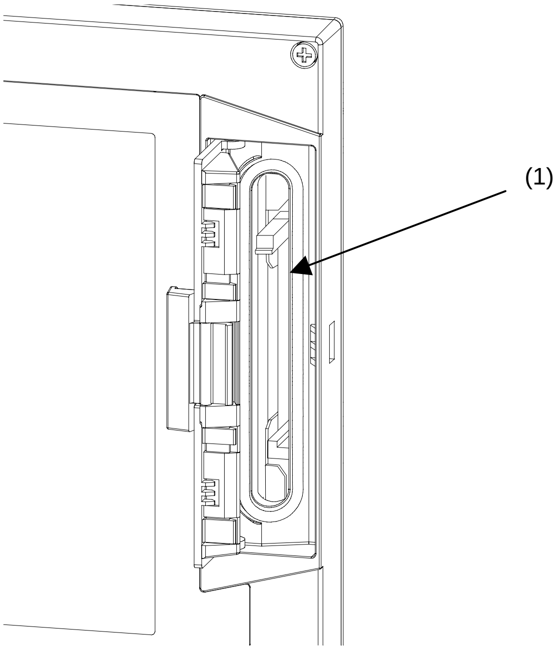
Panel-cut drawing



How to fix with screws from the front of the unit (Reference)



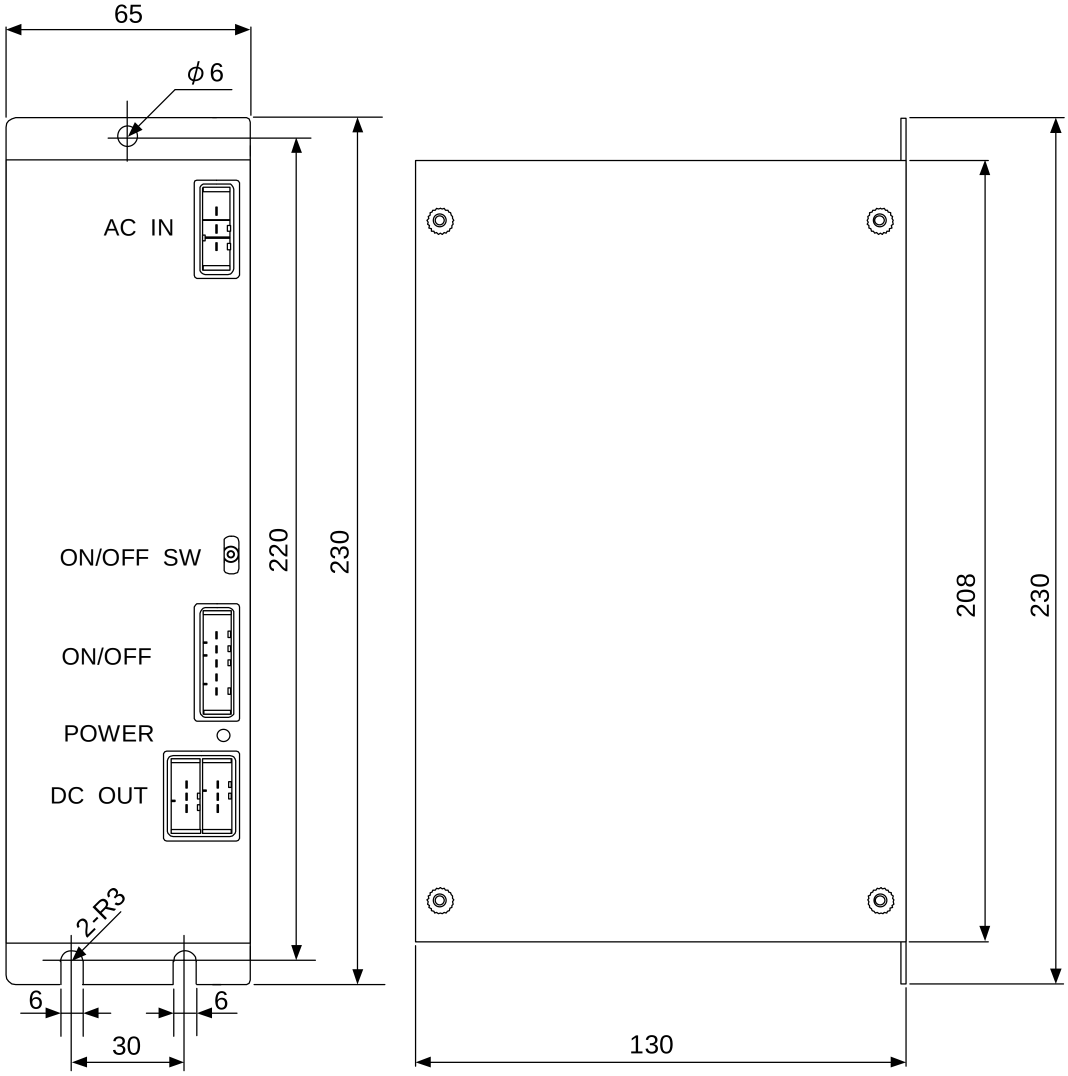
Appendix 3.2.3 Front IC card I/F unit (FCU6-EP105-1) outline drawing



Explanation of the front IC card I/F unit/connector functions

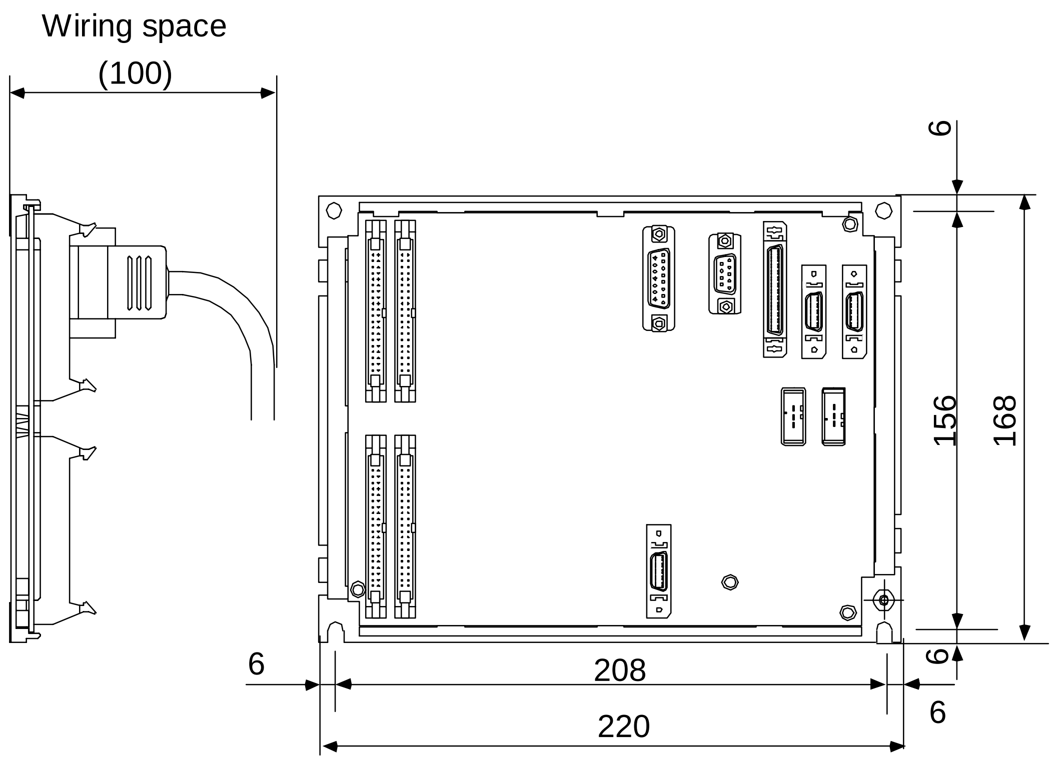
No.	Connector	Function
(1)	Memory Card I/F	PC Card Standard ATA-compliant CF card TYPEI, TYPEII only (5VDC : max 220mA)

Appendix 3.3 External Power Supply Unit (PD25) Outline Drawing

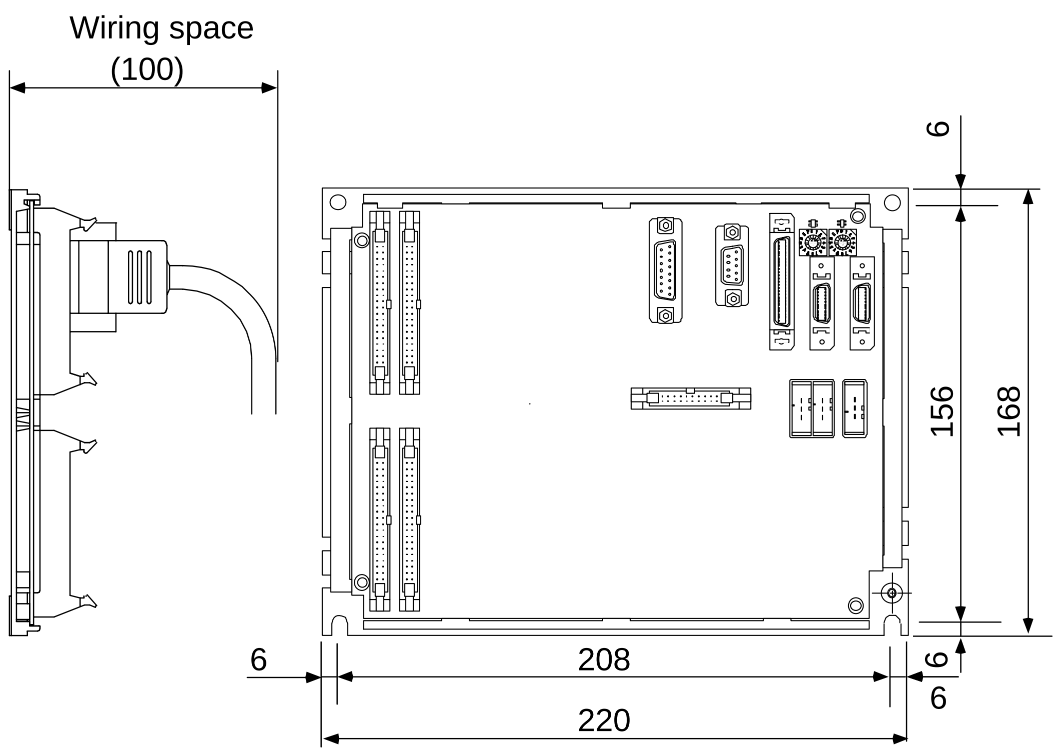


Appendix 3.4 Base I/O Unit Outline Drawing

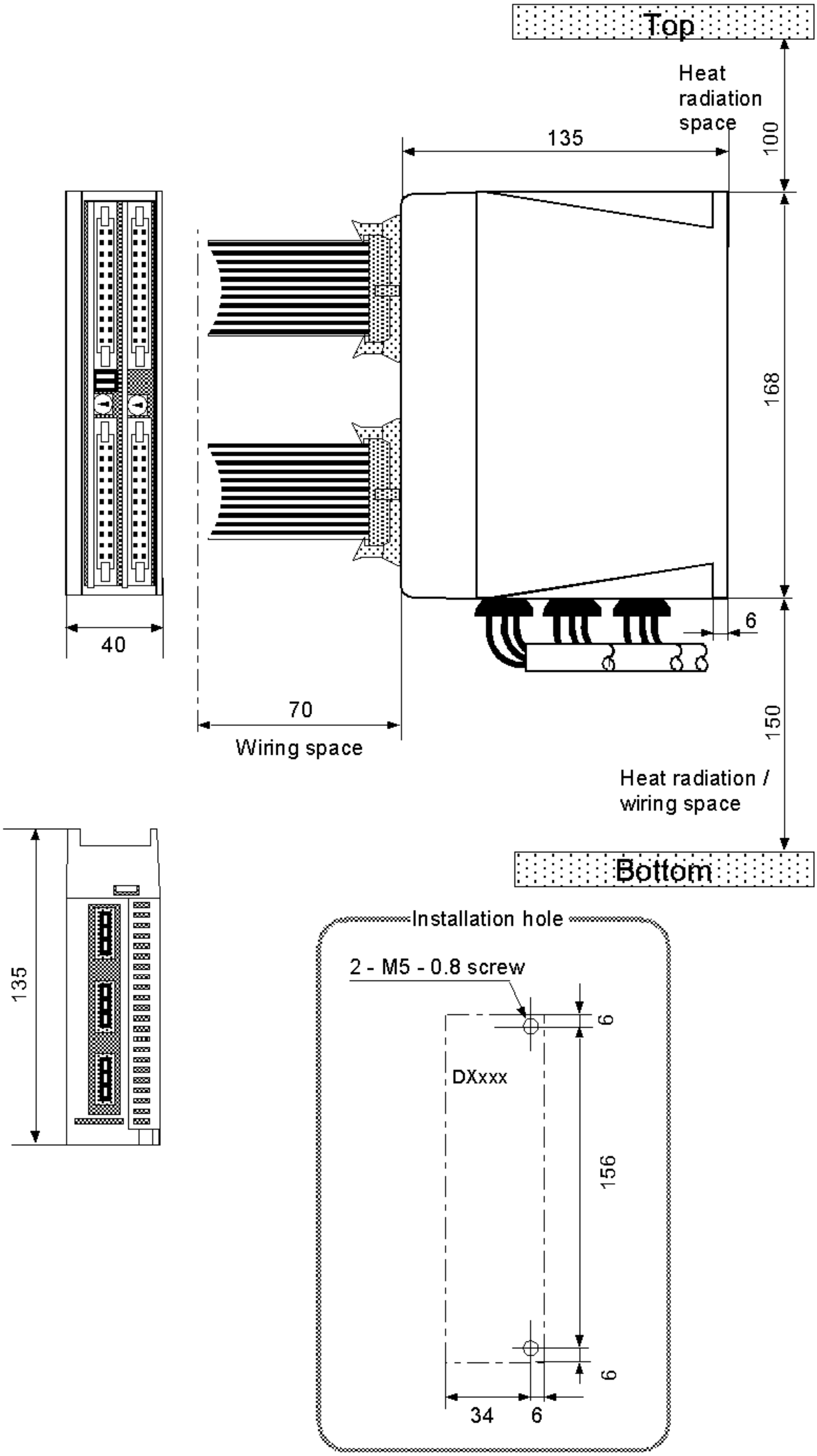
Appendix 3.4.1 FCU6-HR341/HR351 outline drawing



Appendix 3.4.2 FCU6-DX220/DX221 outline drawing



Appendix 3.5 Remote I/O Unit (FCUA-DX1xx) Outline Drawing



EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68 M	E68 L	E60 M	E60 L
1 Control axes								
	1 Control axes							
		1	Number of basic control axes (NC axes)		○ 3	○ 2	○ 3	○ 2
		2	Max. number of control axes (NC axes + Spindles + PLC axes + Auxiliary axes)		8	8	5	5
			Max. number of axes (NC axes + Spindles + PLC axes)		6	6	5	5
			Max. number of servo axes (NC axes + PLC axes)		6	6	4	4
			Max. number of NC axes (in total for all the part systems)		4	4	3	3
			Max. number of spindles (parenthesis means max. number in a part system)		2	2	1	1
			Max. number of PLC axes		2	2	1	1
			Max. number of auxiliary axes (MR-J2-CT)		4	4	1	1
		3	Number of simultaneous contouring control axes		4	4	3	3
		4	Max. number of NC axes in a part system		4	4	3	3
	2 Control part system							
		1	Standard number of part systems		1	1	1	1
		2	Max. number of part systems		○1	○1	○1	○1
	3 Control axes and operation modes							
		1	Tape (RS-232C input) mode		○	○	○	○
		2	Memory mode		○	○	○	○
		3	MDI mode		○	○	○	○
		5	IC card mode					
			2	Front IC card	○	-	-	-
2 Input command								
	1 Data increment							
		1	Data increment and parameter					
		2	Least input increment					
		3	Least command increment					
				Least command increment 1um	○	○	○	○
				Least command increment 0.1um	○	○	-	-
	2 Unit system							
		1	Inch/Metric changeover		○	○	○	○
	3 Program format							
		1	Character code		○	○	○	○
		2	Program format					
			1	Format 1 for Lathe	-	○	-	○
			3	Special format for Lathe	-	○	-	○
			4	Format 1 for Machining center	○	-	○	-
	4 Command value							
		1	Decimal point input I, II		○	○	○	○
		2	Absolute/Incremental command		○	○	○	○
		3	Diameter/Radius designation		-	○	-	○
	5 Command value and setting value range							
		1	Command value and setting value range		○	○	○	○
3 Positioning/Interpolation								
	1 Positioning							
		1	Positioning		○	○	○	○
		2	Unidirectional positioning		○	-	○	–
	2 Linear/Circular interpolation							
		1	Linear interpolation		○	○	○	○
		2	Circular interpolation (Center/Radius designation)		○	○	○	○
		3	Helical interpolation		○	○	○	–
		5	Cylindrical interpolation		○	○	-	-
		6	Polar coordinate interpolation		-	○	-	-

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68		E60	
					M	L	M	L
4 Feed								
	1 Feed rate							
		1	Rapid traverse rate (m/min)		1000	1000	1000	1000
		2	Cutting feed rate (m/min)		1000	1000	1000	1000
		3	Manual feed rate (m/min)		1000	1000	1000	1000
	2 Feed rate input methods							
		1	Feed per minute		○	○	○	○
		2	Feed per revolution		○	○	○	○
		4	F 1-digit feed		○	○	○	○
	3 Override							
		1	Rapid traverse override		○	○	○	○
		2	Cutting feed override		○	○	○	○
		3	2nd cutting feed override		○	○	○	○
		4	Override cancel		○	○	○	○
	4 Acceleration/Deceleration							
		1	Automatic acceleration/deceleration after interpolation		○	○	○	○
		2	Rapid traverse constant inclination acceleration/deceleration		○	○	○	○
	5 Thread cutting							
		1	Thread cutting (Lead/Thread number designation)		○	○	○	○
		2	Variable lead thread cutting		-	○	-	○
		3	Synchronous tapping					
			1	Synchronous tapping cycle	○	○	△	△
			2	Pecking tapping cycle	○	-	-	-
			3	Deep-hole tapping cycle	○	-	-	-
		4	Chamfering		-	○	-	○
	6 Manual feed							
		1	Manual rapid traverse		○	○	○	○
		2	Jog feed		○	○	○	○
		3	Incremental feed		○	○	○	○
		4	Handle feed		○(2)	○(2)	○(2)	○(2)
		5	Manual feed rate B		○	○	○	○
	7 Dwell							
		1	Dwell (Time-based designation)		○	○	○	○
5 Program memory/editing								
	1 Memory capacity							
		1	Memory capacity (number of programs stored)					
			600m (400 programs)		○	○	○	○
	2 Editing method							
		1	Program editing		○	○	○	○
		2	Background editing		○	○	○	○
		3	Buffer correction		○	○	○	○
		4	Word editing		○	○	○	○

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68		E60	
					M	L	M	L
6 Operation and display								
	1 Structure of operation/display panel							
				7.2-type LCD monochrome display	-	-	□	□
				9-type CRT monochrome display	-	-	□	□
				8.4-type LCD (TFT) color display	○	○	□	□
	2 Operation methods and functions							
		1		Memory switch (PLC switch)	○	○	○	○
	3 Display methods and contents							
		1		Status display	○	○	○	○
		2		Position display	○	○	○	○
		3		Program running status display	○	○	○	○
		4		Setting and display	○	○	○	○
		5		MDI data setting and display	○	○	○	○
		7		Clock	○	○	○	○
		8		Hardware/Software configuration display	○	○	○	○
		9		Integrated time display	○	○	○	○
		10		Available languages	○13	○13	○13	○13
		11		Additional languages				
			1	Japanese	○	○	○	○
			2	English	○	○	○	○
			3	German	○*	○*	○*	○*
			4	Italian	○*	○*	○*	○*
			5	French	○*	○*	○*	○*
			6	Spanish	○*	○*	○*	○*
			7	Chinese				
				Traditional Chinese characters	○*	○*	○*	○*
				Simplified Chinese characters	○	○	○	○
			8	Korean	○*	○*	○*	○*
			9	Portuguese	○*	○*	○*	○*
			10	Hungarian	○*	○*	○*	○*
			11	Dutch	○*	○*	○*	○*
			12	Swedish	○*	○*	○*	○*
		12		Screen saver, backlight OFF	○	○	○	○
		13		Screen deletion	○	○	○	○
7 Input/Output functions and devices								
	1 Input/Output data							
		1		Machining program input/output	○	○	○	○
		2		Tool offset data input/output	○	○	○	○
		3		Common variable input/output	○	○	○	○
		4		Parameter input/output	○	○	○	○
		5		History data output	○	○	○	○
		6		Remote program input	○	○	-	-
		7		System configuration data output	○	○	○	○
	2 Input/Output I/F							
		1		RS-232C I/F	○	○	○	○
		2		IC card I/F				
			2	I/F for front IC card	○	○	-	-
	3 Computer link							
		1		Computer link B	○	○	-	-

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68		E60	
					M	L	M	L
8	Spindle, Tool and Miscellaneous functions							
	1	Spindle functions (S)						
	1	Command/Output						
		1	Spindle functions		○	○	○	○
		2	Spindle serial I/F		○	○	△	△
		3	Spindle analog I/F		□	□	○	○
		4	Coil change		○	○	○	○
		5	Automatic coil change		○	○	○	○
	2	Speed control						
		1	Constant surface speed control		○	○	○	○
		2	Spindle override		○	○	○	○
		3	Multiple-spindle control					
			1	Multiple-spindle control I	○	○	-	-
			2	Multiple-spindle control II	○	○	-	-
	3	Position control						
		1	Spindle orientation		○	○	△	△
		2	Spindle position control (Spindle/C axis control)		○	○	△	△
		3	Spindle synchronization					
			1	Spindle synchronization I	-	○	-	-
			2	Spindle synchronization II	○	○	-	-
		11	Spindle holding power improvement		○	○	△	△
	2	Tool functions (T)						
		1	Tool functions		○	○	○	○
	3	Miscellaneous functions (M)						
		1	Miscellaneous functions		○	○	○	○
		2	Multiple M codes in 1 block		○	○	○	○
		3	M code independent output		○	○	○	○
		4	Miscellaneous function finish		○	○	○	○
	4	2nd miscellaneous function (B)						
		1	2nd miscellaneous function		○	○	○	○
9	Tool compensation							
	1	Tool length/poistion offset						
		1	Tool length offset		○	○	○	○
		2	Tool position offset		○	-	-	-
		3	Tool offset for additional axes		-	○	-	○
	2	Tool radius						
		1	Tool radius compensation		○	-	○	-
		3	Tool nose radius compensation (G40/41/42)		-	○	-	○
		4	Automatic decision of nose radius compensation direction (G46/40)		-	○	-	○
	3	Tool offset amount						
		1	Number of tool offset sets					
			3	80	-	○	-	○
			5	200	-	-	○	-
			6	400	○	-	-	-
		2	Offset memory					
			1	Tool shape/wear offset amount	○	○	○	○

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class				E68 M	E68 L	E60 M	E60 L
10 Coordinate system							
	1	Coordinate system type and setting					
		1	Machine coordinate system	○	○	○	○
		2	Coordinate system setting	○	○	○	○
		3	Automatic coordinate system setting	○	○	○	○
		4	Workpiece coordinate system selection (6 sets)	○	○	○	○
		5	Extended workpiece coordinate system selection (48 sets) G54.1P1 to P48	○	○	○	○
		6	Workpiece coordinate system preset (G92.1)	-	○	-	-
		7	Local coordinate system	○	○	○	○
		8	Coordinate system for rotary axis	○	○	○	○
		9	Plane selection	○	○	○	○
		10	Origin set	○	○	○	○
		11	Counter set	○	○	○	○
	2	Return					
		1	Manual reference position return	○	○	○	○
		2	Automatic 1st reference position return	○	○	○	○
		3	2nd, 3rd, 4th reference position return	○	○	○	○
		4	Reference position verification	○	○	○	○
		5	Absolute position detection	△	△	△	△
		6	Tool exchange position return	○	○	○	○
		7	C axis reference position return	-	○	-	△
11 Operation support functions							
	1	Program control					
		1	Optional block skip	○	○	○	○
		3	Single block	○	○	○	○
	2	Program test					
		1	Dry run	○	○	○	○
		2	Machine lock	○	○	○	○
		3	Miscellaneous function lock	○	○	○	○
		4	Graphic check	○	○	○	○
		5	Graphic trace	○	○	○	○
	3	Program search/start/stop					
		1	Program search	○	○	○	○
		2	Sequence number search	○	○	○	○
		3	Collation stop	○	○	-	-
		4	Program restart	○	○	○	○
		5	Automatic operation start	○	○	○	○
		6	NC reset	○	○	○	○
		7	Feed hold	○	○	○	○
		8	Search & Start	○	○	○	○
	4	Interrupt operation					
		1	Manual interruption	○	○	○	○
		2	Automatic operation handle interruption	○	○	○	○
		3	Manual absolute mode ON/OFF	○	○	○	○
		4	Thread cutting cycle retract	-	○	-	-
		5	Tapping retract	○	○	○	○
		6	Manual numerical value command	○	○	○	○
		8	MDI interruption	○	○	○	○
		9	Simultaneous operation of manual and automatic modes	○	○	○	○
		10	Simultaneous operation of JOG and handle modes	○	○	○	○
		11	Reference position retract	○	○	○	○
		14	PLC interruption	○	○	-	-

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class	E68 M	E68 L	E60 M	E60 L
12 Program support functions				
1 Machining method support functions				
1 Program				
1 Subprogram control	○8 layers	○8 layers	○8 layers	○8 layers
3 Scaling	○	-	○	-
2 Macro program				
1 User macro	○4 layers	○4 layers	○4 layers	○4 layers
2 Machine tool builder macro				
1 Machine tool builder macro SRAM	○	○	-	-
3 Macro interruption	○	○	○	○
4 Variable command				
2 200 sets	-	-	○	○
3 300 sets	○	○	-	-
3 Fixed cycle				
1 Fixed cycle for drilling	○	○	○	○
2 Special fixed cycle	○	-	○	-
3 Fixed cycle for turning machining	-	○	-	○
4 Multiple repetitive fixed cycle for turning machining	-	○	-	○
5 Multiple repetitive fixed cycle for turning machining (Type II)	-	○	-	-
6 Small-diameter deep-hole drilling cycle	-	-	-	-
7 Fixed cycle for drilling (Type II)	-	○	-	○
4 Mirror image				
1 Mirror image by parameter setting	○	○	-	-
2 External input mirror image	○	○	-	-
3 G code mirror image	○	-	○	-
4 Mirror image for facing tool posts	-	-	-	-
5 T code mirror image for facing tool posts	-	-	-	-
5 Coordinate system operation				
1 Coordinate rotation by program	○	-	○	-
6 Dimension input				
1 Corner chamfering/Corner R	○	○	○	○
2 Linear angle command	○	○	○	○
3 Geometric command	○	○	○	○
4 Polar coordinate command	○	-	-	-
7 Axis control				
1 High-speed machining				
3 High-speed machining mode III	△	-	-	-
2 Chopping				
1 Chopping	○	○	-	-
5 Circular cutting	○	-	○	-
9 Data input by program				
1 Parameter input by program	○	○	○	○
2 Compensation data input by program	○	○	○	○
10 Machining modal				
1 Tapping mode	○	○	○	○
2 Cutting mode	○	○	○	○
2 Machining accuracy support functions				
1 Automatic corner override	○	○	○	○
2 Deceleration check				
1 Exact stop check mode	○	○	○	○
2 Exact stop check	○	○	○	○
3 Error detect	○	○	○	○
4 Programmable inposition check	○	○	○	○
3 High-accuracy control				
High-accuracy control (G61.1)	○	-	-	-
High-accuracy control (G08)	○	-	-	-
3 Programming support functions				
1 Playback	○	○	○	○
2 Address check	○	○	○	○

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68 M	E68 L	E60 M	E60 L
13 Machine accuracy compensation								
	1 Static accuracy compensation							
		1	Backlash compensation		○	○	○	○
		2	Memory-type pitch error compensation		○	○	○	○
		3	Memory-type relative position error compensation		○	○	○	○
		4	External machine coordinate system compensation		○	○	○	○
		9	Spindle backlash compensation		○	○	-	-
	2 Dynamic accuracy compensation							
		1	Smooth high-gain control (SHG control)		○	○	○	○
		2	Dual feedback		△	△	-	-
		3	Lost motion compensation		○	○	○	○
14 Automation support functions								
	1 External data input							
		1	External search		○	○	○	○
		2	External workpiece coordinate offset		○	○	○	○
		3	External tool offset		○	○	○	○
	2 Measurement							
		1	Skip					
			1	Skip	○	○	○	○
			2	Multiple-step skip	○	○	○	○
			4	PLC skip	○	○	-	-
		5	Automatic tool length measurement		○	○	○	○
		6	Manual tool length measurement 1		○	○	○	○
		7	Manual tool length measurement 2		○	○	-	○
		8	Workpiece coordinate offset measurement		○*	○	-	-
		9	Workpiece position measurement		○	-	-	-
	3 Monitoring							
		1	Tool life management					
			Tool life management I		○	○	○	○
			Tool life management II		○	○	○	○
		2	Number of tool life management sets					
			20/40/80 sets		-	○80	-	○80
			100/200 sets		○200	-	○100	-
		3	Display of integrated time/number of parts		○	○	○	○
		4	Load meter		○	○	○	○
		5	Position switch		○24	○24	○24	○24
		12	Synchronous error observation		○	○	-	-
	5 Others							
		1	Programmable current limitation		○	○	-	-

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class					E68		E60	
					M	L	M	L
15	Safety and maintenance							
	1	Safety switches						
		1	Emergency stop		○	○	○	○
		2	Data protection key		○	○	○	○
	2	Display for ensuring safety						
		1	NC warning display		○	○	○	○
		2	NC alarm display		○	○	○	○
		3	Operation stop cause		○	○	○	○
		4	Emergency stop cause		○	○	○	○
		5	Temperature detection		○	○	○	○
	3	Protection						
		1	Stroke end (Over travel)		○	○	○	○
		2	Stored stroke limit					
			1	Stored stroke limit I/II	○	○	○	○
			2	Stored stroke limit IB	○	○	-	-
			3	Stored stroke limit IIB	○	○	-	-
			4	Stored stroke limit IC	○	○	-	-
		4	Chuck/Tailstock barrier check		-	○	-	○
		5	Interlock		○	○	○	○
		6	External deceleration		○	○	○	○
		8	Door interlock					
			1	Door interlock I	○	○	○	○
			2	Door interlock II	○	○	○	○
		9	Parameter lock		○	○	○	○
		10	Program protect (Edit lock B, C)		○	○	○	○
		11	Program display lock		○	○	○	○
	4	Maintenance and troubleshooting						
		1	History diagnosis		○	○	○	○
		2	Setup/Monitor for servo and spindle		○	○	○	○
		3	Data sampling		○	○	○	○
		4	Waveform display		○	○	○	○
		5	Machine operation history monitor		○	○	○	○
		6	NC data backup					
				RS-232C	○	○	○	○
				Cassette memory	○	○	○	○
				IC card	○	○	-	-
		7	PLC I/F diagnosis		○	○	○	○
		6	Signal tracing		○	○	○	○

EZMotion-NC E60/E68 Series Specifications List

○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class				E68		E60	
				M	L	M	L
16 Cabinet and installation							
17 Servo/Spindle system							
	1 Feed axis						
		1	MDS-C1-V1/C1-V2 (200V)				
			Servo motor: HC**-A51/E51 (1000kp/rev)	□	□	-	-
			Servo motor: HC**-A42/E42 (100kp/rev)	□	□	-	-
		3	MDS-CH-V1/CH-V2 (400V)				
			Servo motor: HC**-A51/E51 (1000kp/rev)	□	□	-	-
			Servo motor: HC**-A42/E42 (100kp/rev)	□	□	-	-
		4	MDS-B-SVJ2 (Compact and small capacity)				
			Servo motor: HC**-A42/E42 (100kp/rev)	□	□	□	□
			Servo motor: HC**-A47 (100kp/rev)	□	□	□	□
			Servo motor: HC**-A33/E33 (25kp/rev)	□	□	□	□
			Servo motor: HC-SF/HC-RF (16kp/rev)	□	□	□	□
			Servo motor: HA-FF/HC-MF (8kp/rev)	□	□	□	□
		6	MDS-R-V1/R-V2 (200V Compact and small capacity)				
			Servo motor: HF**-A48 (260kp/rev)	□	□	□	□
			Servo motor: HF**-A47 (100kp/rev)	□	□	□	□
	2 Spindle						
		1	MDS-C1-SP/C1-SPM/B-SP (200V)				
			Spindle motor: SJ/SJ-V	□	□	-	-
			IPM spindle motor: SJ-PMF	□	□	-	-
		2	MDS-CH-SP/CH-SPH (400V)	□	□	-	-
		3	MDS-B-SPJ2 (Compact and small capacity)				
			Spindle motor: SJ-P/SJ-PF	□	□	△	△
	3 Auxiliary axis						
		1	Index/Positioning servo: MR-J2-CT				
			Servo motor: HC-SF/HC-RF (16kp/rev)	□	□	□	□
			Servo motor: HA-FF/HC-MF (8kp/rev)	□	□	□	□
	4 Power supply						
		1	Power supply: MDS-C1-CV/B-CVE	□	□	-	-
		2	AC reactor for power supply	□	□	-	-
		3	Ground plate	△	△	△	△

EZMotion-NC E60/E68 Series Specifications List

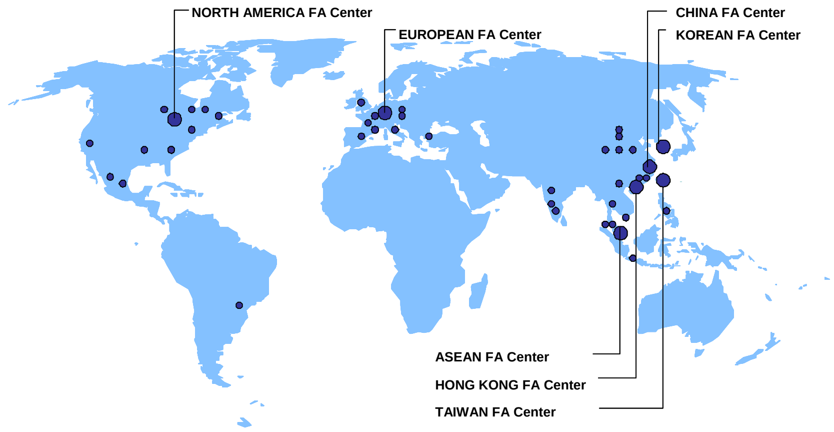
○:Standard, ○*:Display only, △:Optional, ☆:Planning, □:Selection

Class				E68		E60	
				M	L	M	L
18 Machine support functions							
	1 PLC						
		1 PLC basic function					
			1 Built-in PLC basic function	○	○	○	○
		2 Built-in PLC processing mode					
			2 MELSEC development tool I/F	○	○	○	○
		3 Built-in PLC capacity (Number of steps)					
			4000(PLCemulation)	-	-	○	○
			6000(PLCemulation)	○	○	-	-
			32000	△	△	-	-
		4 Machine contact input/output I/F		○	○	○	○
		5 Ladder monitor		○	○	○	○
		6 PLC development					
			1 On-board development	○	○	○	○
			2 MELSEC development tool	○	○	○	○
		9 PLC password lock		○	○	○	○
		13 PLC message					
			1 Japanese	○	○	○	○
			2 English	○	○	○	○
			3 German	○	○	○	○
			4 Italian	○	○	○	○
			5 French	○	○	○	○
			6 Spanish	○	○	○	○
			7 Chinese				
			Simplified Chinese characters	○	○	○	○
			9 Portuguese	○	○	○	○
			10 Hungarian	○	○	○	○
			11 Dutch	○	○	○	○
			12 Swedish	○	○	○	○
		14 User PLC version up		○	○	-	-
	2 Machine construction						
		1 Servo OFF		○	○	○	○
		2 Axis detach		○	○	○	○
		4 Inclined axis control		-	○	-	○
		5 Index table indexing		○	○	-	-
		6 NSK table connection control		○	○	-	-
		7 Auxiliary axis control		△	△	△	△
	3 PLC operation						
		1 Arbitrary feed in manual mode		○	○	○	○
		3 PLC axis control		△	△	△	△
	4 PLC interface						
		1 CNC control signal		○	○	○	○
		2 CNC status signal		○	○	○	○
		5 DDB		○	○	○	○
	5 Machine contact I/O						
		DI:64/DO:64		□	□	-	-
		DI:64/DO:48/AO:1		□	□	○	○
		DI:96/DO:80		□	□	-	-
		DI96/DO:80/AO:1		□	□	-	-
		Additional DI/DO (DI:64/DO:48)		□	□	□	□
		Additional DI/DO (DI:32/DO:32)		□	□	□	□
		Operation board IO DI:32/DO:32		□	□	□	□
		Operation board IO DI:64/DO:48		□	□	□	□
		Remote IO 32/32		△	△	△	△
		Remote IO 64/48		△	△	△	△
	7 Installing S/W for machine tools						
		3 Simple customization		○	○	○	○

Revision History

Date of revision	Manual No.	Revision details
Mar. 2006	IB(NA)1500171-A	First edition created.
Aug. 2006	IB(NA)1500171-B	• Contents were revised to correspond to E60 system S/W version C. • FCU6-DUN26, color display for E60 was added. • Mistakes were corrected.

Global service network



North America FA Center (MITSUBISHI ELECTRIC AUTOMATION INC.)

Illinois CNC Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL. 60061, U.S.A.
TEL: +1-847-478-2500 (Se FAX: +1-847-478-2650 (Se

California CNC Service Center
5665 PLAZA DRIVE, CYPRESS, CA. 90630, U.S.A.
TEL: +1-714-220-4796 FAX: +1-714-229-3818

Georgia CNC Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GA., 30097, U.S.A.
TEL: +1-678-258-4500 FAX: +1-678-258-4519

New Jersey CNC Service Center
200 COTTONTAIL LANE SOMERSET, NJ. 08873, U.S.A.
TEL: +1-732-560-4500 FAX: +1-732-560-4531

Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500 FAX: +1-269-673-4092

Ohio CNC Service Satellite
62 W. 500 S., ANDERSON, IN., 46013, U.S.A.
TEL: +1-847-478-2608 FAX: +1-847-478-2690

Texas CNC Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TX. 76051, U.S.A.
TEL: +1-817-251-7468 FAX: +1-817-416-1439

Canada CNC Service Center
4299 14TH AVENUE MARKHAM, ON. L3R OJ2, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935

Mexico CNC Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 FAX: +52-55-9171-7698

Monterrey CNC Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 FAX: +52-81-8365-4171

Brazil MITSUBISHI CNC Agent Service Center
(AUTOMOTION IND. COM. IMP. E EXP. LTDA.)
ACESSO JOSE SARTORELLI, KM 2.1 18550-000 BOITUVA – SP, BRAZIL
TEL: +55-15-3363-9900 FAX: +55-15-3363-9911

European FA Center (MITSUBISHI ELECTRIC EUROPE B.V.)

Germany CNC Service Center
GOTHAER STRASSE 8, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 FAX:+49-2102486-591

South Germany CNC Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: +49-711-3270-010 FAX: +49-711-3270-0141

France CNC Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 FAX: +33-1-49-01-07-25

Lyon CNC Service Satellite

U.K CNC Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-282-846 FAX:-44-1707-278-992

Italy CNC Service Center
ZONA INDUSTRIALE VIA ARCHIMEDE 35 20041 AGRATE BRIANZA, MILANO ITALY
TEL: +39-039-60531-342 FAX: +39-039-6053-206

Spain CNC Service Satellite
CTRA. DE RUBI, 76-80 -APDO.420 08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 FAX:

Turkey MITSUBISHI CNC Agent Service Center
(GENEL TEKNIK SISTEMLER LTD. STI.)
DARULACEZE CAD. FAMAS IS MERKEZI A BLOCK NO.43 KAT2 80270 OKMEYDANI ISTANBUL, TURKEY
TEL: +90-212-320-1640 FAX: +90-212-320-1649

Poland MITSUBISHI CNC Agent Service Center (MPL Technology Sp. z. o. o)
UL SLICZNA 34, 31-444 KRAKOW, POLAND
TEL: +48-12-632-28-85 FAX:

Wroclaw MITSUBISHI CNC Agent Service Satellite (MPL Technology Sp. z. o. o)
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 FAX: +48-71-333-77-53

Czech MITSUBISHI CNC Agent Service Center
(AUTOCONT CONTROL SYSTEM S.R.O.)
NEMOCNICNI 12, 702 00 OSTRAVA 2 CZECH REPUBLIC
TEL: +420-596-152-426 FAX: +420-596-152-112

ASEAN FA Center (MITSUBISHI ELECTRIC ASIA PTE. LTD.)

Singapore CNC Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 FAX: +65-6476-7439

Thailand MITSUBISHI CNC Agent Service Center (F. A. TECH CO., LTD)
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 12,14 RAMA III RD BANGPONGPANG, YANNAWA, BANGKOK 10120. THAILAND
TEL: +66-2-682-6522 FAX: +66-2-682-6020

Malaysia MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN MALAYSIA
TEL: +60-3-5631-7605 FAX: +60-3-5631-7636

JOHOR MITSUBISHI CNC Agent Service Satellite
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
NO. 16, JALAN SHAHBANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 FAX: +60-7-557-3404

Indonesia MITSUBISHI CNC Agent Service Center
(PT. AUTOTEKNINDO SUMBER MAKMUR)
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 FAX: +62-21-3917-164

India MITSUBISHI CNC Agent Service Center (MESSUNG SALES & SERVICES PVT. LTD.)
B-36FF, PAVANA INDUSTRIAL PREMISES M.I.D.C., BHOASRI PUNE 411026, INDIA
TEL: +91-20-2711-9484 FAX: +91-20-2712-8115

BANGALORE MITSUBISHI CNC Agent Service Satellite
(MESSUNG SALES & SERVICES PVT. LTD.)
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 FAX: +91-80-532-0480

Delhi MITSUBISHI CNC Agent Parts Center (MESSUNG SALES & SERVICES PVT. LTD.)
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32ND MILESTONE GURGAON 122001, INDIA
TEL: +91-98-1024-8895 FAX:

Philippines MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM CORPORATION)
UNIT No.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 FAX: +63-2-807-2417

Vietnam MITSUBISHI CNC Agent Service Center (SA GIANG TECHNO CO., LTD)
47-49 HOANG SA ST. DAKAO WARD, DIST.1 HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 FAX: +84-8-910-2593

China FA Center (MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD.)

China CNC Service Center
2/F., BLOCK 5 BLDG.AUTOMATION INSTRUMENTATION PLAZA, 103 CAOBAO RD. SHANGHAI 200233, CHINA
TEL: +86-21-6120-0808 FAX: +86-21-6494-0178

Shenyang CNC Service Center
TEL: +86-24-2397-0184 FAX: +86-24-2397-0185

Beijing CNC Service Satellite
9/F, OFFICE TOWER1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE, DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 FAX: +86-10-6518-8030

China MITSUBISHI CNC Agent Service Center
(BEIJING JIAYOU HIGHTECH TECHNOLOGY DEVELOPMENT CO.)
RM 709, HIGH TECHNOLOGY BUILDING NO.229 NORTH SI HUAN ZHONG ROAD, HAIDIAN DISTRICT , BEIJING 100083, CHINA
TEL: +86-10-8288-3030 FAX: +86-10-6518-8030

Tianjin CNC Service Satellite
RM909, TAIHONG TOWER, NO220 SHIZILIN STREET, HEBEI DISTRICT, TIANJIN, CHINA 300143
TEL: -86-22-2653-9090 FAX: +86-22-2635-9050

Shenzhen CNC Service Satellite
RM02, UNIT A, 13/F, TIANAN NATIONAL TOWER, RENMING SOUTH ROAD, SHENZHEN, CHINA 518005
TEL: +86-755-2515-6691 FAX: +86-755-8218-4776

Changchun Service Satellite
TEL: +86-431-50214546 FAX: +86-431-5021690

Hong Kong CNC Service Center
UNIT A, 25/F RYODEN INDUSTRIAL CENTRE, 26-38 TA CHUEN PING STREET, KWAI CHUNG, NEW TERRITORIES, HONG KONG
TEL: +852-2619-8588 FAX: +852-2784-1323

Taiwan FA Center (MITSUBISHI ELECTRIC TAIWAN CO., LTD.)

Taichung CNC Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Taipei CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Tainan CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Korean FA Center (MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD.)

Korea CNC Service Center
DONGSEO GAME CHANNEL BLDG. 2F. 660-11, DEUNGCHON-DONG KANGSEO-KU SEOUL, 157-030 KOREA
TEL: +82-2-3660-9607 FAX: +82-2-3663-0475

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	EZMotion-NC E60/E68 Series
MODEL CODE	008－397
Manual No.	IB-1500171(ENG)-B

(0608) MEE