Changes for the Better

MITSUBISHI ELECTRIC

CNC
MELDAS C6/C64

SPECIFICATIONS MANUAL

BNP-B2266C(ENG)

Introduction

This manual describes the specifications of MELDAS C6/C64.
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 it's specification according to the model.

- ○ : Standard
- △ : Option
- □ : 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

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)

C6		C64		
T system	L system	M system	L system	T system
1	2	3	2	1

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

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.
For example, if 14 NC axes are used, this alone is the maximum number of control axes, so a spindle, PLC axis and auxiliary axis cannot be connected.
The connection order is the NC axis, PLC axis, spindle and auxiliary axis.

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

C6		C64		
T system	L system	M system	L system	T system
7	7	14	14	14

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

C6		C64		
T system	L system	M system	L system	T system
4	6	14	14	14

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

C6		C64		
T system	L system	M system	L system	T system
2	4	14	14	14

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

C6		C64		
T system	L system	M system	L system	T system
2	4	14	12	14

Max. number of spindles
Includes analog spindles.

C6		C64		
T system	L system	M system	L system	T system
2 (1)	2 (1)	3	4	7 (1)

Values in parentheses indicate the maximum number of spindles per part system.

Max. number of PLC axes

C6		C64		
T system	L system	M system	L system	T system
–	–	7	7	7

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

C6		C64		
T system	L system	M system	L system	T system
Δ5	Δ5	Δ7	Δ7	Δ7

1.1.3 Number of Simultaneous Contouring Control Axes

Simultaneous control of all axes is possible as a principle in the same part system.
However, for actual use, the machine tool builder specification will apply.

C6		C64		
T system	L system	M system	L system	T system
2	2	4	4	2

1.1.4 Max. Number of NC Axes in a Part System

C6		C64		
T system	L system	M system	L system	T system
2	2	6	4	2

1.2 Control Part System

1.2.1 Standard Number of Part Systems

C6		C64		
T system	L system	M system	L system	T system
1	1	1	1	1

1.2.2 Max. Number of Part Systems

C6		C64		
T system	L system	M system	L system	T system
Δ2	Δ2	Δ3	Δ3	Δ7

For actual use, the machine tool builder specification will apply.

1.3 Control Axes and Operation Modes

1.3.2 Memory Mode

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

1.3.3 MDI Mode

C6		C64		
T system	L system	M system	L system	T 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.

2. Input Command

2.1 Data Increment

Least command increment: 1 μm (Least input increment: 1 μm)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Least command increment: 0.1 μm (Least input increment: 0.1 μm)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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

(1) The least input 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. This increment is applied per part system (all part systems, PLC axis).

Increment type	Input increment (parameter)	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least input 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.

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

Increment type	Command increment (parameter)	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Command increment	10	0.001	0.001	0.0001	0.001
	100	0.01	0.01	0.001	0.01
	1000	0.1	0.1	0.01	0.1
	10000	1.0	1.0	0.1	1.0

(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.

2.2 Unit System

2.2.1 Inch/Metric Changeover; G20/G21

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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.

Parameter \ Data		Machining program		Screen data (Offset amount, etc.)	Parameter
I_inch	0	G20	Inch unit system	Metric unit system	Not affected
		G21	Metric unit system		
	1	G20	Inch unit system	Inch unit system	
		G21	Metric unit system		
M_inch	0	Not affected		Not affected	Metric unit system
	1				Inch unit system

(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

C6		C64		
T system	L system	M system	L system	T 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 (G code list 2, 3)

C6		C64		
T system	L system	M system	L system	T 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.

2.3.2.4 Format 1 for Machining Center (G code list 1)

C6		C64		
T system	L system	M system	L system	T system
○	–	○	–	○

2.4 Command Value

2.4.1 Decimal Point Input I, II

C6		C64		
T system	L system	M system	L system	T 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 increment.
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
G1 Y200 F100 (*1)	Y200μm, F100mm/min	Y200mm, F100mm/min
G4 X1.5	Dwell 1.5 s	←
G4 X2	2ms	2s

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

2.4.2 Absolute/Incremental Command; G90/G91

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

(1) T system, 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)

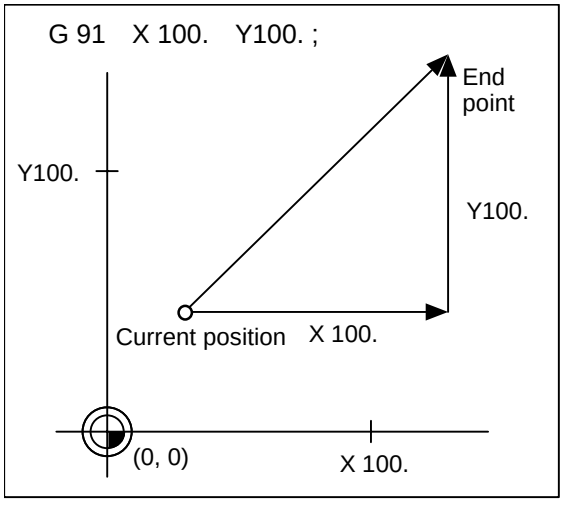
These G codes can be commanded multiple times in one block.

Example

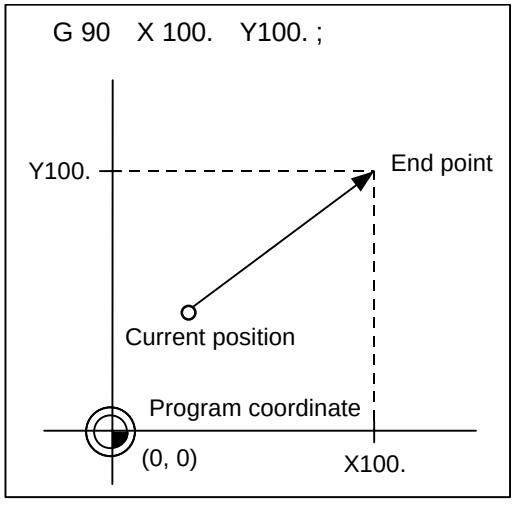
G90 X100.	G91 Y200.	G90 Z300.	;
Absolute value	Incremental value	Absolute value	

(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.

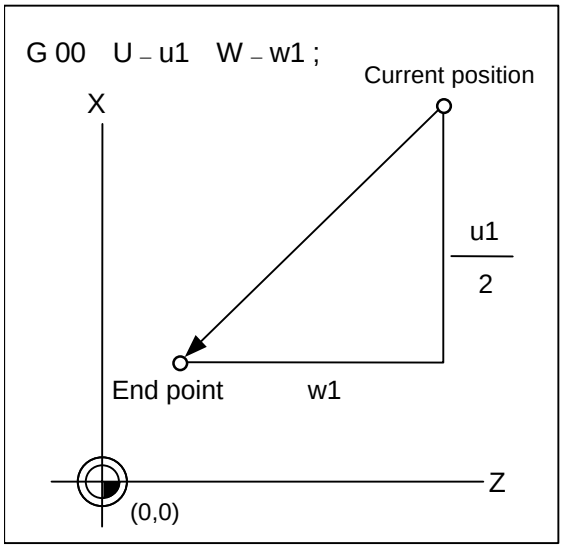
Absolute command (absolute value command) ... X, Z

Incremental command (incremental value command) ... U, W

Example

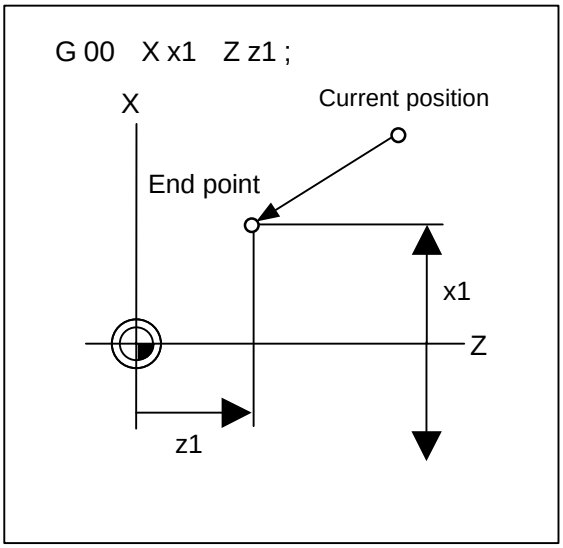
G00	X100.	W200.	;
	Absolute value	Incremental value	

(Incremental value command)



The above drawing shows the case for the diameter command.

(Absolute value command)



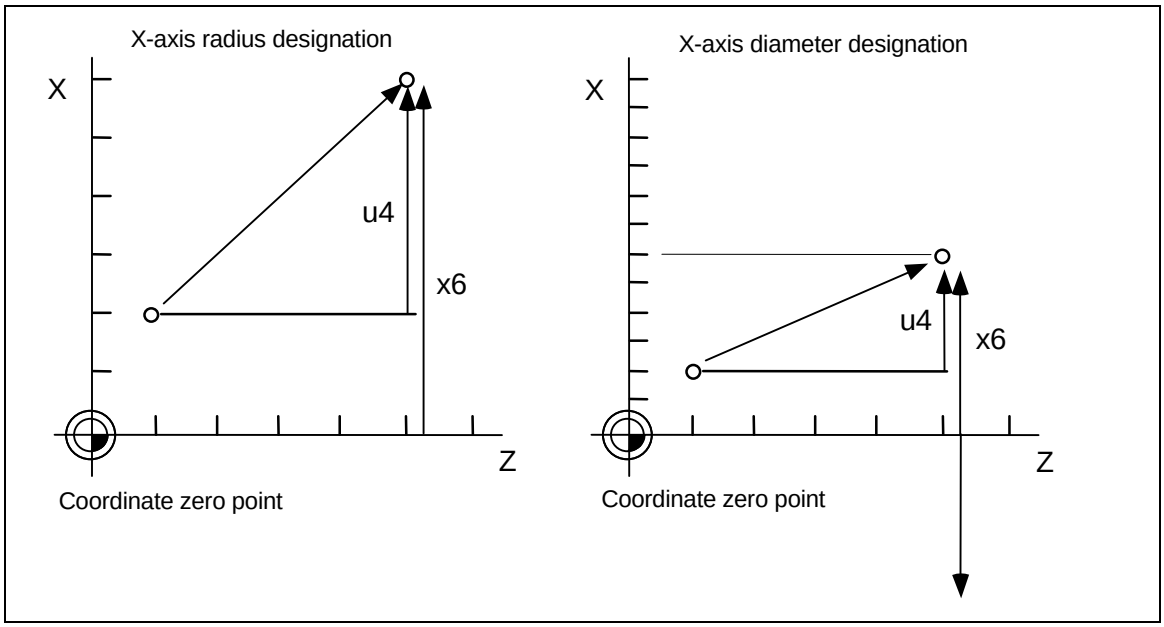
The above drawing shows the case for the diameter command.

(Note) In addition to the above command method using the above axis addresses, the absolute value command and incremental value command can be switched by commanding the G code (G90/G91). (Select with the parameters.)

2.4.3 Diameter/Radius Designation

C6		C64		
T system	L system	M system	L system	T system
–	○	–	○	–

For axis 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 workpiece 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

<Brief summary of format details>

[T system, 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+44/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 Z+53 α+53	X+44 Z+44 α+44	X+53 Z+53 α+53	X+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 K+53 R+53	I+44 K+44 R+44	I+53 K+53 R+53	I+44 K+44 R+44 (Note 5)
	0.0001(°) mm/ 0.00001 inch	I+44K+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+44/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		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.

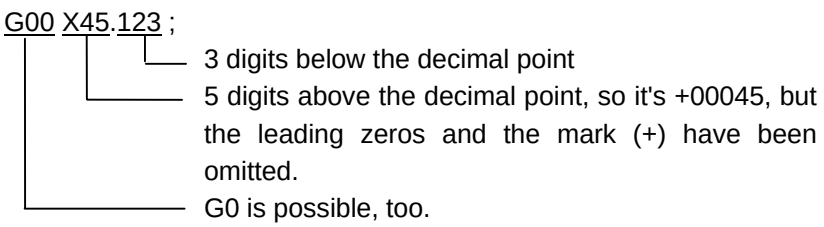
(Note 4) The meanings of the details are as follows :

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 :



(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.

<List of Command Value and Setting Value Ranges>

	Linear axis		Rotary axis
	Input unit: mm	**Input unit: inch**	**Degree (°)**
Least setting increment	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 100000 inch/min 0.0001 to 10000 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)	±9999.999 mm ±999.9999 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 mm to +99999.999 mm –9999.9999 mm to +9999.9999 mm	–9999.9999 inch to +9999.9999 inch –999.99999 inch 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 99.9999 inch/rev 0.00001 to 9.99999 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.

3. Positioning/Interpolation

3.1 Positioning; G0, G60

3.1.1 Positioning; G0

C6		C64		
T system	L system	M system	L system	T 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 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

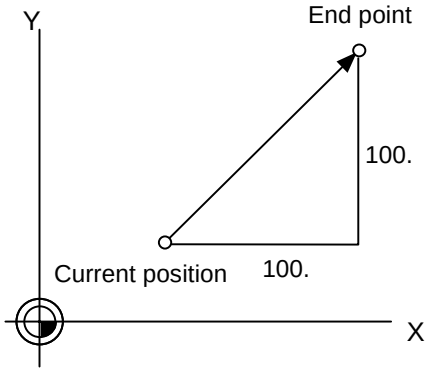
x1, y1, z1: numerical values denoting the position data

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 which 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 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)

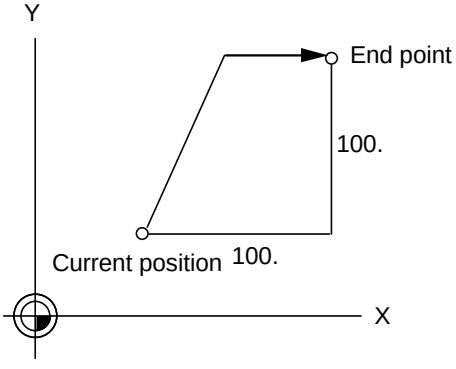
(Example) Linear type (Moves lineary to the end point.)

G 00 G 91 X 100. Y 100. ;



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

G 00 G 91 X 100. Y 100. ;



(Note) If the acceleration/deceleration conditions differ between the axes, the path will not be linear to the end point even when using the linear type.

(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; G60

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

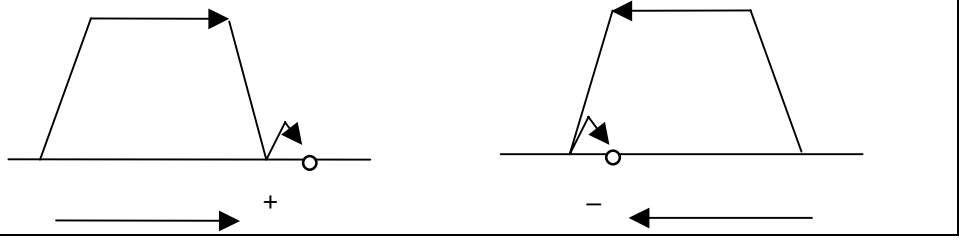
The G60 command always moves the tool to the final position in a direction determined with parameters.
The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

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

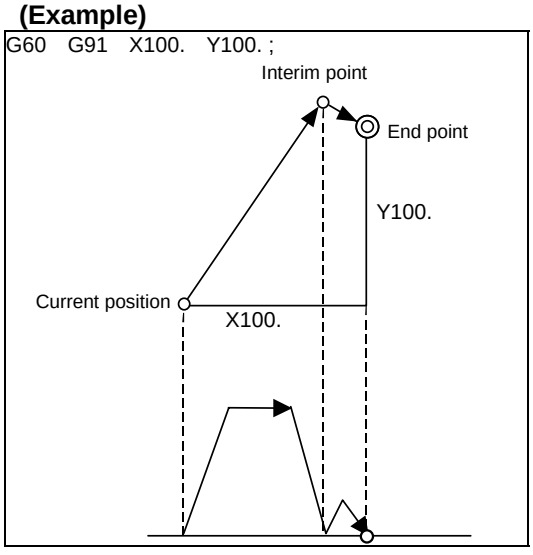
With the above command, the tool is first moved to a position distanced from the end point position by an amount equivalent to the creep distance (parameter setting) and then moved to its final position.
For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse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

G60 G91 X100. Y100. ;



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) On the creep distance, the tool 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; G1, G2/G3

3.2.1 Linear Interpolation; G1

C6		C64		
T system	L system	M system	L system	T 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.

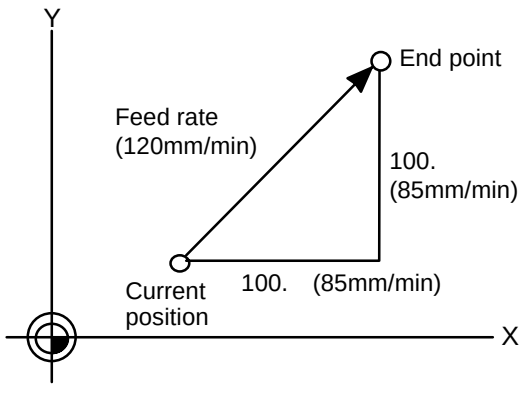
G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)

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 which 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); G2/G3

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

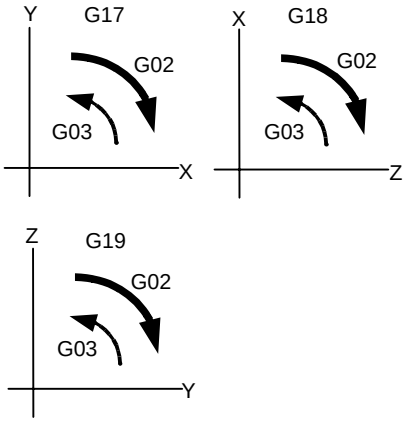
This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command value supplied in the program.

G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)

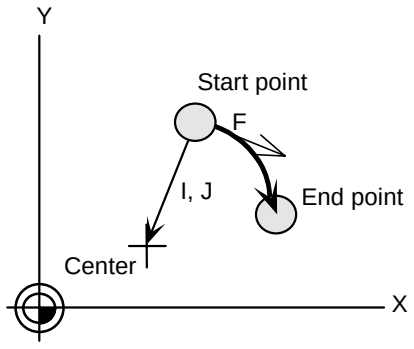
G02, G03 : Arc rotation direction
Xx1, Yy1 : End point coordinate values
Ii1, Jj1 : Arc center coordinate values
Ff1 : Feed rate

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

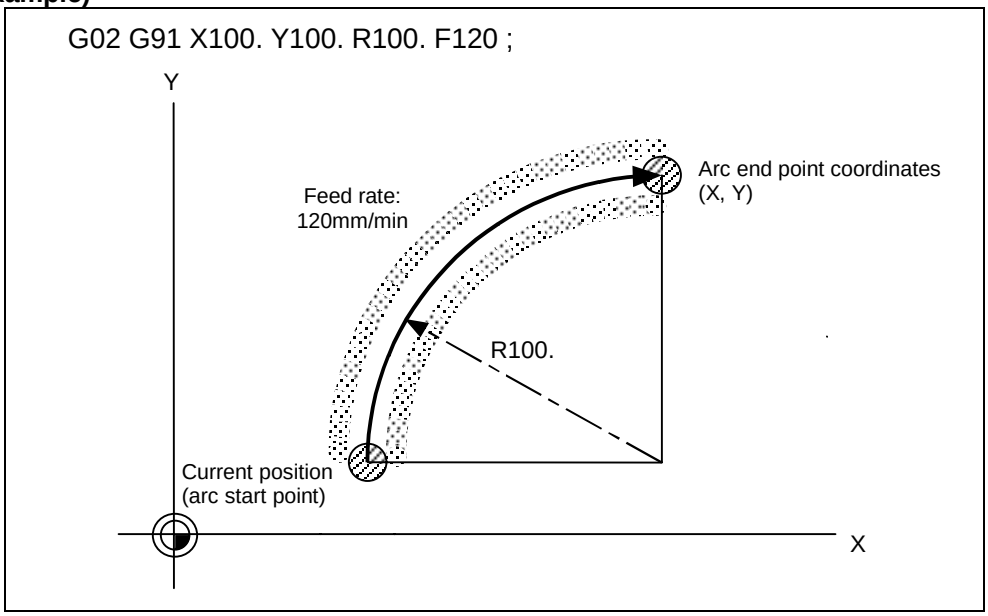
Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

G02(G03) Xx1 Yy1 Rr1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)	
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

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	–	–

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 value of linear axis
Ii1, Jj1 : Arc center coordinate values
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 coordinate 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.

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

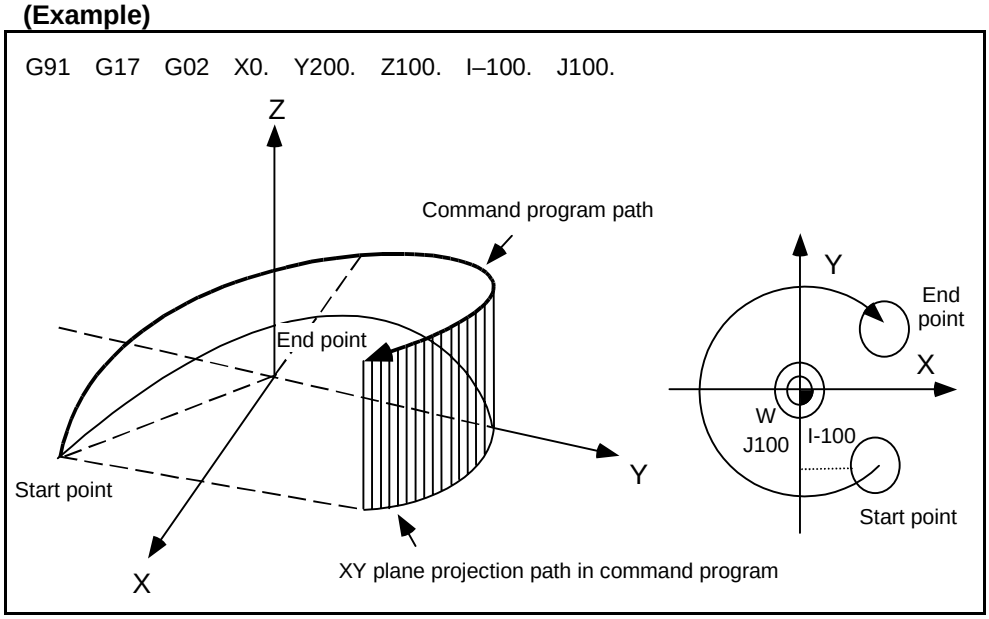
$\theta = \theta e - \theta s = \arctan(ye/xe) - \arctan(ys/xs)$

Where xs, ys are the start point coordinates ($0 \leq \theta < 2\pi$)

xe, ye are the end point coordinates

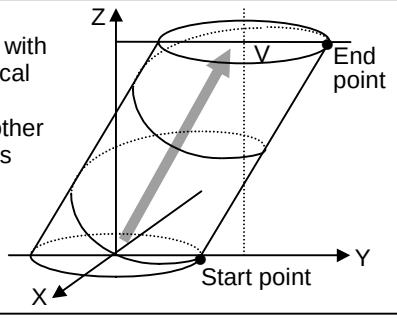
The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination under the specifications. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

(Example)



(Note 1) Helical shapes 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.



4. Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

C6		C64		
T system	L system	M system	L system	T system
1000	1000	1000	1000	1000

[T system, M system]

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

- Rapid Traverse Rate setting range

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

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

[L system]

The rapid traverse rate can be set independently for each axis.
The rapid traverse rate is effective for G00, G27, G28, G29, G30 and G53 commands.
Override can be applied to the rapid traverse rate using the external signal supplied.

- Rapid Traverse Rate setting range

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

C6		C64		
T system	L system	M system	L system	T system
1000	1000	1000	1000	1000

[T system, M system]

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

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

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

- The cutting feed rate is effective for G01, G02, G03, G33 commands, etc. As to others, refer to the interpolation specifications.

[L system]

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 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

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

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

- The cutting feed rate is effective for G01, G02, G03, G33 commands, etc. As to others, refer to interpolation specifications.

4.1.3 Manual Feed Rate (m/min)

C6		C64		
T system	L system	M system	L system	T system
1000	1000	1000	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 value setting 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 value setting method.

4.2 Feed Rate Input Methods; G94/G95

4.2.1 Feed per Minute

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

[T 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).

Metric input (mm)

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

Inch input (inch)

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

- 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 to OFF.
- The F command increments are common to all part systems.

[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).

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)		0.001~1000000.000	0.0001 ~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)		0.0001~39370.0787	0.00001~3937.00787

- 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 to OFF.

4.2.2 Feed per Revolution

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 F command increment and command range are as follows.

[T system, M system]

Metric input (mm)

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

Inch input (inch)

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

- 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 to OFF.
- The F command increments are common to all part systems.

[L system]

Metric input (mm)

Least input increment		(B) 0.001 mm	(C) 0.0001 mm
F command increment (mm/rev)	**without decimal point**	F1 = 0.0001	F1 = 0.0001
	with decimal point	F1. = 1	F1. = 1
Command range (mm/rev)		0.0001~999.999	0.00001~99.99999

Inch input (inch)

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

- 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 to OFF.

4.2.4 F1-digit Feed

C6		C64		
T system	L system	M system	L system	T 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 applied when F0 is issued which is the same as the G00 command. When one of the codes F1 to F5 is issued, the cutting feed rate set to support the code serves as the valid rate command. When a command higher than F5 is issued, it serves as a regular direct command with feed rate value of 5 digits following address F.
When an F1-digit command has been issued, the "In F1-digit" external output signal is output.

4.3 Override

4.3.1 Rapid Traverse Override

C6		C64		
T system	L system	M system	L system	T 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.
Code method commands are assigned as combinations of Y device bit signals from the PLC.

(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.

4.3.2 Cutting Feed Override

C6		C64		
T system	L system	M system	L system	T 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 device 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.

4.3.3 2nd Cutting Feed Override

C6		C64		
T system	L system	M system	L system	T 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.

4.3.4 Override Cancel

C6		C64		
T system	L system	M system	L system	T 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 (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.

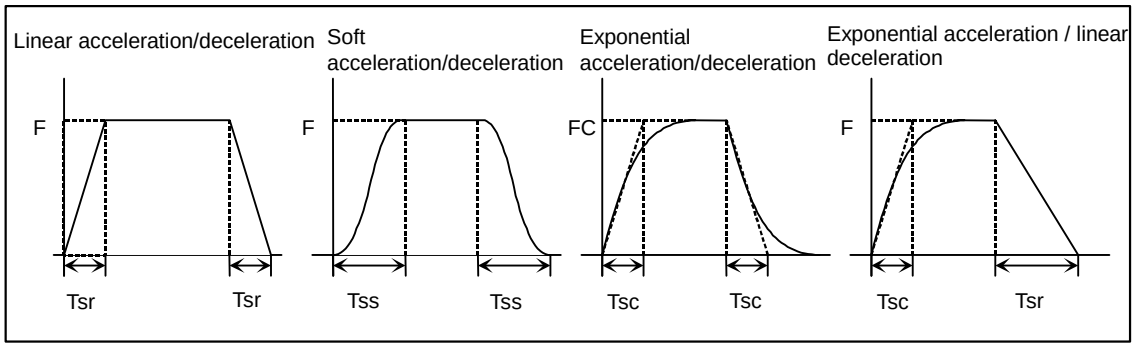
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration after Interpolation

C6		C64		
T system	L system	M system	L system	T 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.



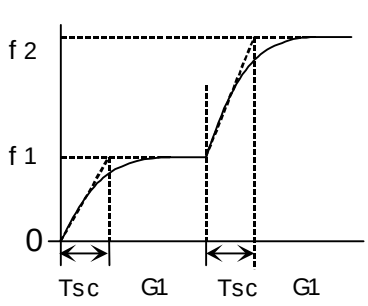
(Note 1) The rapid traverse feed acceleration/deceleration patterns are effective for the following:

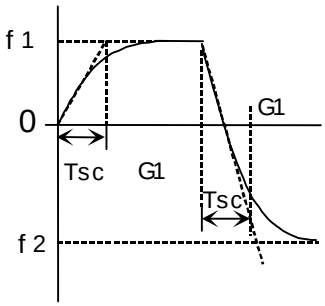
G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG, incremental feed, return to reference position.

(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

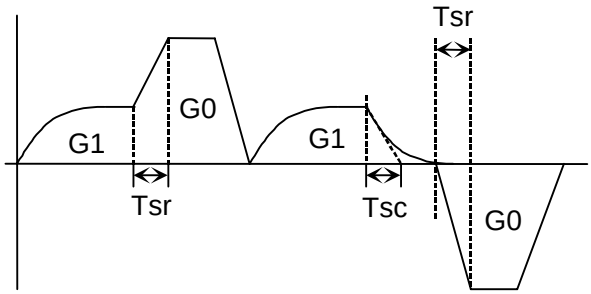
(1) Continuous G1 blocks

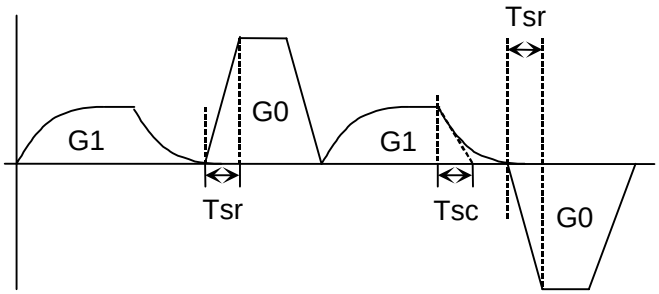




The tool does not decelerate between blocks.

(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

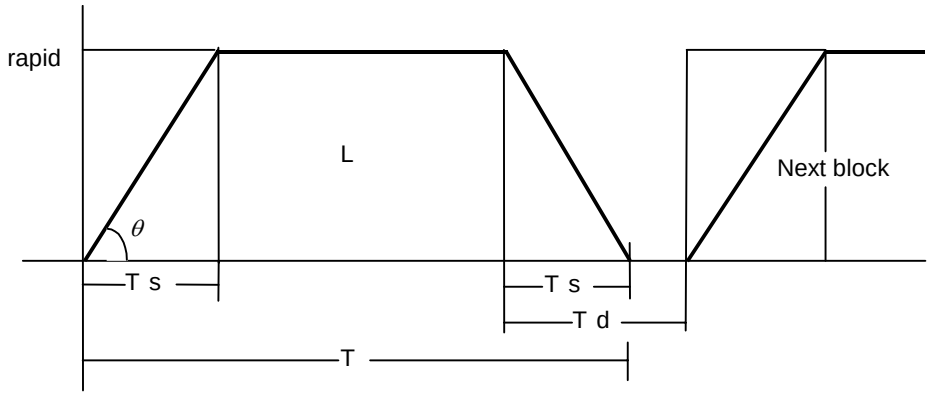
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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



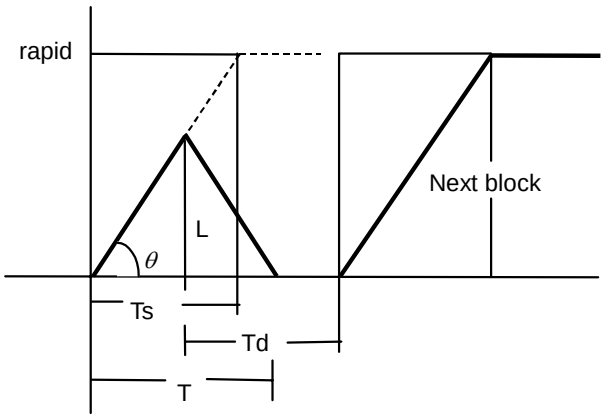
rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

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

$$Td = Ts + (0 \sim 1.7\ ms)$$

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

(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

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

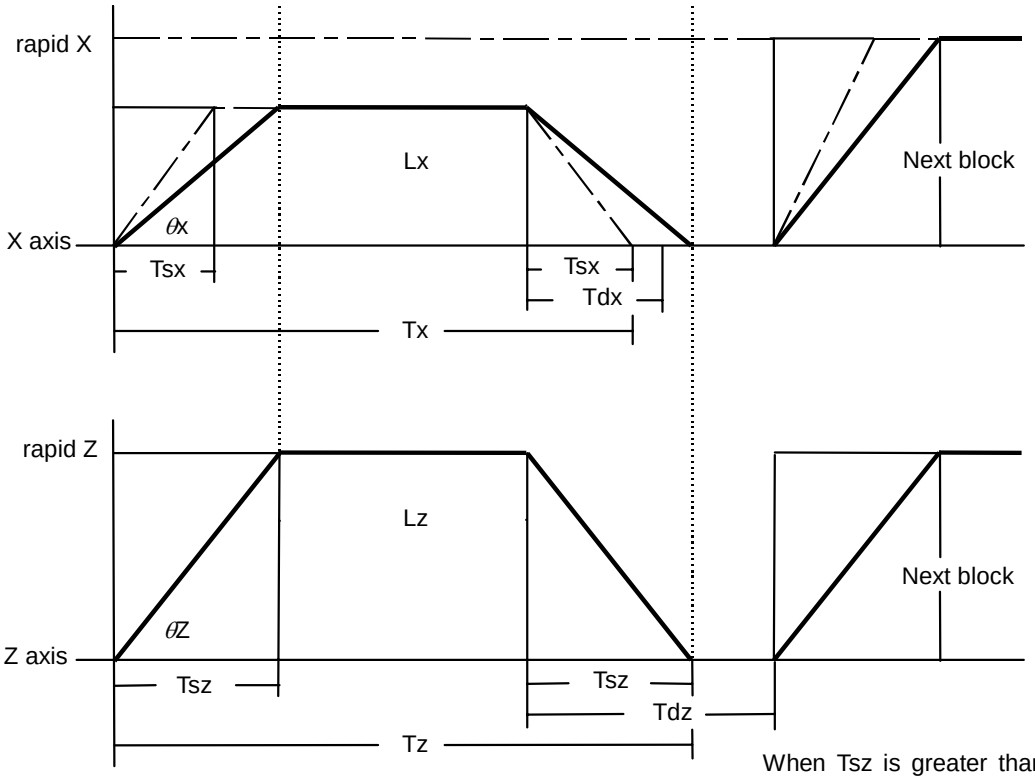
$$Td = \frac{T}{2} + (0 \sim 1.7\ ms)$$

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

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); G33

C6		C64		
T system	L system	M system	L system	T system
Δ	○	Δ	○	Δ

(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, within 0 to 360°)
Ff1 : Thread lead
Ee1 : Thread lead (precise lead threads)

The tables below indicate the thread lead ranges.

[T system, M system]

Metric command		
Least input increment (mm)	**F (mm/rev)**	**E (mm/rev)**
0.001	0.001 ~999.999	0.00001 ~999.99999
0.0001	0.0001 ~99.9999	0.000001 ~99.999999

Inch command		
Least input increment (inch)	**F (inch/rev)**	**E (inch/rev)**
0.0001	0.0001 ~39.3700	0.000001 ~39.370078
0.00001	0.00001 ~3.93700	0.000001 ~3.937007

[L system]

Metric command		
Least input increment (mm)	**F (mm/rev)**	**E (mm/rev)**
0.001	0.0001 ~999.9999	0.00001 ~999.99999
0.0001	0.00001 ~99.99999	0.00001 ~99.99999

Inch command		
Least input increment (inch)	**F (inch/rev)**	**E (inch/rev)**
0.0001	0.00001 ~99.999999	0.000010 ~9.9999999
0.00001	0.000001 ~9.9999999	0.0000010 ~0.99999999

The direction of the axis with 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, within 0 to 360°)
Ee1 : Thread number per inch

The tables below indicate the thread leads.

[T system, M system]

Metric command	
Least input increment (mm)	**Thread number command range (thread/inch)**
0.001	0.03~999.99
0.0001	255~9999.999

Inch command	
Least input increment (inch)	**Thread number command range (thread/inch)**
0.0001	0.0255~9999.9999
0.00001	0.25401~999.9999

[L system]

Metric command	
Least input increment (mm)	**Thread number command range (thread/inch)**
0.001	0.03~999.99
0.0001	0.255~9999.999

Inch command	
Least input increment (inch)	**Thread number command range (thread/inch)**
0.0001	0.0101~9999.9999
0.00001	0.10001~999.99999

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

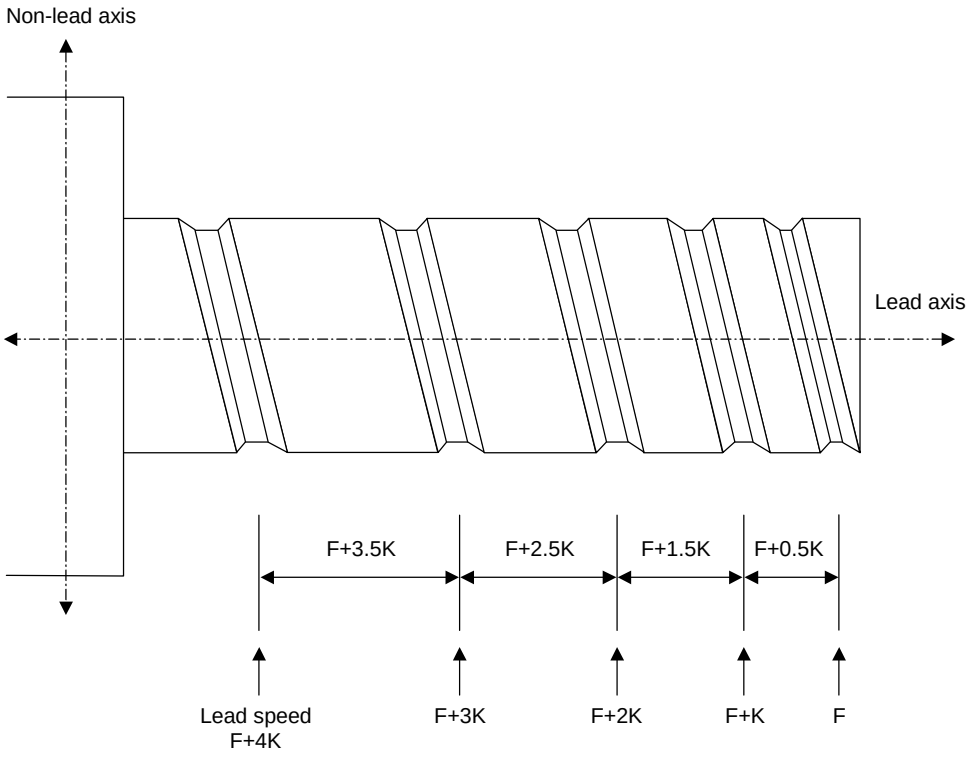
4.5.2 Variable Lead Thread Cutting; G34

C6		C64		
T system	L system	M system	L system	T 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__;

G34 : Variable lead thread cutting command
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; G74, G84

4.5.3.1 Synchronous Tapping Cycle

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

This function performs tapping through the synchronized control of the spindle and servo axis. 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 , R1 ;

G84	: Synchronous tapping mode ON, forward tapping
G74	: Synchronous tapping mode ON, reverse tapping
Xx1, Yy1	: Hole position data, hole drilling coordinate position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, hole R position
Pp1	: Hole machining data, dwell time at hole bottom
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Ss1	: Spindle speed
,R1	: Synchronous system selection

(2) Tapping thread number assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Pp1 Ee1 Ss1 , R1 ;

G84	: Synchronous tapping mode ON, forward tapping
G74	: Synchronous tapping mode ON, reverse tapping
Xx1, Yy1	: Hole position data, hole drilling coordinate position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, hole R position
Pp1	: Hole machining data, dwell time at hole bottom
Ee1	: Tap thread number per 1-inch feed of Z axis
Ss1	: Spindle speed
,R1	: Synchronous system selection

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

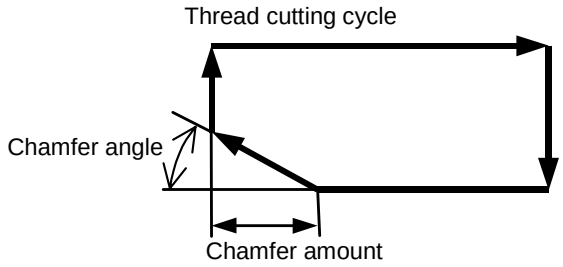
The tapping mode will be canceled with the following G commands.

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

4.5.4 Chamfering

C6		C64		
T system	L system	M system	L system	T system
–	○	–	○	–

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

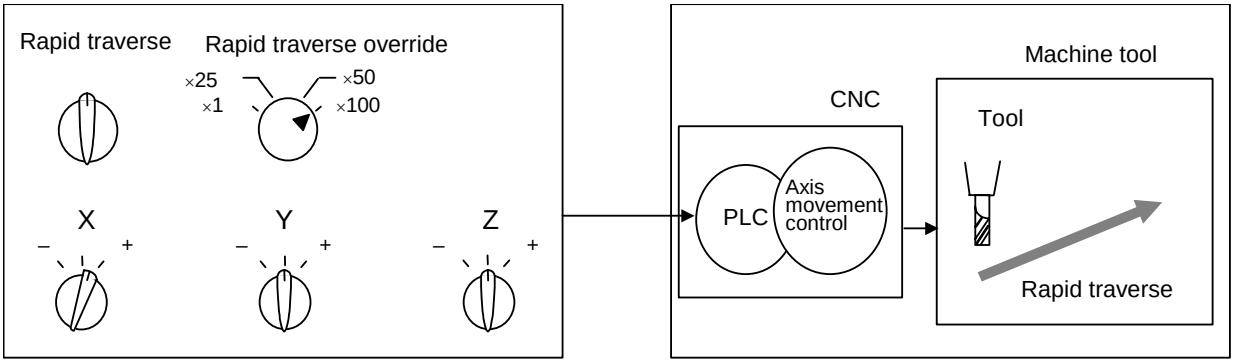


4.6 Manual Feed

4.6.1 Manual Rapid Traverse

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

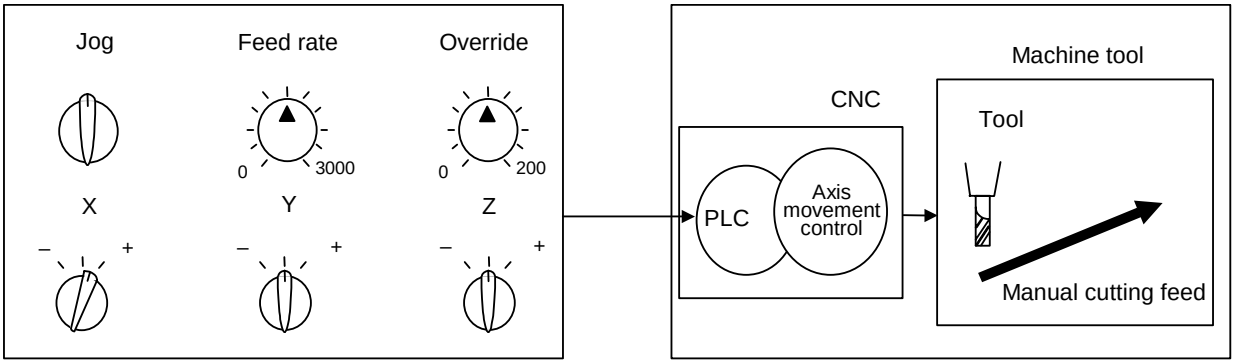
When the manual rapid traverse mode is selected, the tool can be moved at the rapid traverse rate for each axis separately. Override can also be applied to the rapid traverse rate by means of the rapid traverse override function.
Rapid traverse override is common to all part systems.



4.6.2 Jog Feed

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

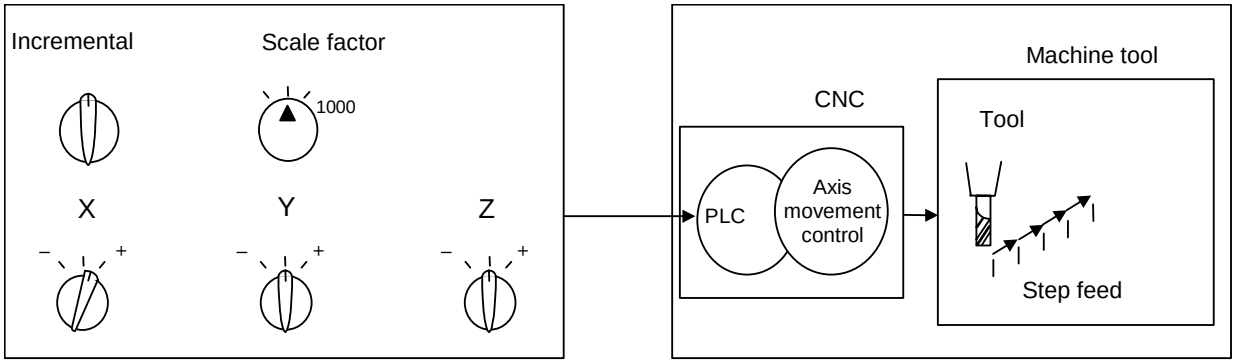
When the jog feed mode is selected, the tool can be moved in the axis direction (+ or –) in which the machine is to be moved at the per-minute feed. The jog feed rate is common to all part systems.



4.6.3 Incremental Feed

C6		C64		
T system	L system	M system	L system	T 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.
The incremental feed amount parameter and its magnification rate are common to all part systems.



4.6.4 Handle Feed

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(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 arbitrary value.

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

(3-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.

4.7 Dwell; G04

4.7.1 Dwell (Time-based Designation)

C6		C64		
T system	L system	M system	L system	T 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

"x1" of the time-based dwell can be designated in the range from 0.001 to 99999.999 seconds.

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)

(Note) The tape length will be the total of two part systems when using the 2-part system specifications.

40 m (64 programs)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

80 m (128 programs)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

160 m (200 programs)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

320 m (200 programs)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

600 m (400 programs)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

5.2 Editing Method

5.2.1 Program Editing

C6		C64		
T system	L system	M system	L system	T 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.

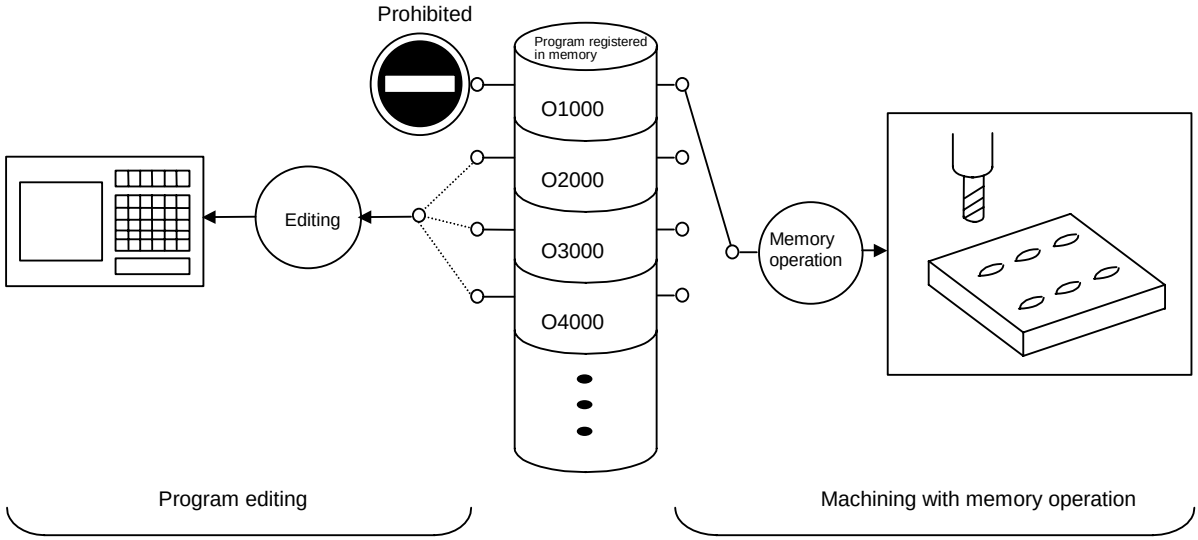
(4) Program editing

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

5.2.2 Background Editing

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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



(1) The data of the machining programs being used in memory operation can be displayed and scrolled on the setting and display unit, but data cannot be added, revised or deleted.
(2) The editing functions mentioned in the preceding section 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.

6. Operation and Display

6.1 Structure of Operation/Display Panel

The following display units can be used for the setting and display unit.

(1) 7.2-type monochrome LCD display unit

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(2) 10.4-type monochrome LCD display unit

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(3) 9-type monochrome CRT display unit

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(4) External personal computer display (Ethernet connection)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(5) Graphic operation terminal (GOT)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

6.2 Operation Methods and Functions

6.2.1 Memory Switch (PLC Switch)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The toggle switches (PLC switches) can be defined on the screen.
These switches can be turned ON/OFF on the screen, 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.
There are a total of 32 switch points.

6.3 Display Methods and Contents

6.3.1 Status Display

C6		C64		
T system	L system	M system	L system	T 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 unit, 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

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

(1) Present position counter

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Each axis' present position including tool length offset amount, tool radius compensation amount and workpiece coordinate offset amount is indicated.

(2) Workpiece coordinate counter

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The remaining distance of the movement command during the execution (incremental distance from the present position to the end point of the block) is indicated during the automatic start and automatic stop.

(4) Machine position counter

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Each axis' coordinate value in the basic machine coordinate system whose zero point is the characteristic position determined depending on the machine is indicated.

6.3.3 Program Running Status Display

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Program now being executed is displayed.

6.3.4 Setting and Display

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

6.3.5 MDI Data Setting and Display

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The clock is built-in, and the date (year, month, date) and time (hour, minute, second) 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

6.3.9 Integrated Time Display

C6		C64		
T system	L system	M system	L system	T 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 Available Languages (Japanese, English)

C6		C64		
T system	L system	M system	L system	T system
○2 languages	○2 languages	○2 languages	○2 languages	○2 languages

This function makes it possible to switch between Japanese and English which are the standard languages used for the screen displays.
The display can also be switched to Polish.

6.3.11 Additional Languages (Japanese, English, Polish)

6.3.11.1 Japanese

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

6.3.11.2 English

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

6.3.11.13 Polish

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

6.3.13 Screen Deletion

C6		C64		
T system	L system	M system	L system	T 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.

6.4 Display Unit Switch

6.4.1 Single-NC and Multi-Display Unit Switch

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

When multiple display units are connected to one NC, the active display unit can be selected with the changeover switch.

The functions that can be used with the display unit differ according to the functions and connection method.

Change-over target	Connection method	Display	Operation	Reset input	READY lamp	Remote I/O connection
Single-NC and multi-display unit switch	Cascade connection	Displayed only on selected display unit (No display on others)	Only selected display unit is valid	Input not possible	Displayed only on selected display unit (Others are OFF)	Connection not possible
	LAN connection	Display on all display units				
Multi-NC and common-display unit	Cascade connection	Only selected NC is displayed	Only selected NC is valid	Input not possible	Only selected NC is displayed	
	Daisy chain connection					Connectable with restrictions
	LAN connection	Only selected NC is displayed (Two NCs are simultaneously displayed when using 2-screen display)				Connection not possible

(Note) The new communication terminal (GOT) is required for the LAN connection. The connection format may differ according to the LAN device being used.

6.4.2 Multi-NC and Common-Display Unit

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

When a multiple number of NC systems are to be used, this function enables a single display unit to be used as the display for all the systems.
This function is useful when, for instance, the NC systems are used for dedicated machines on a line.

6.4.4 Multi-NC and Common-external PC Display

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

When a multiple number of NC systems are to be used, this function enables a single personal computer to be used as the display for all the systems.
This function is useful when, for instance, the NC systems are used for dedicated machines on a line.

6.4.5 Display Unit Detachable

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

This function enables the displays to be connected or detached without turning OFF the NC system's power.

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)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Tool offset data input / output

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Common variable input / output

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Parameter input / output

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

History data output

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

(Note) Options are required for the devices used for input and output.

7.2 Input/Output I/F

7.2.1 RS-232C I/F

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Port 2 of the RS-232C interface can be used.

Port	Port 2
Transmission speed	~ 19.2kbps
Handshake method	DC code method, RTS/CTS method possible

This port can be used for inputting/outputting data, and for printing, etc. (The application is designated with the parameters.)

7.2.2 IC Card I/F

7.2.2.1 I/F for IC Card in Control Unit

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

An IC card can be used as an NC data input/output device.
A 2MB or larger, 2GB or smaller flash ATA card (commercially-available part) can be used for the IC card.

The data backed up onto the flash ATA card is stored in DOS format. When using a personal computer compatible with the flash ATA card, the backed up data can be stored on a personal computer's hard disk, etc.

8. Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

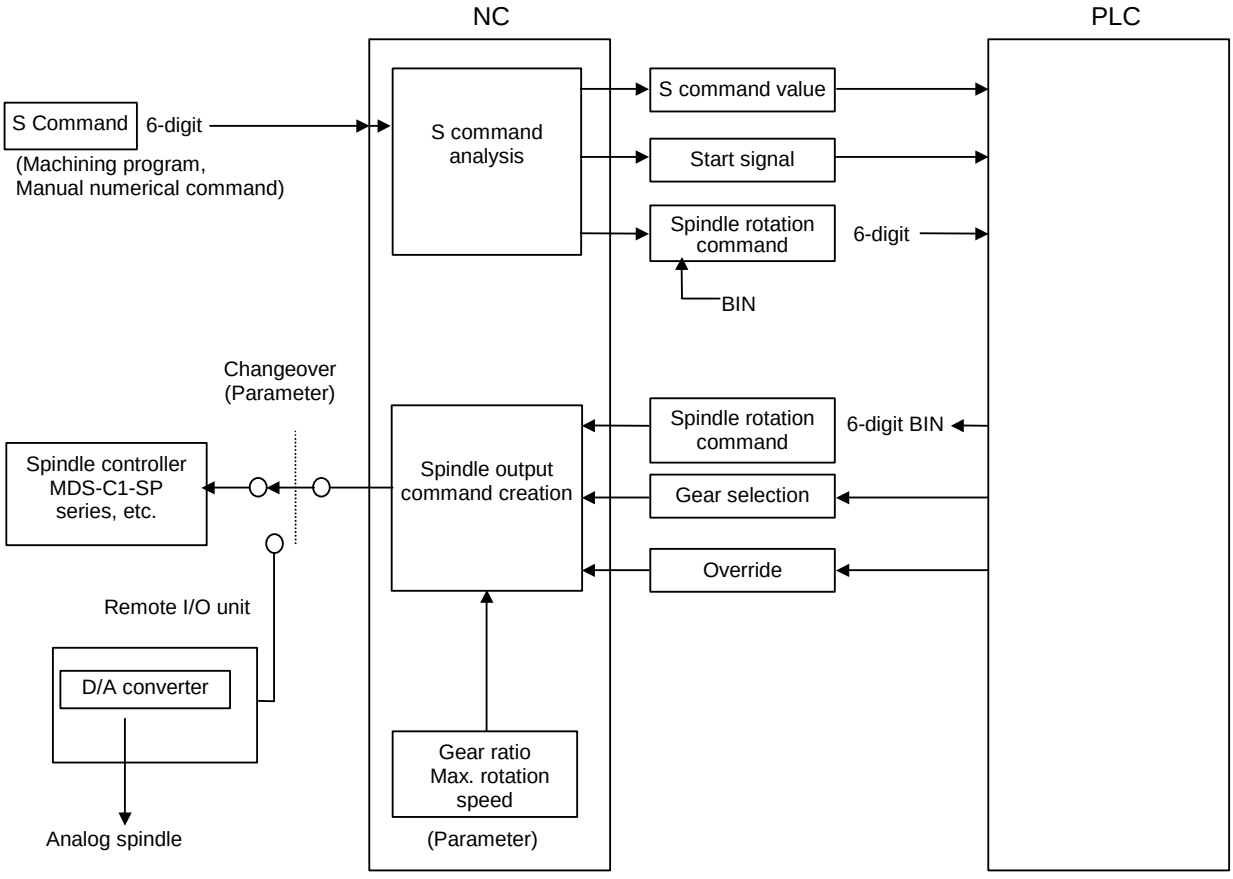
8.1.1 Command/Output

8.1.1.1 Spindle Functions

C6		C64		
T system	L system	M system	L system	T 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 an 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.
Up to seven sets 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

This I/F is used to connect the digital spindle (AC spindle motor and spindle drive unit (SP, SPJ2)).

8.1.1.3 Spindle Analog I/F

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Spindle control can be executed using an analog spindle instead of the digital spindle.
In this case, the remote I/O unit DX120/DX121 is 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 10 V
(2) Resolution ... 1/4095 (–12 multiplier of 2)
(3) Load conditions ... 10 kΩ
(4) Output impedance ... 220 Ω

8.1.1.4 Coil Change

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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

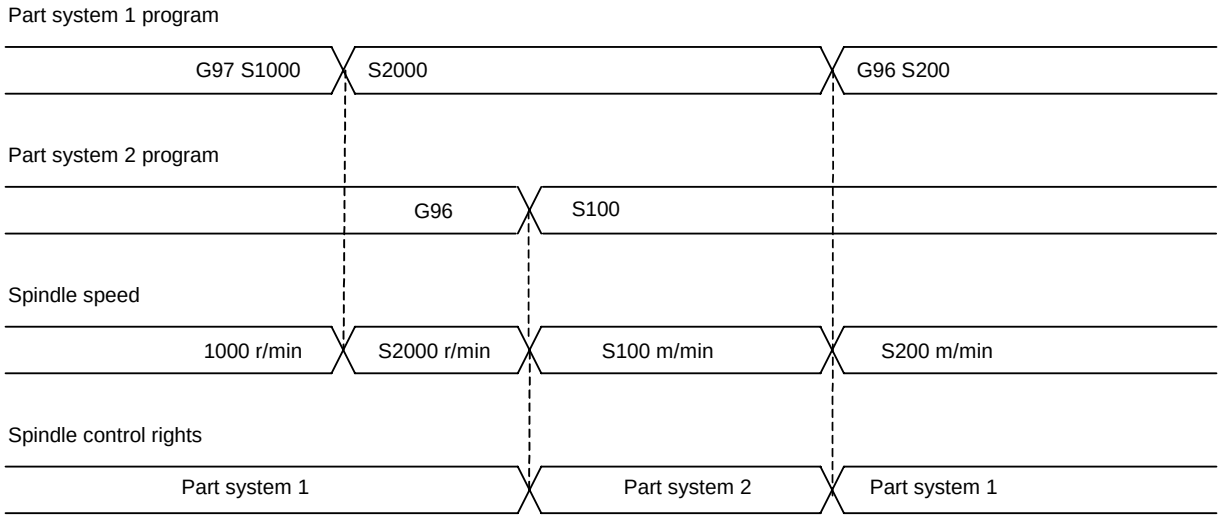
C6		C64		
T system	L system	M system	L system	T system
–	Δ	–	Δ	–

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.

(Note) If there is only one spindle, the spindle will not operate normally if the constant surface speed control command, S command or spindle related M command is commanded randomly from each part system. These commands must be commanded from only one certain part system, or commanded simultaneously with standby.
The controller will execute the following control for the constant surface speed control and S commands. The part system from which an S command was issued last will have the spindle control rights. That part system will judge whether the constant surface speed command mode is valid or canceled, and will execute spindle control.



8.1.2.2 Spindle Override

C6		C64		
T system	L system	M system	L system	T 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 (value 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%.

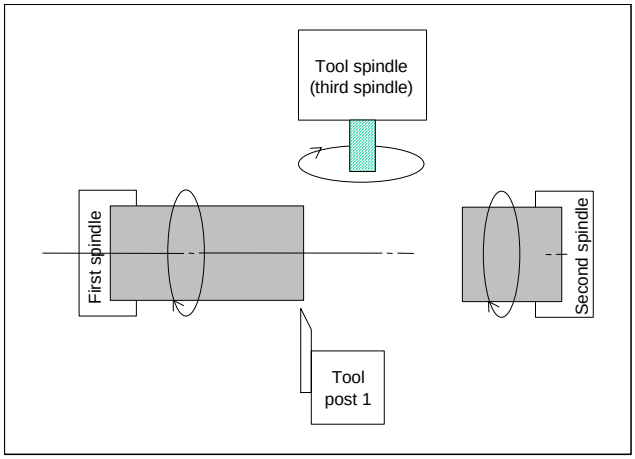
(Note 1) Selection between type 1 and type 2 can be designated by user PLC processing.

8.1.2.3 Multiple-spindle Control

When using a machine tool equipped with several spindles (up to seven spindles), this function controls those spindles.

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

The figure below shows an example of the configuration for a machine which is equipped with second and third spindles.



8.1.2.3.1 Multiple-spindle Control I

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	Δ	–

(1) Spindle selection commands

Using the spindle selection command (such as G43.1 [G group 20]), this function makes it possible to switch the spindle among the first through seventh spindles to which the subsequent S command (S******) is to apply.

Command format

G43.1;	Selected spindle control mode ON; the selected spindle number is set using a parameter.
G44.1;	Second spindle control mode ON

(2) Spindle control commands (using an extended word address (SO=******))

In addition to using the "S******" S commands, it is also possible to assign commands which differentiate the applicable spindle among the first through seventh spindles by using the SO=******.

The S command can be issued from a machining program for any part system.

The number of spindle axes differs according to the model, so check the specifications.

The C6 T and L System and C64 T System cannot control multiple spindles in one part system.

Command format

SO=******;

O : Number assigned as the spindle number (1: first spindle; 2: second spindle; ··· 7: seventh spindle); variables can be designated.

******: Rotational speed or surface speed value assigned by 6-digit analog command; variables can be designated.

8.1.3 Position Control

8.1.3.1 Spindle Orientation

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

(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.)

8.1.3.3 Spindle Synchronization

8.1.3.3.1 Spindle Synchronization I

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	Δ	–

In a machine with two or more spindles, this function controls the rotation speed and phase of one selected spindle (synchronized spindle) in synchronization with the rotation of the other selected 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 H__ D__ R__ A__ ;

H__ : Selects the basic spindle.
D__ : Selects the spindle to be synchronized with the basic spindle.
E__ : Designates the synchronized spindle phase shift amount.
A__ : Designates the spindle synchronization acceleration/deceleration time constant.

8.1.3.3.2 Spindle Synchronization II

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	Δ	–

In a machine with two or more spindles, this function controls the rotation speed and phase of one selected spindle (synchronized spindle) in synchronization with the rotation of the other selected 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.

8.2 Tool Functions (T)

8.2.1 Tool Functions

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

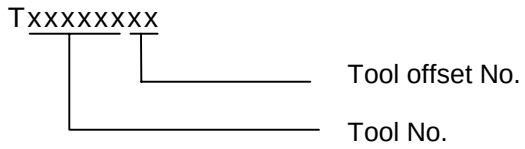
(1) T system, M system

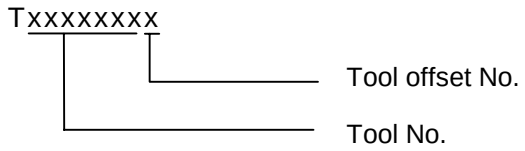
When an 8-digit number following address T (T00000000 – T99999999) is assigned, 8-digit code data and start signal will be output to PLC.
Only one set of T commands can be commanded in a block.
Processing and complete sequences must be incorporated on the PLC side for all T commands.

(Note 1) There are some screens in the setting and display unit that cannot display all eight digits.

(2) L system

The command is issued with an 8-digit number following address T (T0 – 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.





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) There are some screens in the setting and display unit that cannot display all eight digits.

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

When a 2-digit number following address M (M00 – 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 M00000000 to M99999999.
M98 and M99 have specific purposes and can not be used.

(Note 1) There are some screens in the setting and display unit that cannot display all eight digits.

8.3.2 Multiple M Codes in 1 Block

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Four sets of M commands can be issued simultaneously in a block.
Respective processing and completion sequences are required for all M commands included in a block (except M98 and M99).

(Note 1) The code data and start signals of all the M commands in the same block are transferred simultaneously 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

When the M00, M01, M02 or M30 command is assigned during an automatic operation (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 independent output	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.

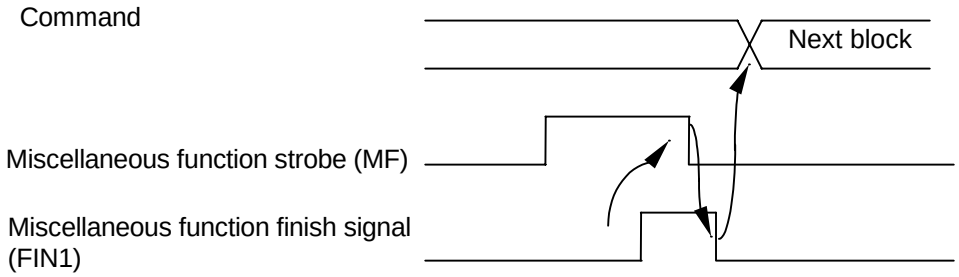
8.3.4 Miscellaneous Function Finish

C6		C64		
T system	L system	M system	L system	T 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).

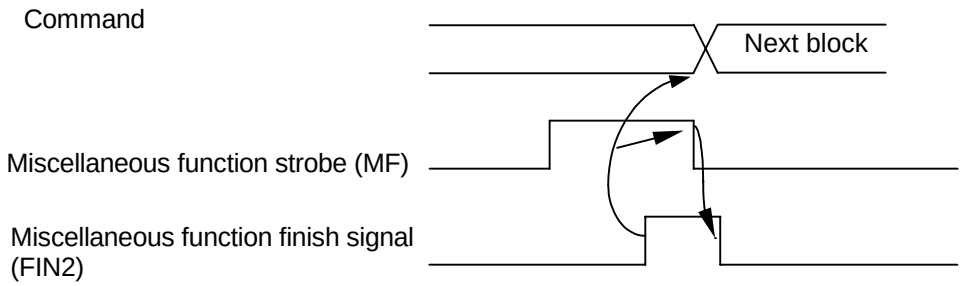
Miscellaneous function finish signal 1 (FIN1)

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.



8.3.5 M Code Output during Axis Positioning

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	Δ	Δ

This function controls the timing at which miscellaneous functions are output, and it outputs a miscellaneous function when axis reaches at the designated position movement.

The command format is as follows.

G117 Xx1 Zz1 Cc1 □□□□ ;

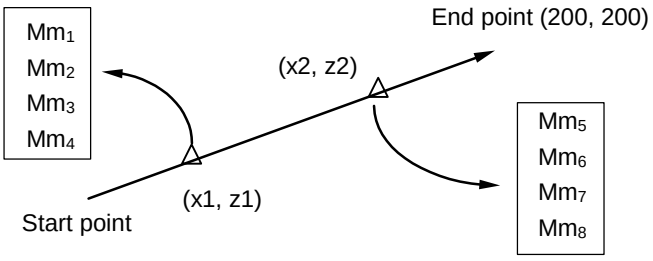
G117 : Command of M code output during axis positioning
Xx1, Zz1, Cc1 : Movement start points
□□□□ : Miscellaneous function

The miscellaneous function can be commanded in the G117 block within the following range.

- M command : Up to four sets
- S command : Up to two sets
- T command : Up to one set
- 2nd miscellaneous function command : Up to one set

The G117 command can be commanded in up to two continuous blocks.

(Example) G117 Xx_1 Zz_1 Mm_1 Mm_2 Mm_3 Mm_4 ;
G117 Xx_2 Zz_2 Mm_5 Mm_6 Mm_7 Mm_8 ;
G01 X200 Z200 ;



8.4 2nd Miscellaneous Function (B)

8.4.1 2nd Miscellaneous Function

C6		C64		
T system	L system	M system	L system	T 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) There are some screens in the setting and display unit that cannot display all eight digits.

9. Tool Compensation

9.1 Tool Length/Position Offset; G43 to G49

9.1.1 Tool Length Offset

C6		C64		
T system	L system	M system	L system	T 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) T system, 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)

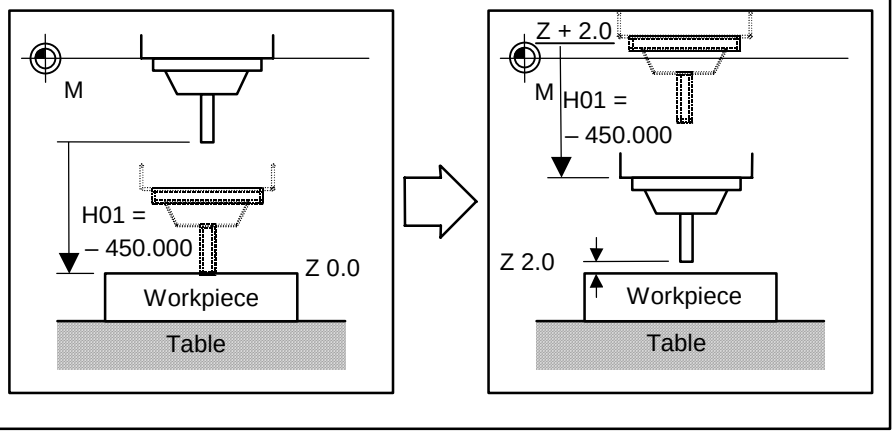
Offset can be canceled by the following G commands.
G49;
G43 H0;
G44 H0;

(Note) When the tool length offset axis is returned to the reference point, 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 ;
```

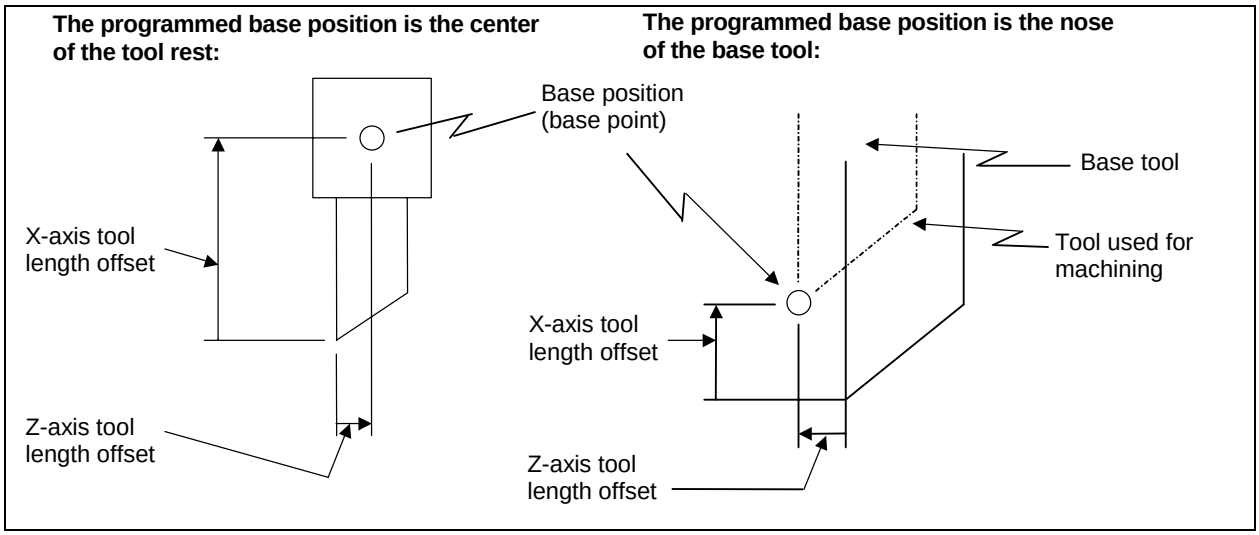
(Note) The tool length offset amount is set as a negative value such as H01 = –450.000.



(2) L system

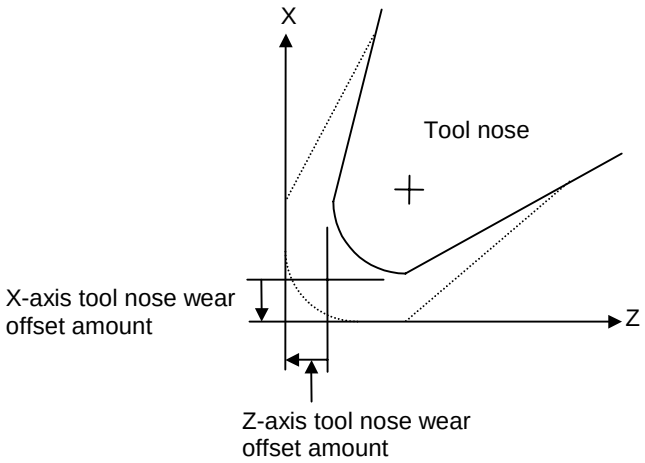
(a) Shape offset

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

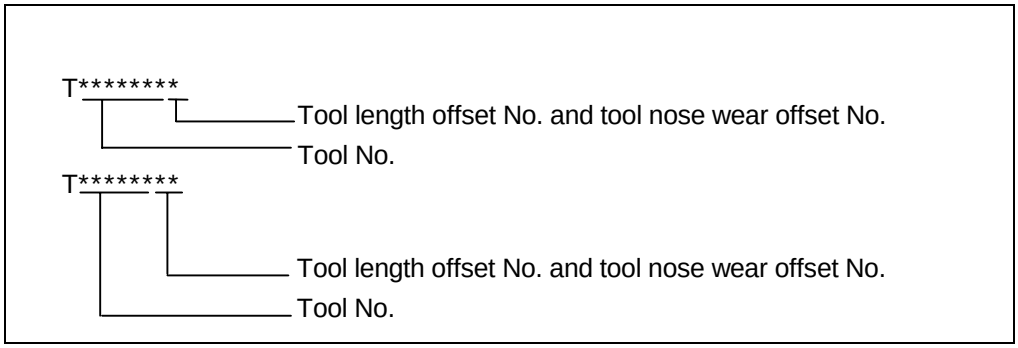
The wear of a tool nose can be offset.



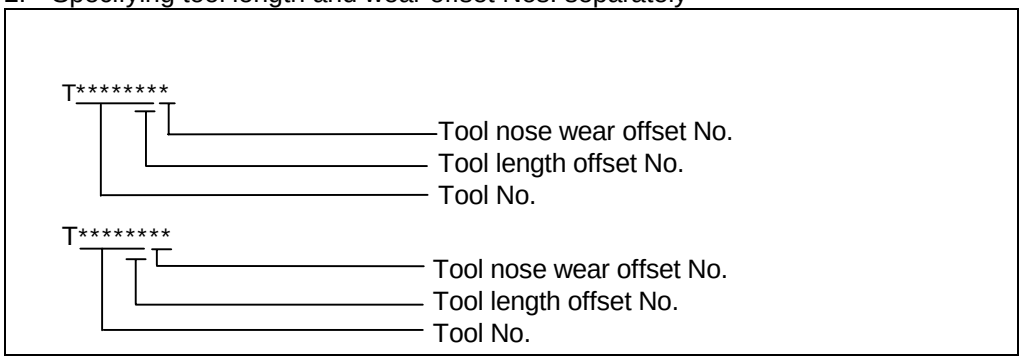
(c) Command format

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 L system is valid only for the X and Z axes. If an additional axis (Y axis) is added, the tool offset will be validated for the additional axis. (Refer to 9.1.3.)

9.1.3 Tool Offset for Additional Axes

C6		C64		
T system	L system	M system	L system	T system
○	–	○	–	○

The tool offset for the L system is valid only for the X and Z axes. If an additional axis (Y axis) is added, the tool offset will be validated for the additional axis.
The additional axis is the third or fourth axis which is selected using a parameter.

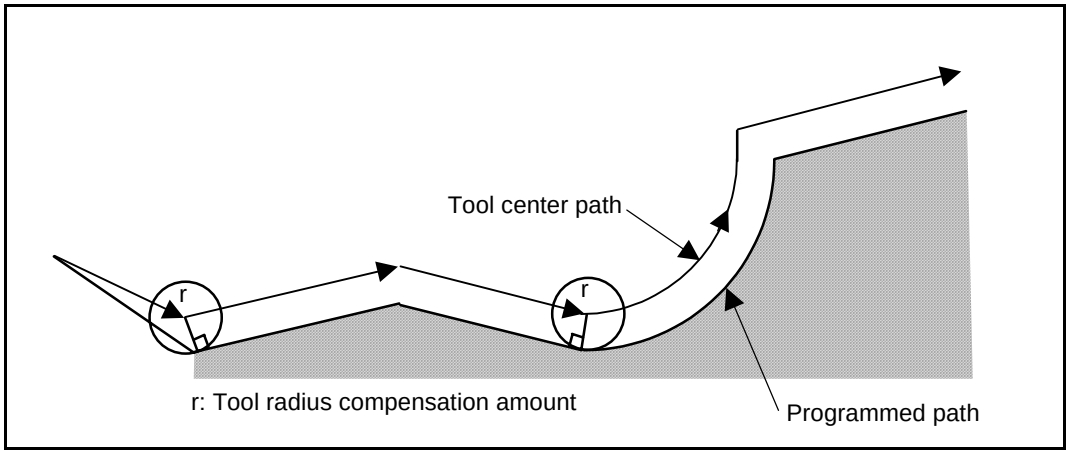
9.2 Tool Radius; G38 to G42, G46

9.2.1 Tool Radius Compensation; G38 to G42

C6		C64		
T system	L system	M system	L system	T 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 G01 G41 Xx1 Yy1 Dd1 ;

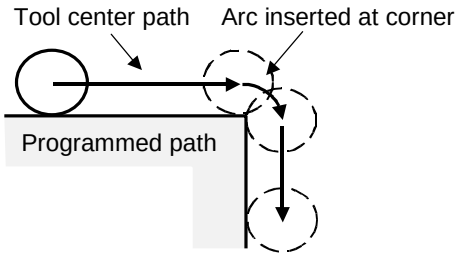
G17	: Compensation plane
G01	: Cutting command
G41	: Left compensation
Xx1,Yy1	: Movement axis
Dd1	: Compensation No.

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



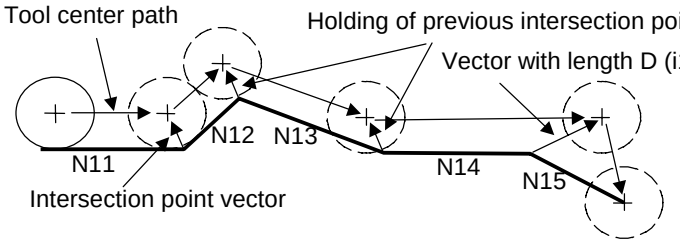
The compensation vector can be changed in following two ways.

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.

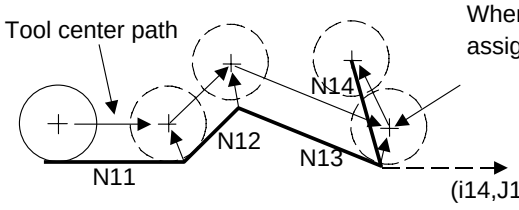


N11G01Xx11;
N12G38Xx12Yy12;
N13G38Xx13Yy13;
N14G38Xx14Ii14Jj14Dd14;
N15G40Xx15Yy15;

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.



When i and j commands are assigned to G40

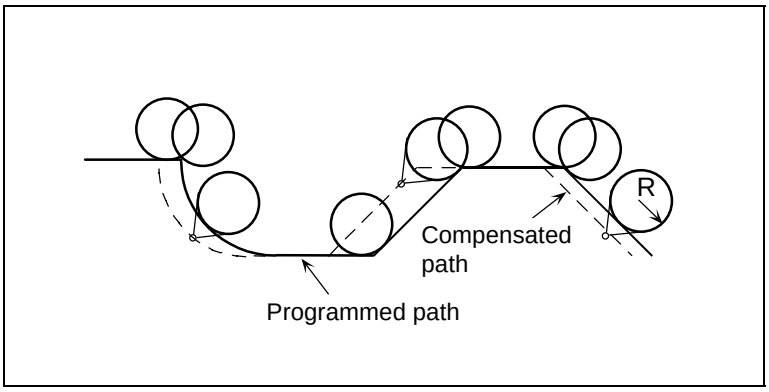
N11G01Xx11;
N12Xx12Yy12;
N13Xx13Yy13;
N14G40Xx14Ii14Jj14;

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

C6		C64		
T system	L system	M system	L system	T 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.

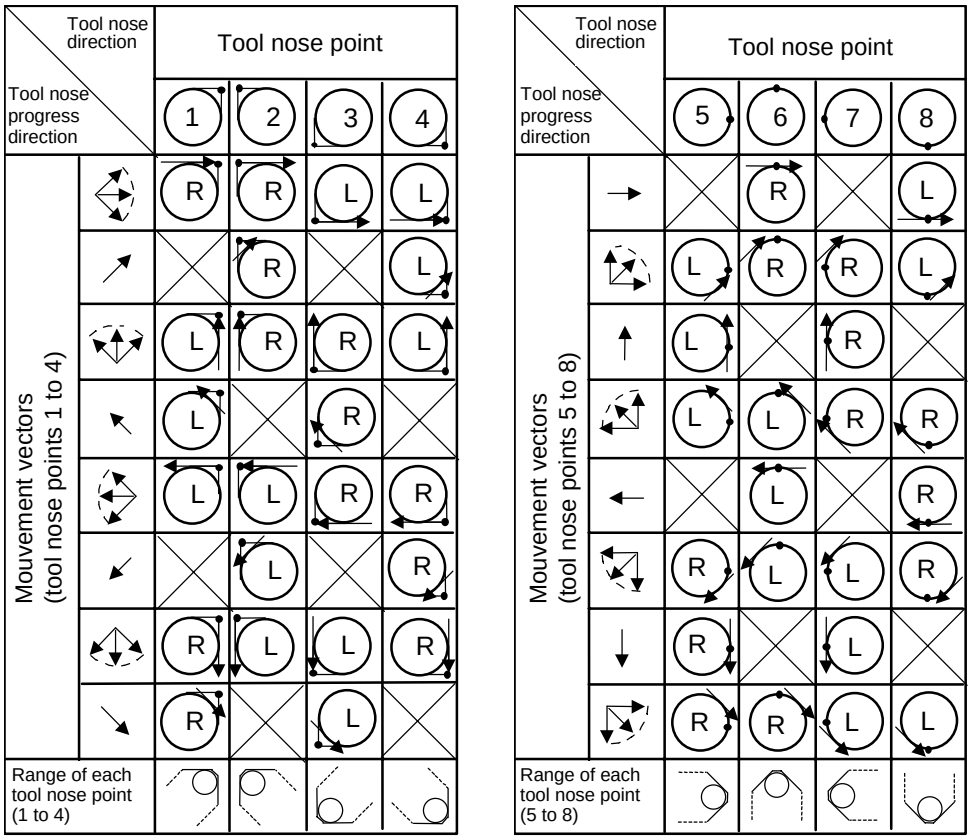
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

C6		C64		
T system	L system	M system	L system	T 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.

9.3.1.2 40 sets

C6		C64		
T system	L system	M system	L system	T system
○	–	○	–	○

9.3.1.3 80 sets

C6		C64		
T system	L system	M system	L system	T system
Δ	○	Δ	○	Δ

9.3.1.4 100 sets

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

9.3.1.5 200 sets

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

C6		C64		
T system	L system	M system	L system	T 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 [T system, 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.

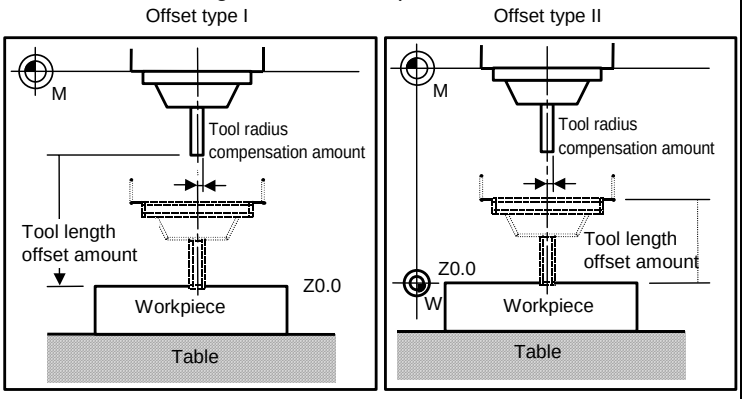
(b) Type 2: 1-axis offset amounts/with wear offset [T system, M system]

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

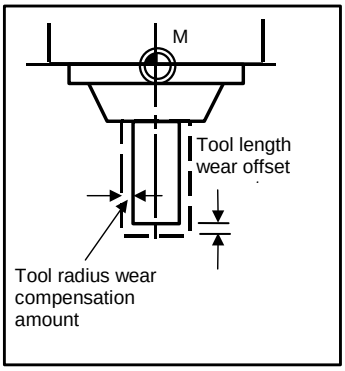
With type 2, four kinds of offset amount data are registered in one offset No.: the tool length offset amount, tool length wear offset amount, tool radius compensation amount, and tool radius wear compensation amount.

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.)



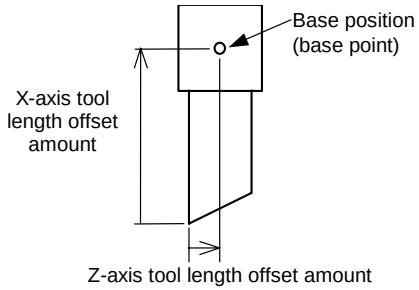
Wear offset amount when using type 2



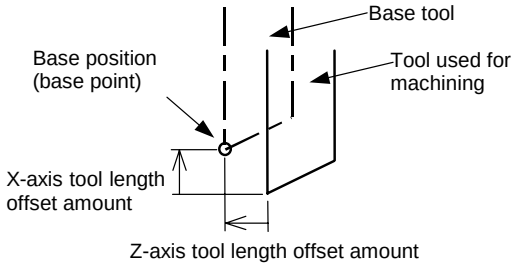
(c) Type 3: 2-axis offset amounts [L system]

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.

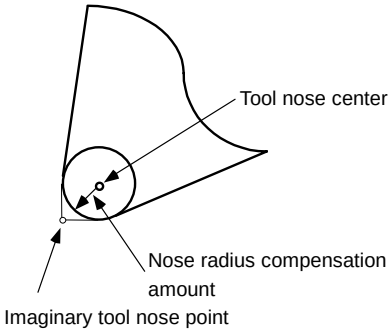
1. The programmed base position is the center of the tool rest:

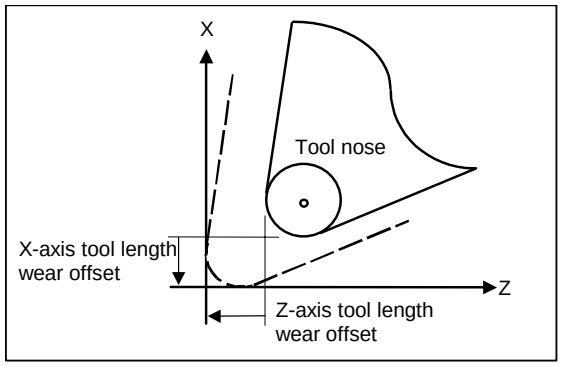


2. The programmed base position is the tip of the base tool:



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.



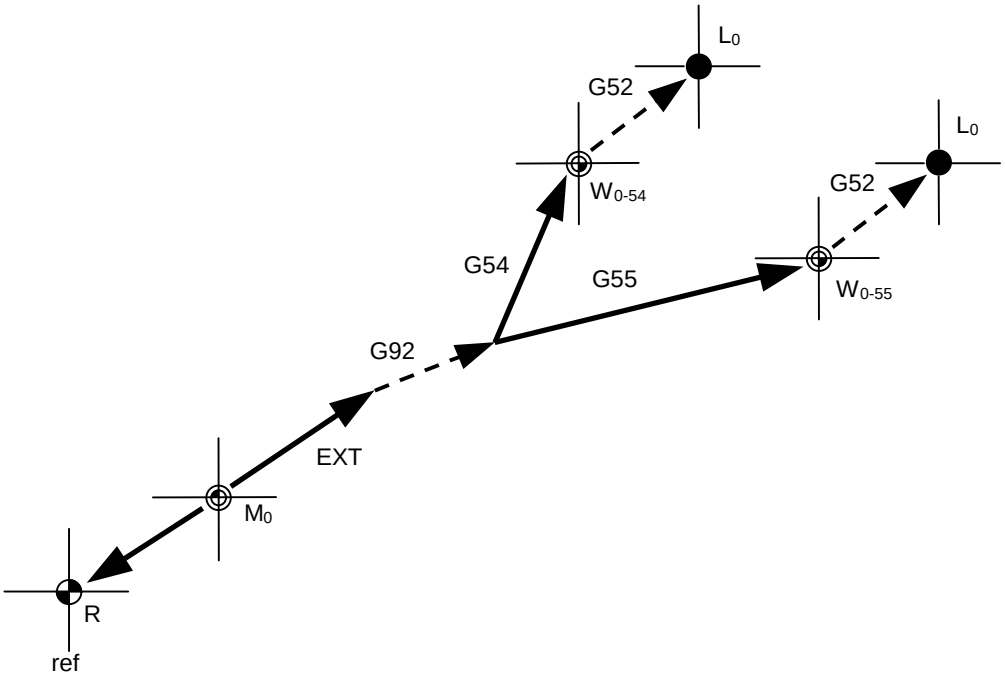


The X-axis tool length offset amount, Z-axis tool length offset amount and nose radius compensation amount are set as plus amounts.
The offset type (1, 2 or 3) is set using a parameter.

10. Coordinate System

10.1 Coordinate System Type and Setting; G52 to G59, G92

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	——→	Offset set with parameters
G52	Local coordinate system offset *1)	- - - -→	Offset set with program
W_{0-54}	Workpiece coordinate system zero point (G54)		(0 when power is turned ON)
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 point		

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

10.1.1 Machine Coordinate System; G53

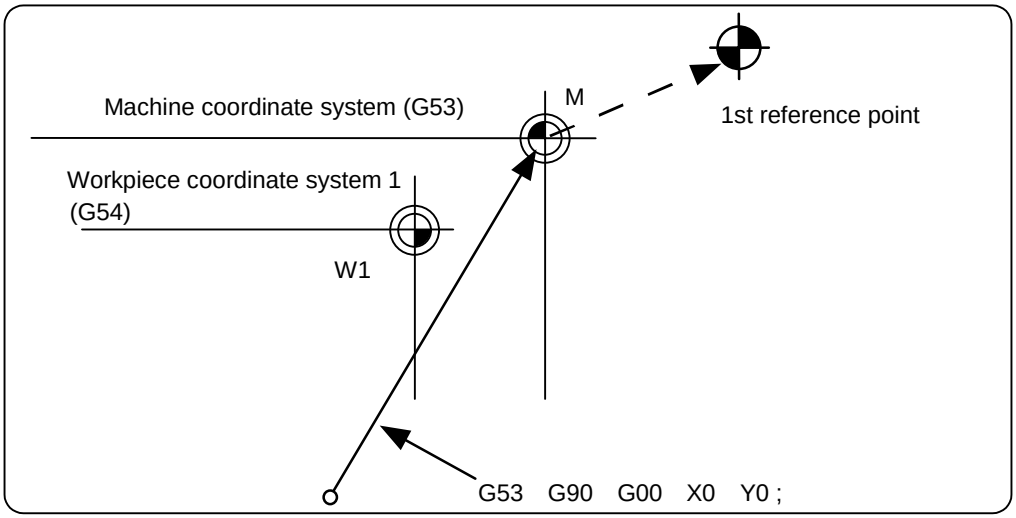
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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 point 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 [T system, 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 which is effective only in the block where it is assigned. The workpiece coordinate system being selected is not changed by this command.

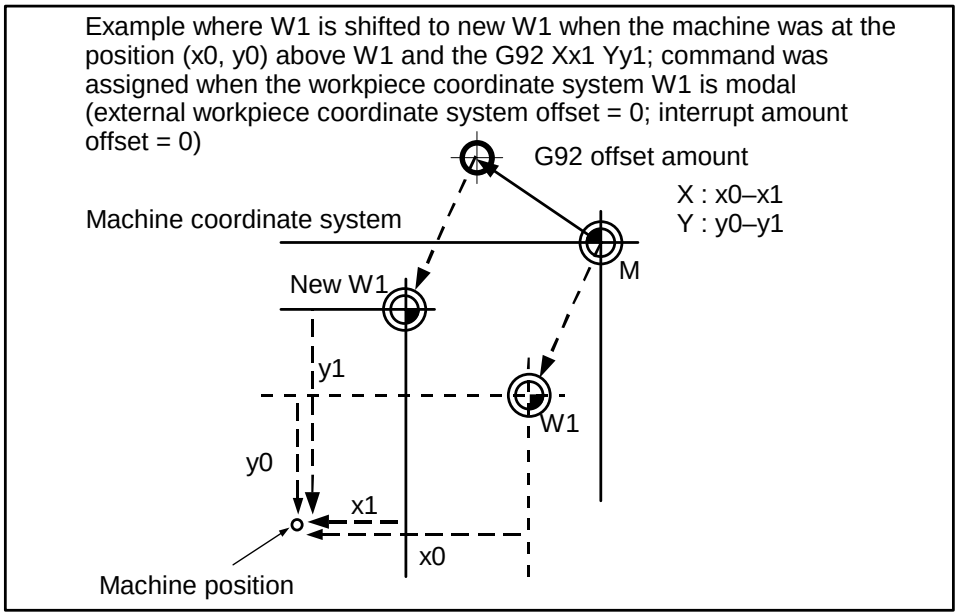


10.1.2 Coordinate System Setting; G92

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

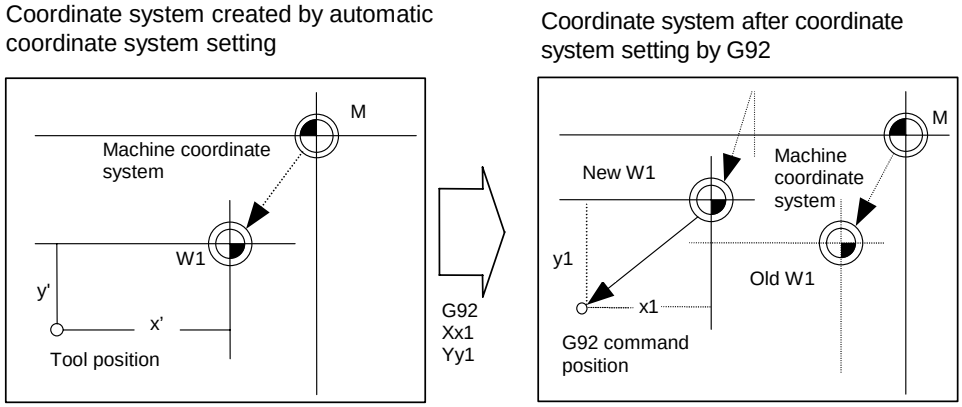
When a coordinate system setting is assigned using the G92 command, the G92 offset amount is applied so that the machine position in the current workpiece coordinate system is set to the coordinate values assigned by the G92 command, as shown in the figure below, and the workpiece coordinate systems are shifted accordingly. The machine does not run , 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 point 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.



(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) Carry out dog-type reference point 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

When the tool has arrived at the reference point by means of the first manual or automatic dog-type reference point 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 system
(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. Thus, where the No. 1 reference point is set in the machine is the base for the setting.

10.1.4 Workpiece Coordinate System Selection (6 sets); G54 to G59

C6		C64		
T system	L system	M system	L system	T 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.

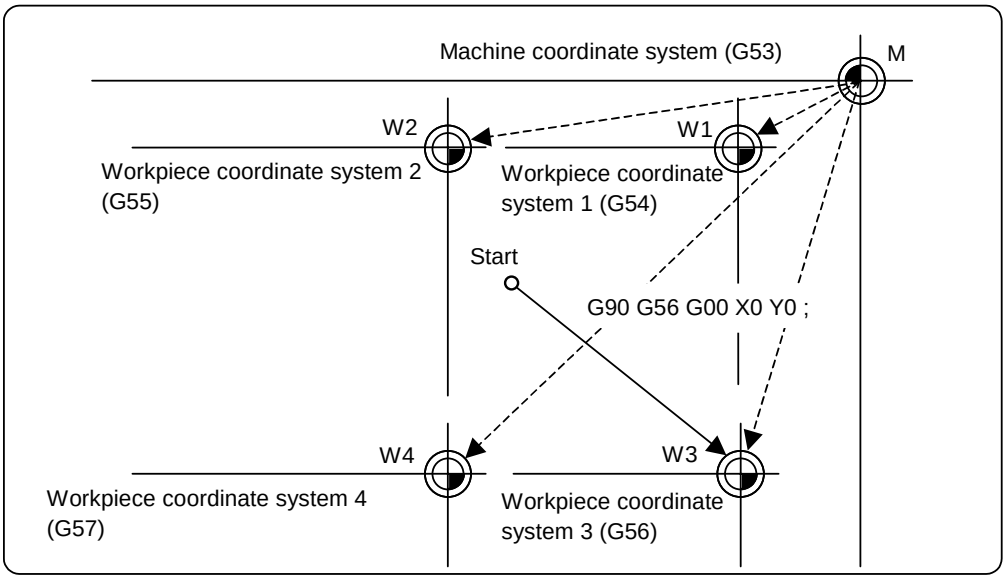
G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

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 values of end point

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



10.1.5 Extended Workpiece Coordinates System Selection

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

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

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 value command)
G54.1Pn : Coordinate system selection
G00 : Movement mode
Xx1, Yy1, Zz1 : Coordinate values 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.

10.1.7 Local Coordinate System; G54G52 to G59G52

C6		C64		
T system	L system	M system	L system	T 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.

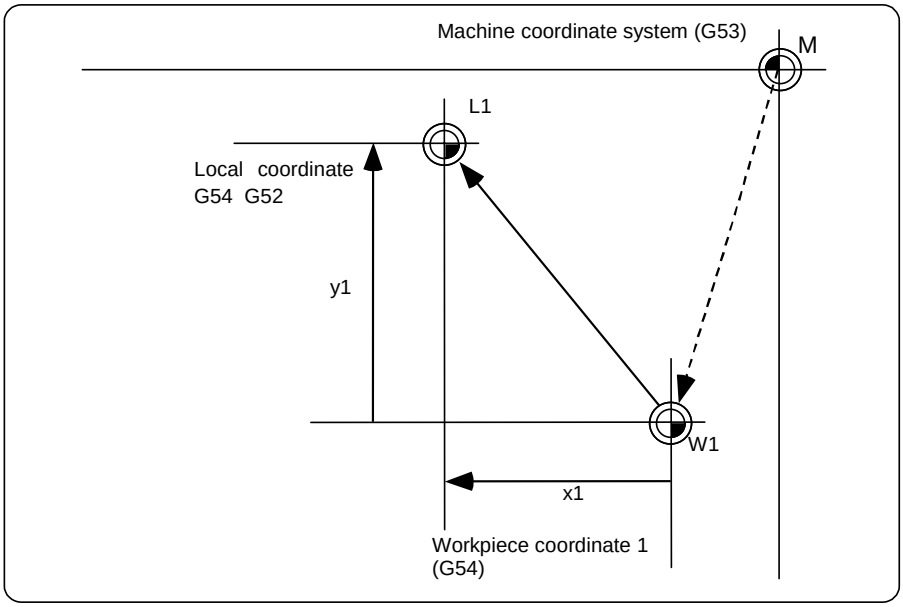
G code	Function
G54 G52	Local coordinate system on the workpiece coordinate system 1
G55 G52	Local coordinate system on the workpiece coordinate system 2
G56 G52	Local coordinate system on the workpiece coordinate system 3
G57 G52	Local coordinate system on the workpiece coordinate system 4
G58 G52	Local 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 system selection
G52 : Local coordinate system 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 point return.



10.1.8 Coordinate System for Rotary Axis

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The coordinate system of rotary axis ranges from 0 to ±360°. Note that, however, it can be displayed from 0 to 359.999.
In absolute value command mode, the rotary axis can make a turn or less (not greater than ±360°). The turning direction depends on the specified sign. A negative sign (–) turns the axis in the negative direction and a positive sign (+) turns it in the positive (+) direction.
Note that a parameter can be used to move the axis to the end point taking a short cut.
In incremental value command mode, the rotary axis moves the specified distance only.

10.1.9 Plane Selection; G17 to G19

C6		C64		
T system	L system	M system	L system	T 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.

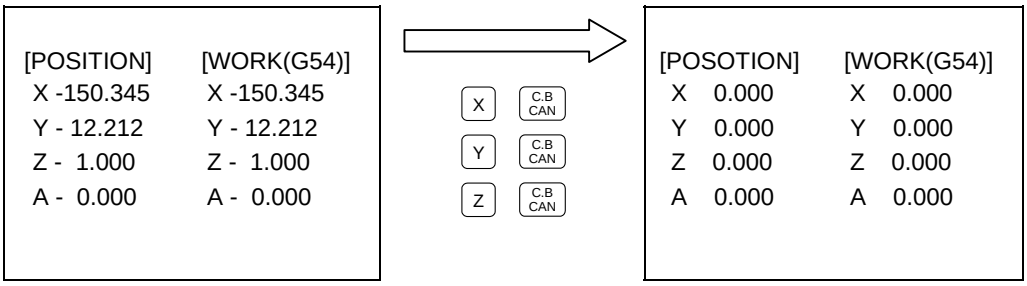
Example

G19 X100. ;	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Using the setting and display unit, 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) ; ".



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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Using the setting and display unit, 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; G27 to G30

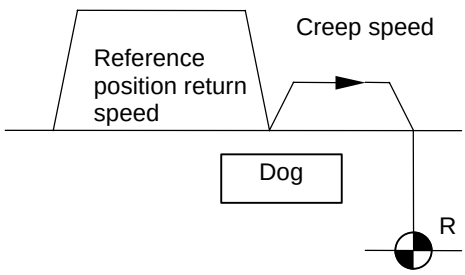
10.2.1 Manual Reference Point Return

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

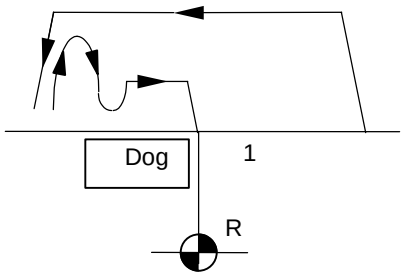
This function enables the tool to be returned manually to the position (reference point) which is characteristic to the machine.

(1) Return pattern to reference point

(a) Dog type

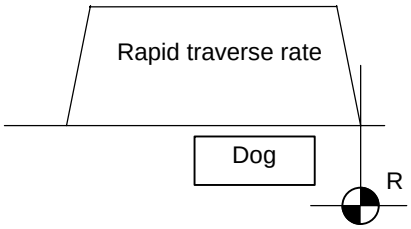


When starting in same direction as final advance direction



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 Point Return; G28, G29

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The machine can be returned to the first reference point 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 point.
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 point return
G29	Start position return (The tool first returns to the interim position of the 1st reference point return start from the 1st reference point, 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)

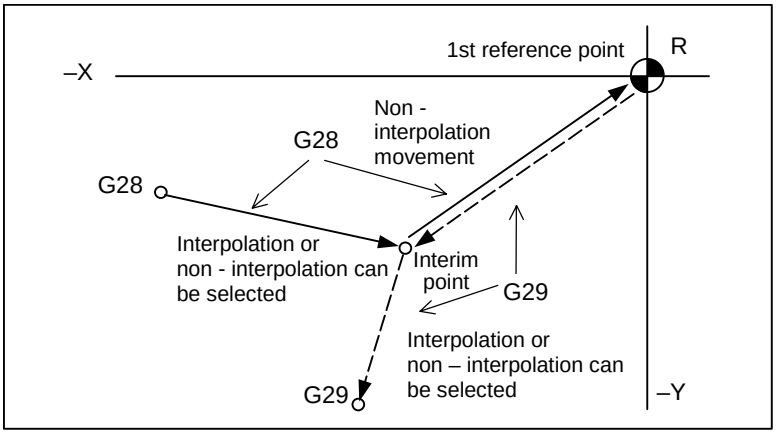
Each axis is first positioned by rapid traverse to the position (interim point) assigned for the assigned axis and then is returned independently to the 1st reference point.

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 which 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 point 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 point return pattern is the same as for manual reference point return.

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

(Note 3) If, at the time of the first reference point 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 point 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 point, 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 point 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 Point Return; G30

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

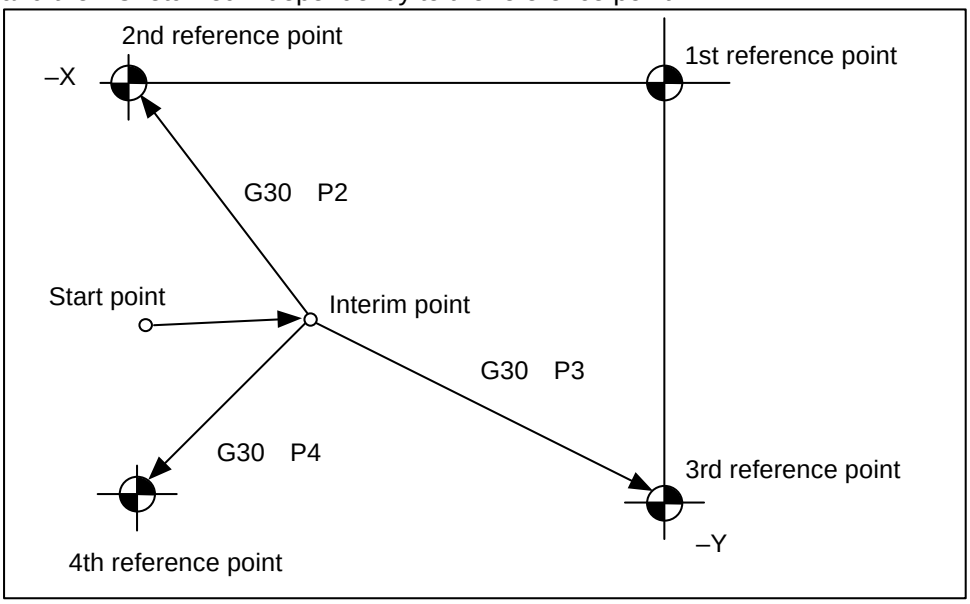
G code	Function
G30 P2	2nd reference point return
G30 P3	3rd reference point return
G30 P4	4th reference point return

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 point.



(Note 1) The second reference point return is performed if the P address is omitted.

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

(Note 3) If, at the time of the reference point 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 point return, the tool length offset has not been canceled, it will be canceled and the offset amount also cleared upon completion of reference point 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 points.

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

10.2.4 Reference Point Verification; G27

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

By commanding G27, a machining program, which has been prepared so that the tool starts off from the reference point and returns to the reference point, can be checked to see whether the tool will return properly to the reference point.
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 point verification
P2 : 2nd reference point verification
P3 : 3rd reference point verification
P4 : 4th reference point verification

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

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

(Note 1) The number of axes for which reference point 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 point 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	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 point return. (High-speed return will always be used for the reference point return command.)
For the absolute position detection method, there are two method 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 point 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 Change Position Return; G30.1 to G30.6

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

[T system, 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)

(Note 1) 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" indicate that the Z axis returns to the tool change position, then the X axis does.

(Note 2) G30.6 is only for the T system and 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.

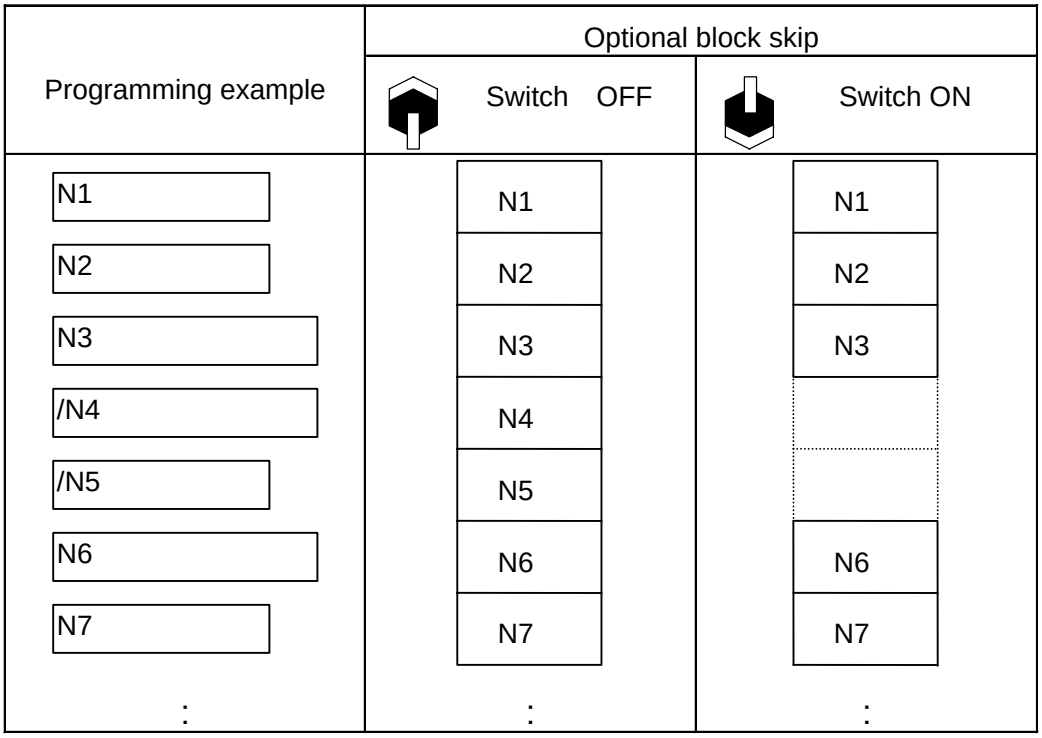
11. Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

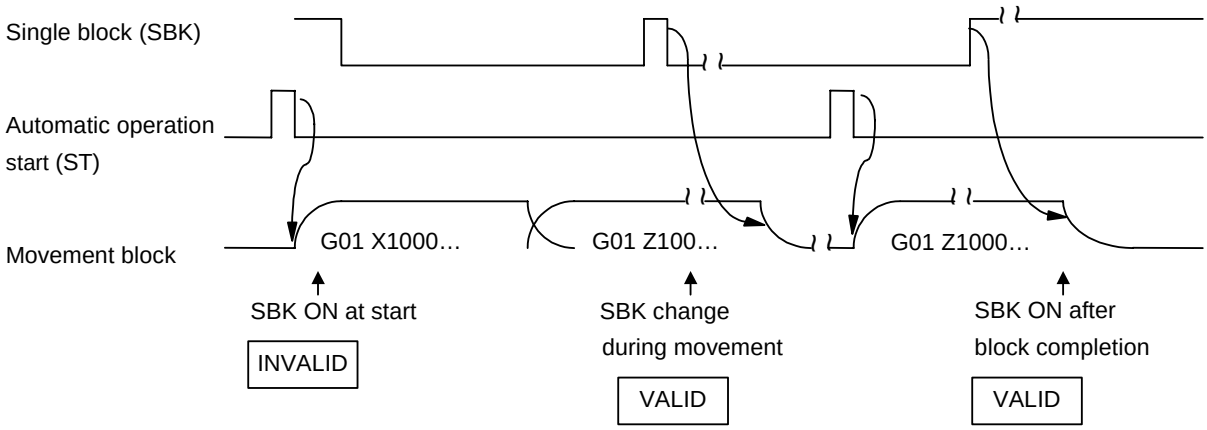
When "/" (slant code) is programmed at the head of a block, and the optional block skip input signal from the external source is turned ON for automatic operation, the block with the "/" code is skipped. If the optional block skip signal is turned OFF, the block with the "/" code will be executed without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



11.1.3 Single Block

C6		C64		
T system	L system	M system	L system	T 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.
Single block in the multi-part system also functions as the above single block in each independent part system.



11.2 Program Test

11.2.1 Dry Run

C6		C64		
T system	L system	M system	L system	T 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 selector switch OFF	Rapid traverse selector 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

When the machine lock input signal is set to ON, the NC operations can be executed without assigning commands to the NC axes.
Either the machine lock speed or command speed can be selected using a parameter as the feed rate during machine lock.
The M, S, T and B commands are executed as usual, and so machine lock is completed by returning the FIN signal.

(1) Reference point return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but when the interim point is reached the counter is moved to the zero point and the block is completed.
(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. (Using a parameter, the machine lock signal can be made to take effect immediately.)
(4) Whether the POSITION counter is to be held or the movement amount operated by machine lock is to be canceled when resetting is initiated during machine lock can be selected using a parameter.

11.2.3 Miscellaneous Function Lock

C6		C64		
T system	L system	M system	L system	T 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 which 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.

11.3 Program Search/Start/Stop

11.3.1 Program Search

C6		C64		
T system	L system	M system	L system	T 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

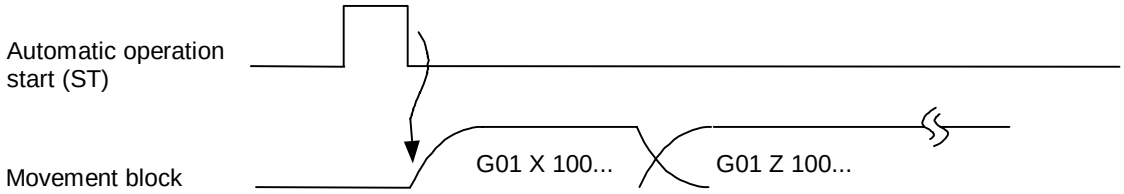
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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.
Machining programs are stored in the memory inside the NC system.

11.3.5 Automatic Operation Start

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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



Automatic operation startup is performed on a part system by part system basis.

11.3.6 NC Reset

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

This function enables the controller to be reset.

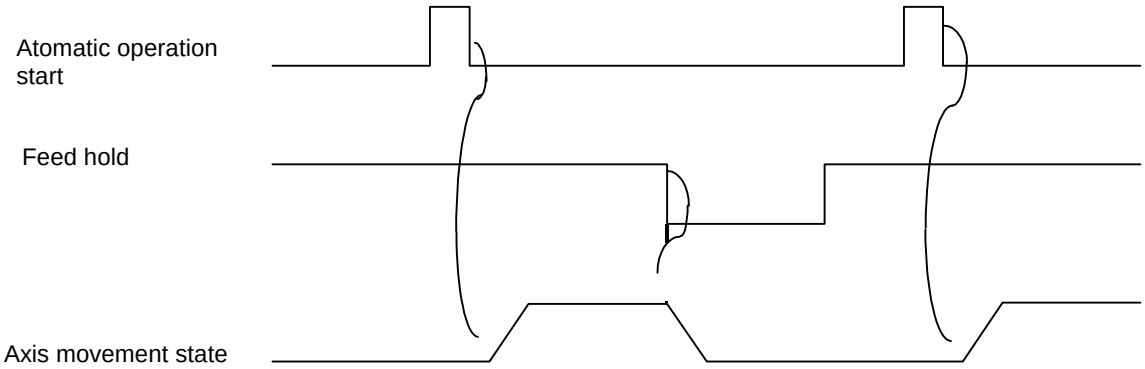
	PLC signal name / Target	Reset 1	Reset 2	Reset & Rewind
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	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

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Manual interrupt is a function that enables manual operations to be performed during 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

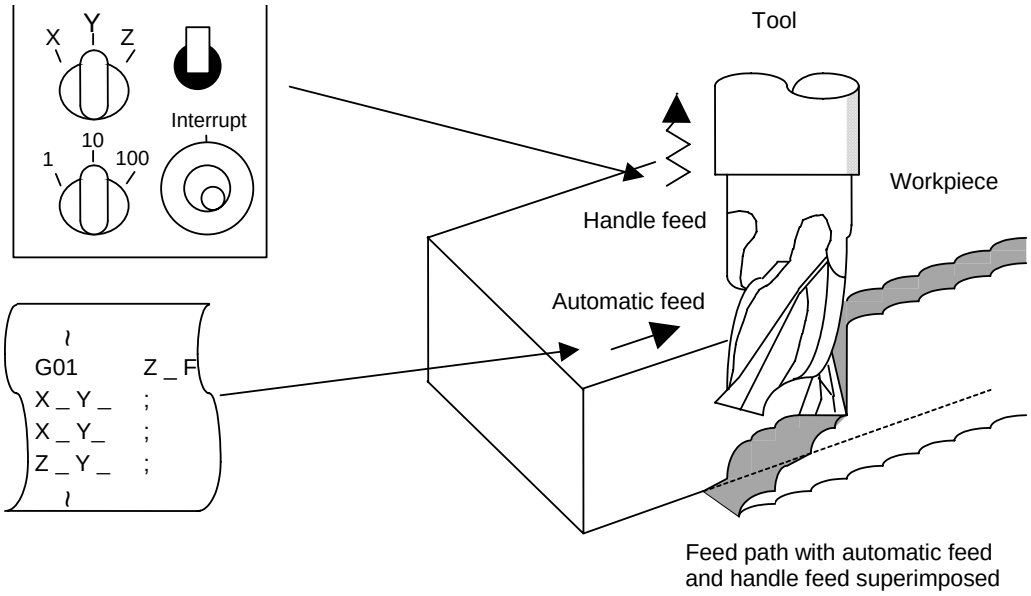
  (Refer to "11.4.9 Simultaneous Operation of Manual and Automatic Modes".)

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 "11.4.3 Manual Absolute Mode ON/OFF").

11.4.2 Automatic Operation Handle Interruption

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

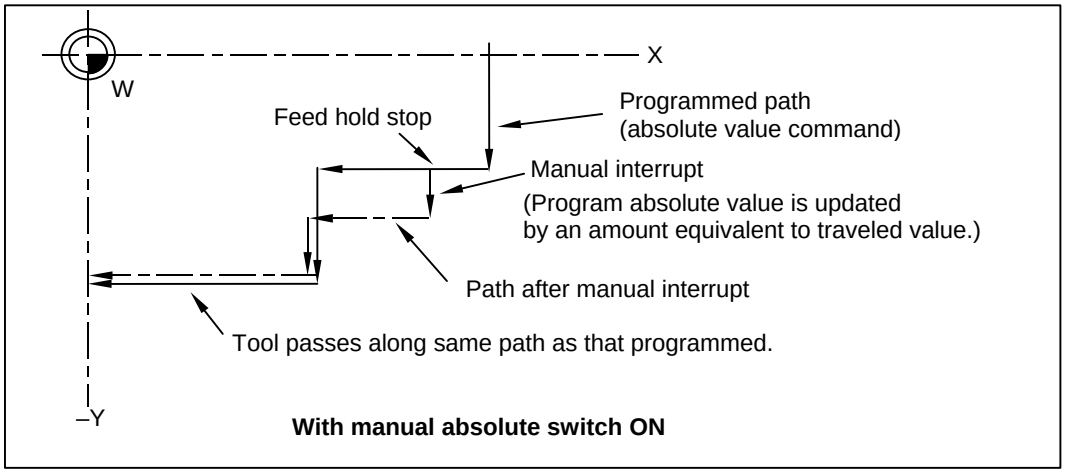
The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during automatic operation.
If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due 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 which is 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.



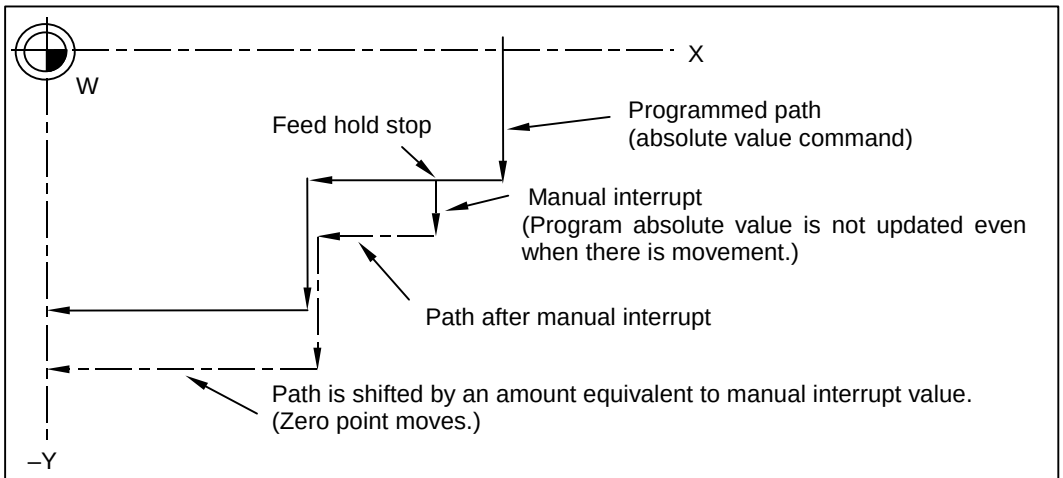
11.4.3 Manual Absolute Mode ON/OFF

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The program absolute values 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



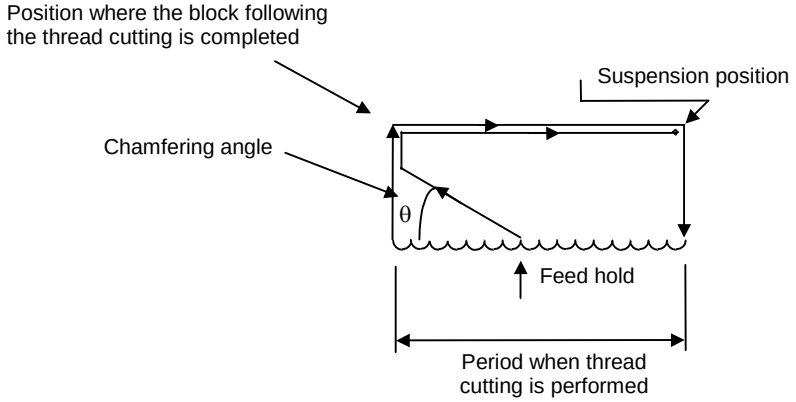
With manual absolute switch OFF

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

11.4.4 Thread Cutting Cycle Retract

C6		C64		
T system	L system	M system	L system	T system
–	Δ	–	Δ	–

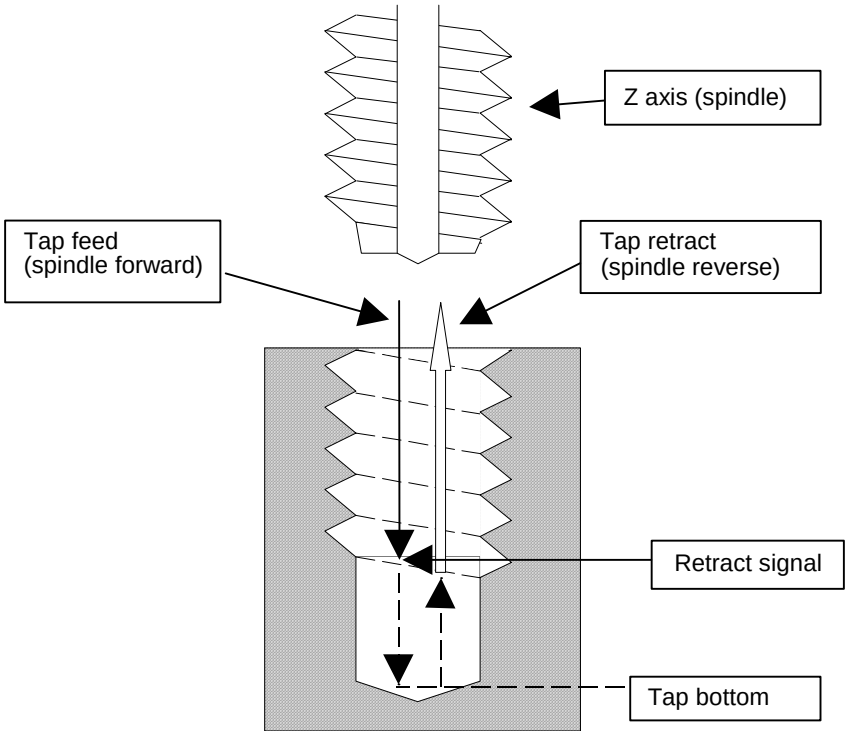
This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting in a thread cutting cycle.
If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is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



11.4.5 Tapping Retract

C6		C64		
T system	L system	M system	L system	T 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.

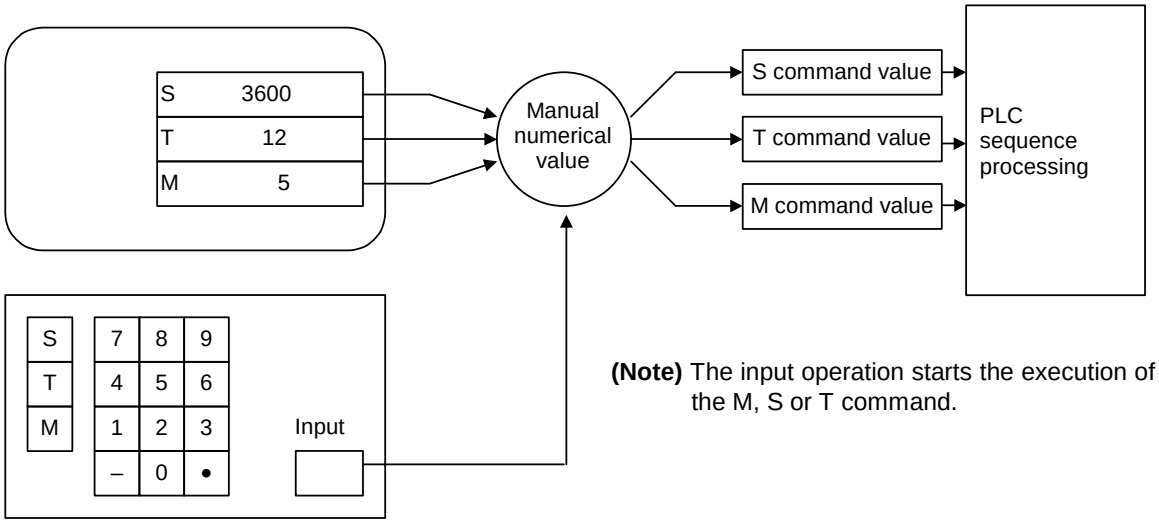


This function can be used by an interruption initiated by reset or emergency stop.
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical Value Command

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

On the screen of the setting and display unit, 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

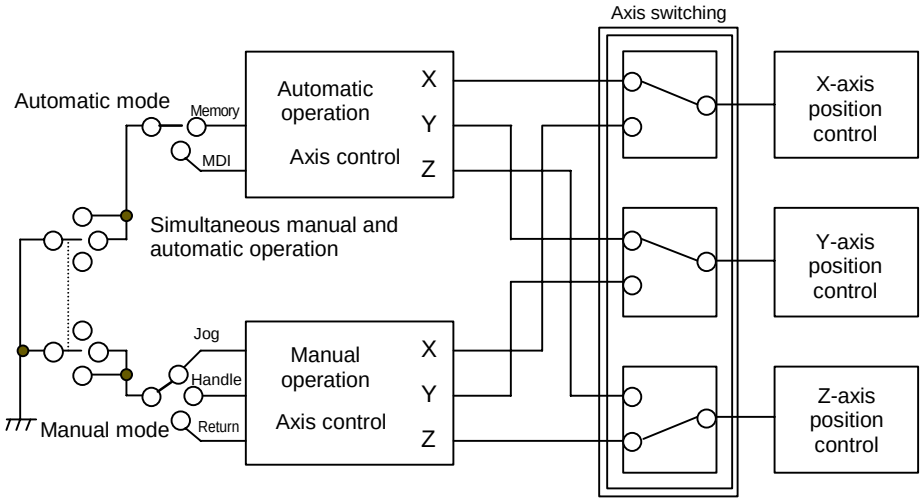
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

This function enables manual operations to be performed during automatic operation by selecting an automatic operation mode (MDI or memory) and manual mode (handle, step, jog or manual reference point 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

C6		C64		
T system	L system	M system	L system	T 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 Point Retract

C6		C64		
T system	L system	M system	L system	T 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 point.
The reference point to be retracted to can be selected from the 1st reference point to 4th reference point 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 point 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 can not be resumed.
(f) The retract signal is invalid if the coordinate system is not established. An operation error will occur when the retract signal is input in such case.

12. Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

12.1.1.1 Subprogram Control

C6		C64		
T system	L system	M system	L system	T system
○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers	○ 8 layers

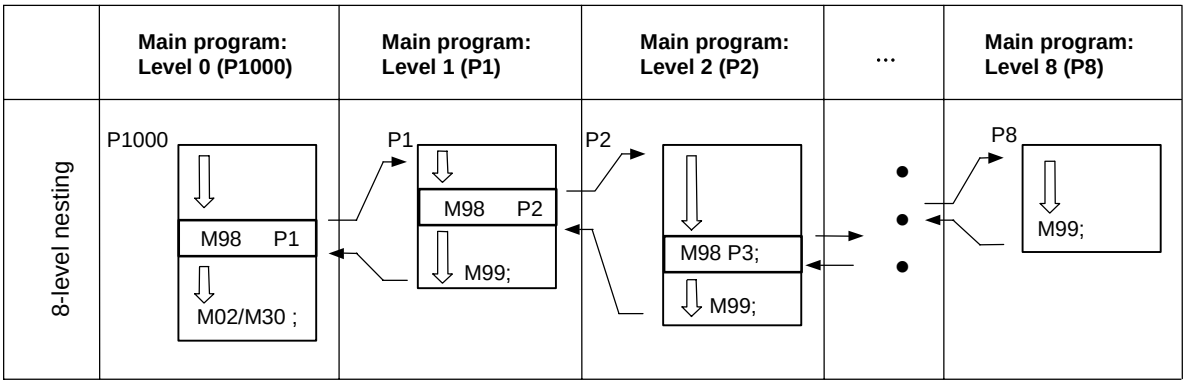
When the same pattern is repeated during machining, the machining pattern is registered as one subprogram and the subprogram is called from the main program as required, thereby realizing the same machining easily. Efficient use of program can be made. The call is designated with the program number and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

M98 Pp1 Hh1 Ll1 ;

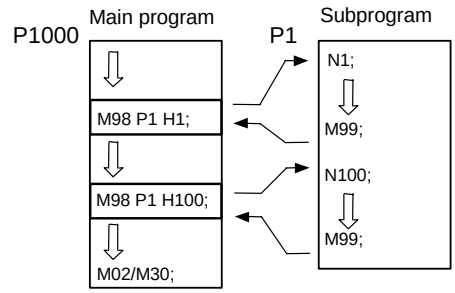
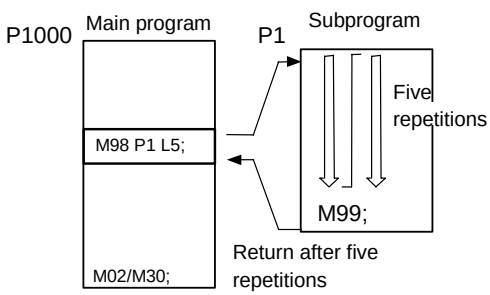
M98 : Call command
Pp1 : Subprogram number
Hh1 : Sequence number
Ll1 : Number of repetitions

(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
	

12.1.2 Macro Program

12.1.2.1 User Macro

C6		C64		
T system	L system	M system	L system	T system
Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 4 layers	Δ 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 which 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.

1. 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.)

(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.
Select codes to be entered which are not the codes basically required by the machine and which are not M codes M0, M1, M2, M30 and M96 through M99.

(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.)

(Note 2) The registration of the program number used for calling the G code macro or M code macro can be done independently for each system. [T system, M system]

12.1.2.3 Macro Interruption

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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.

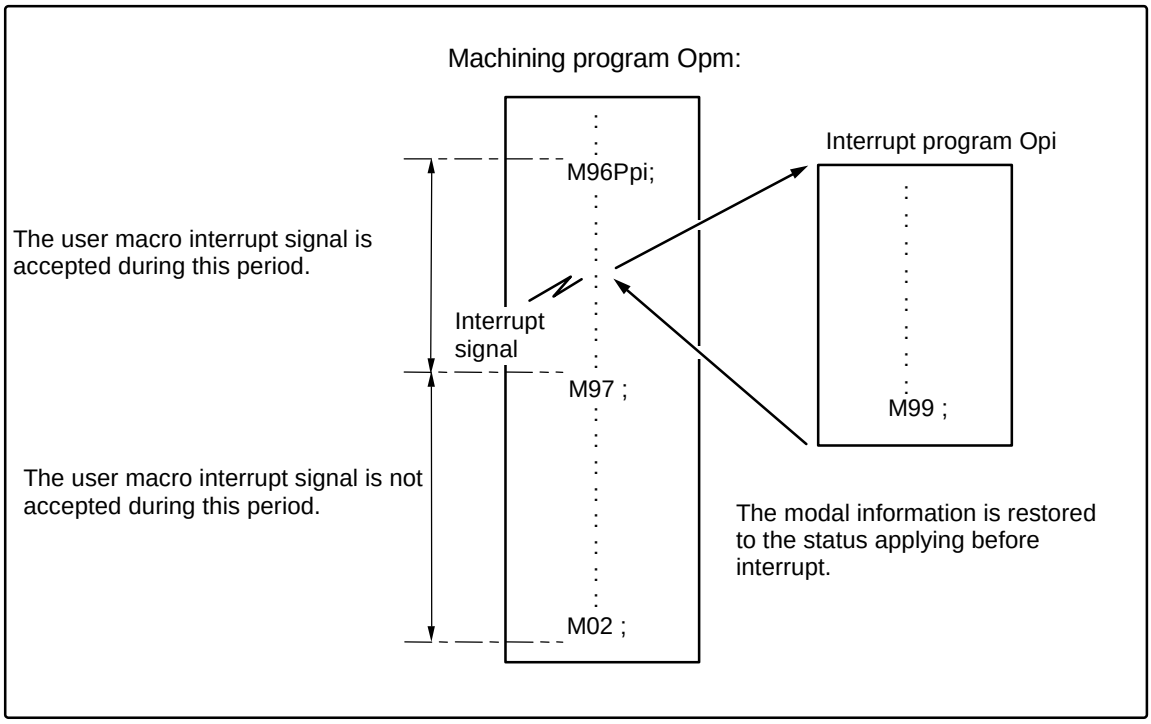
[Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.

[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

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.

Variable set option	Variables common to all part systems	Variables for each part system
(50+50 × number of part systems) sets	#500 ~ #549 (50 sets)	#100 ~ #149 (50 sets)
(100+100 × number of part systems) sets	#500 ~ #599 (100 sets)	#100 ~ #199 (100 sets)
(200+100 × number of part systems) sets	#500 ~ #699 (200 sets)	#100 ~ #199 (100 sets)

2. 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".

12.1.2.4.6 (50+50 x number of part systems) sets

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

12.1.2.4.7 (100+100 x number of part systems) sets

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

12.1.2.4.8 (200+100 x number of part systems) sets

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

12.1.3 Fixed Cycle

List of fixed cycles

Type of fixed cycle	T system, M system	L system		Remarks
	G code system 1	G code system 2	G code system 3	
Fixed cycle for drilling	G70 : G89	G80 : G89	G80 : G89	Refer to 12.1.3.1. Refer to 4.5.3.
		G79	G83.2	
	G98	G98	G98	
	G99	G99	G99	
Special fixed cycles	G34 G35 G36	-	-	Refer to 12.1.3.2.
Fixed cycles for turning machining	-	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	Refer to 12.1.3.4. Refer to 12.1.3.5.

12.1.3.1 Fixed Cycle for Drilling

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(1) T system, M system ; 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.

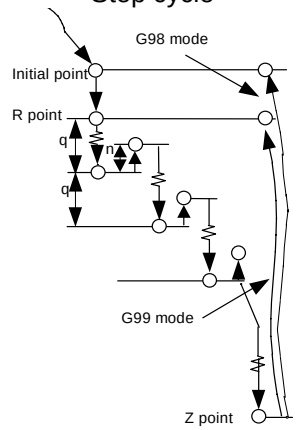
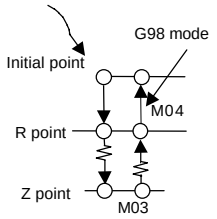
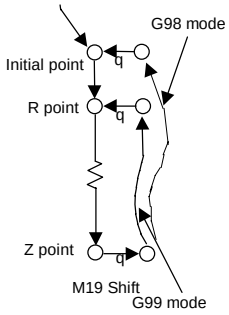
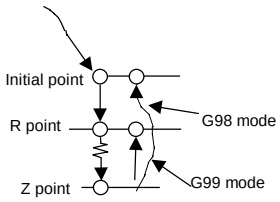
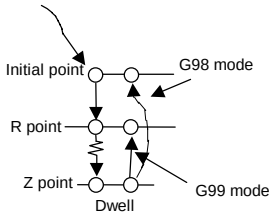
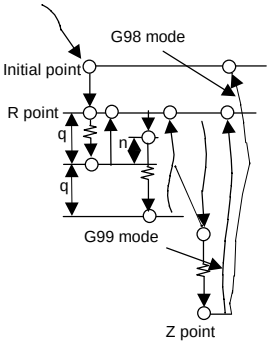
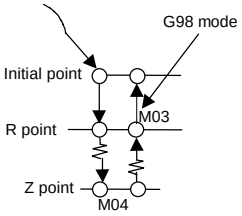
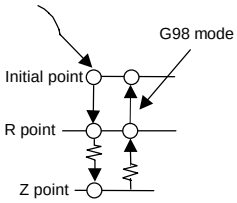
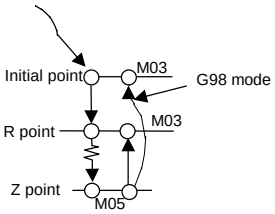
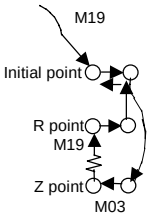
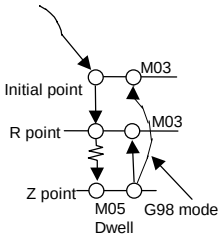
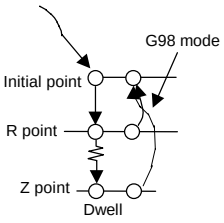
G code	Function
G98	Initial point level return
G99	R point level return

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, 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 system; 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 : Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89)
G87/G88/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.

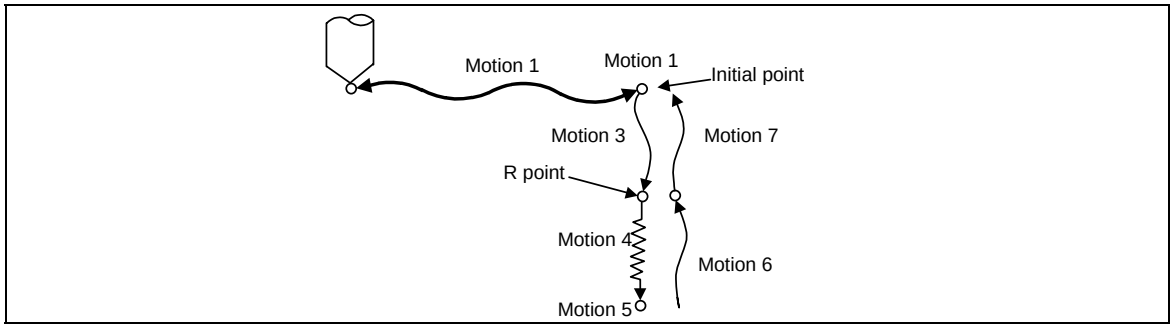
Xx1, Cc1 : Data for positioning X (Z) and C axes
The data is unmodal. To execute the same hole machining mode consecutively, specify the data for each block.

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
,Ss1	:	Designates spindle rotation speed at retract
Dd1	:	Designates tap spindle No. for G84 (G88)
,Rr1	:	Changes between synchronous/asynchronous in G84 (G88)

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:

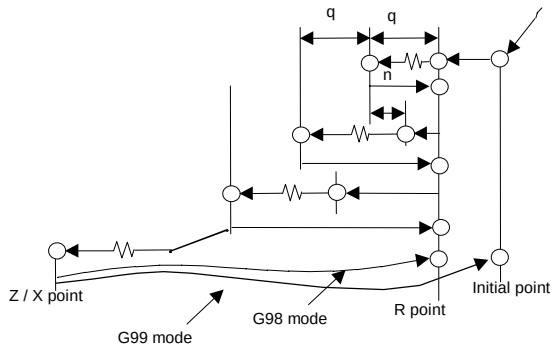
G98: Initial level return
G99: R point level return

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.

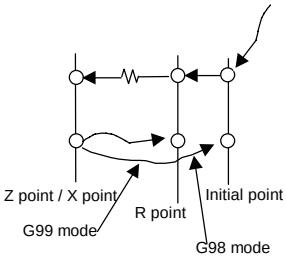
Deep-hole drilling cycle (G83, G87)

G83/G87
Deep-hole drilling cycle (G83: Z-axis direction, G87: X-axis direction)

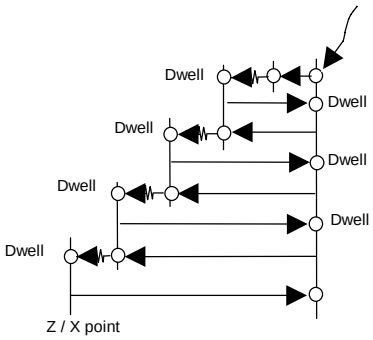
When Q command is given



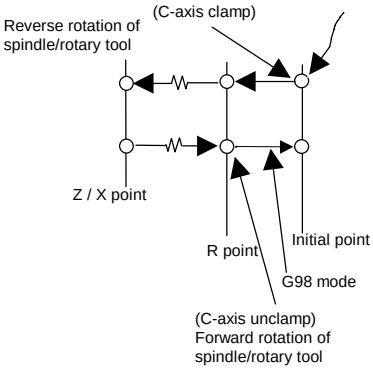
When Q command is not given



G83.2
Deep-hole drilling cycle

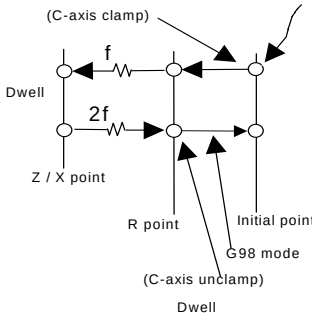


G84/88
Tapping cycle



Output or no output can be set using a parameter for the C-axis clamp/unclamp M code

G85/89
Boring cycle



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

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; G34 to G37

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

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.

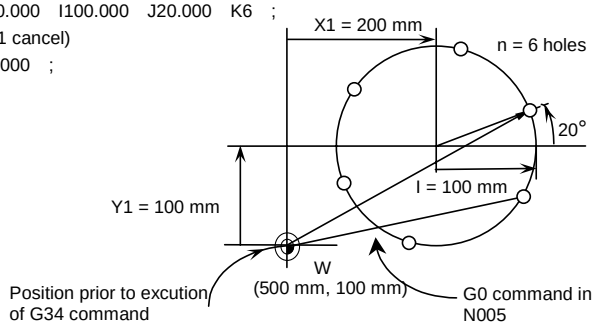
G34 Xx Yy Ir Jθ Kn ;

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.

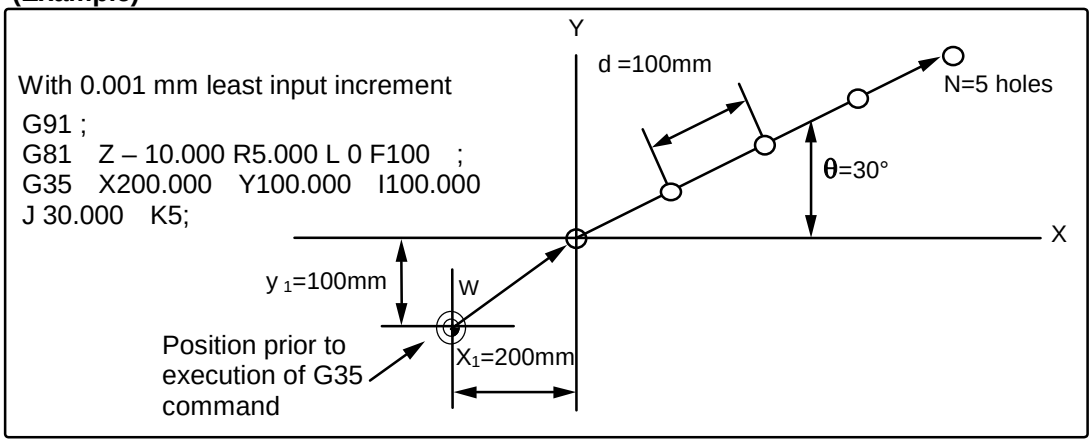
(2) Line at angle (G35)

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)



(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.

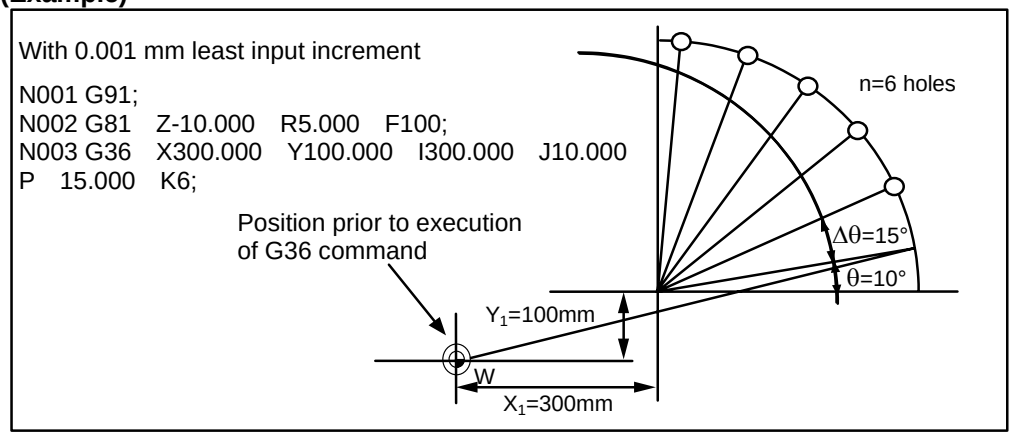
G36 Xx Yy Ir Jθ PΔθ Kn ;

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)

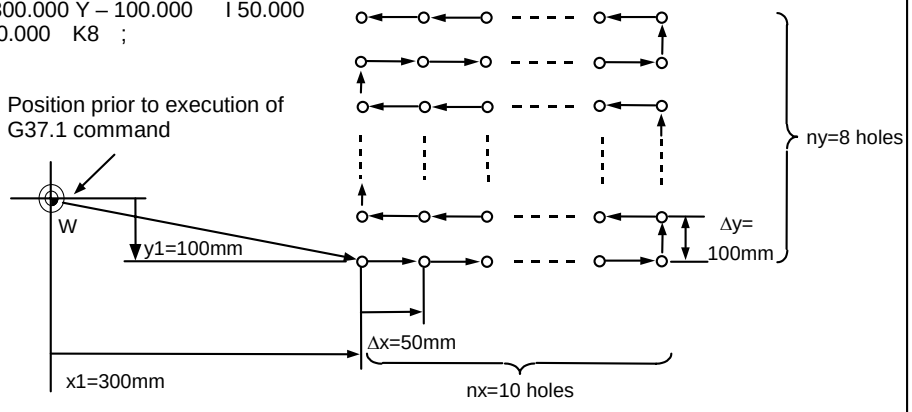
With the starting point at on the position designated by X and Y, this function enables the tool to drill the holes on the lattice with "nx" number of holes at parallel intervals of Δx to the X axis. Drilling proceeds in the X-axis direction. The drilling operation at each of the hole positions is based 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; G77 to G79

C6		C64		
T system	L system	M system	L system	T 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:

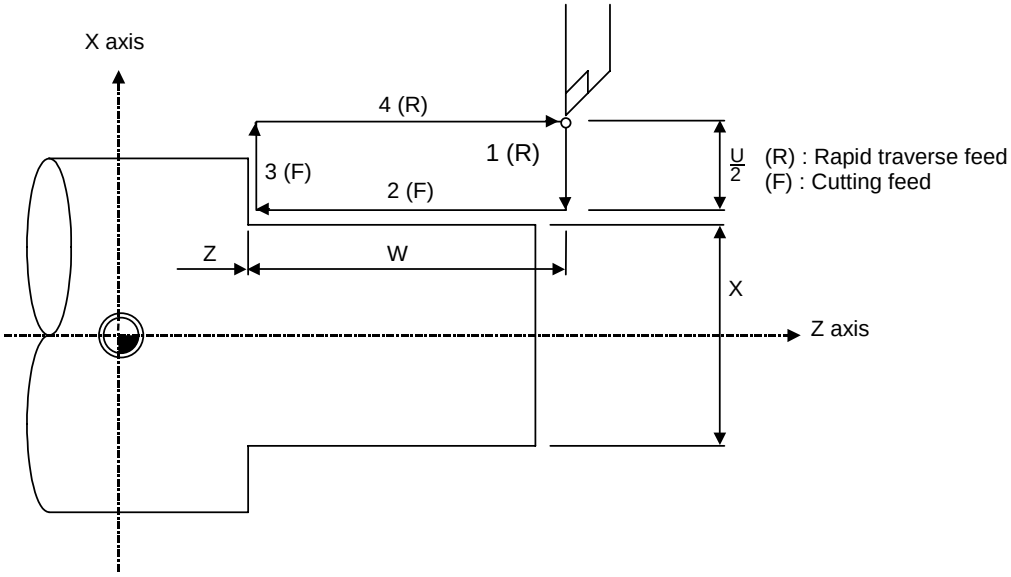
G00, G01, G02, G03
G09
G10, G11
G27, G28, G29, G30
G31
G33, G34
G37
G92
G52, G53
G65

(1) Longitudinal cutting cycle (G77)

(a) Longitudinal cutting

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

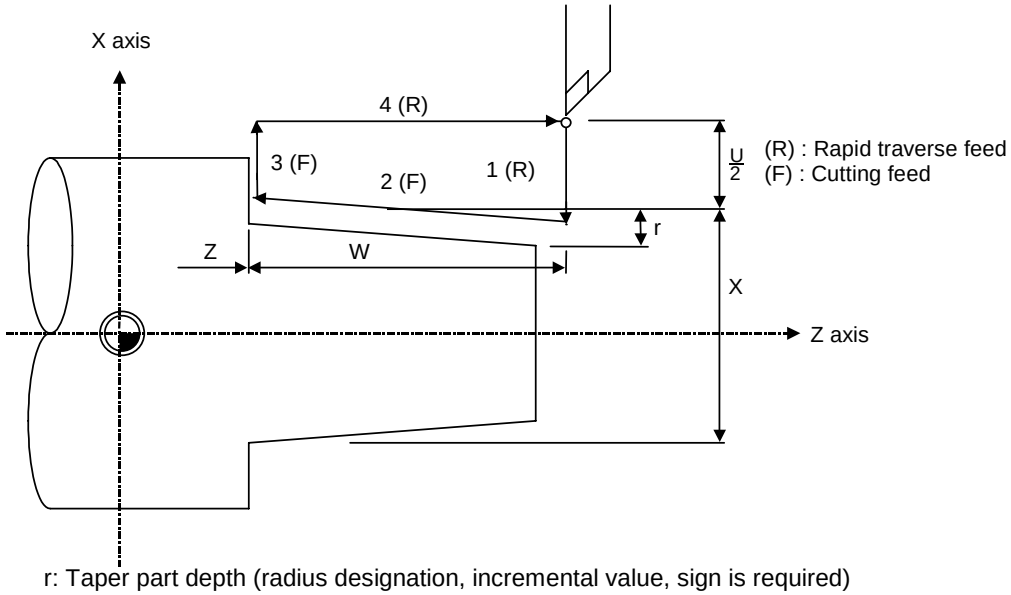
G77 X/U_ Z/W_ F_ ;



(b) Taper cutting

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

G77 X/U_ Z/W_ R_ F_ ;



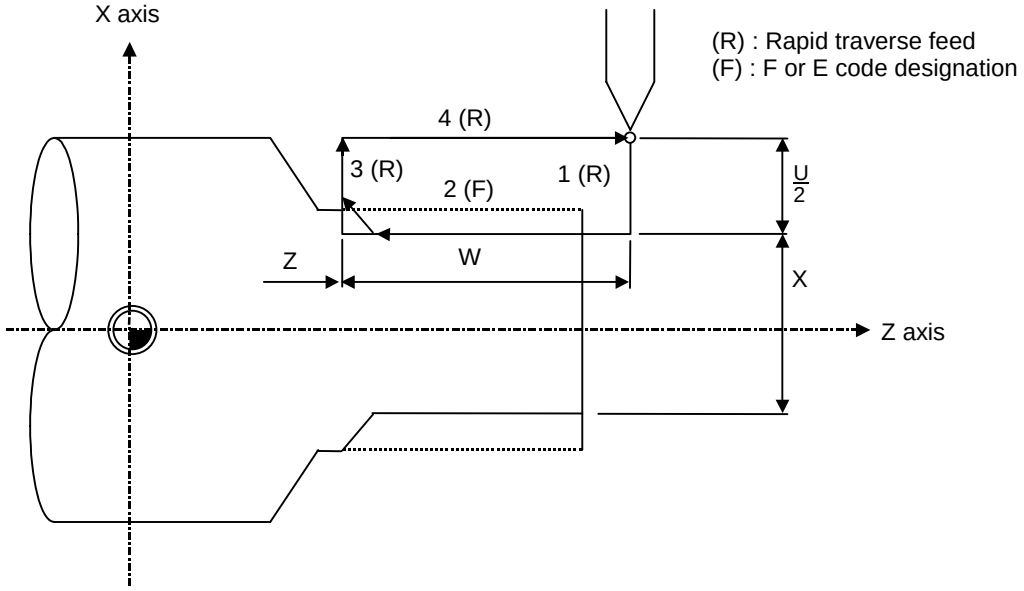
r: Taper part depth (radius designation, incremental value, sign is required)

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

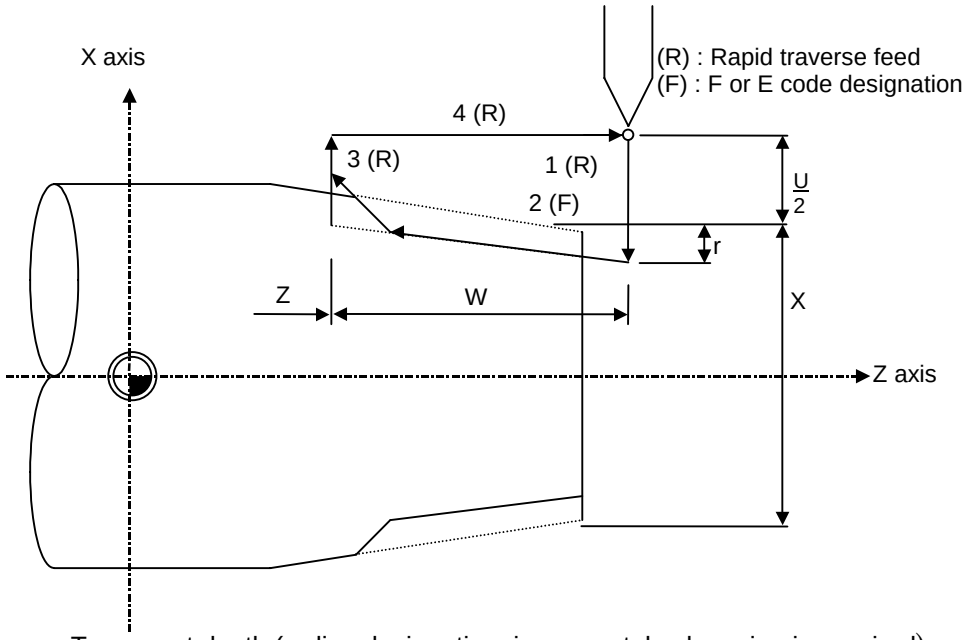
G78 X/U_ Z/W_ F/E_ ;



(b) Taper thread cutting

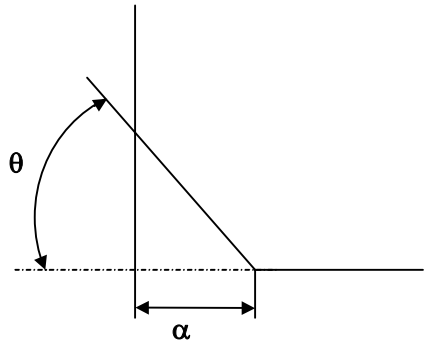
Taper thread cutting can be performed by the following block:

G78 X/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.

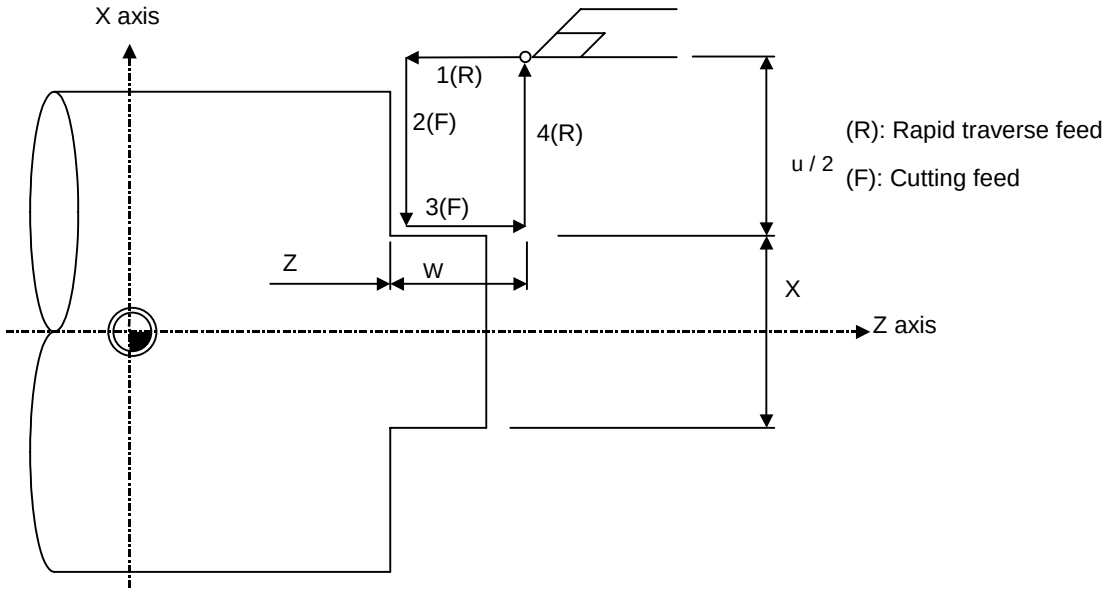
θ : Thread cutting-up angle
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°.

(3) Face cutting cycle (G79)

(a) Straight cutting

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

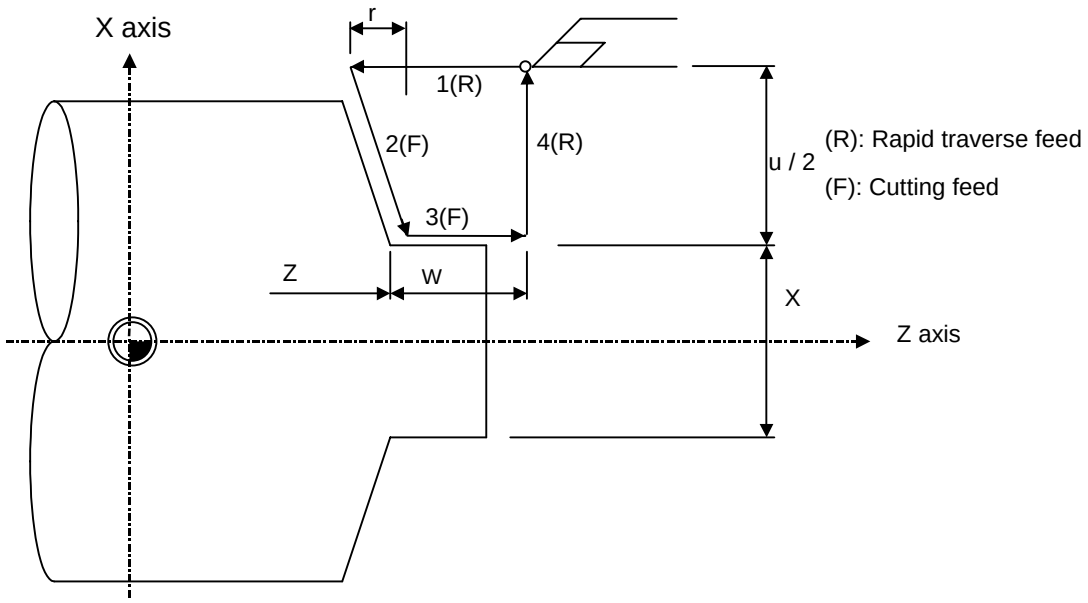
G79 X/U_ Z/W_ F_ ;



(b) Taper cutting

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

G79 X/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; G70 to G76

C6		C64		
T system	L system	M system	L system	T system
–	○	–	○	–

(a) Longitudinal rough cutting cycle I (G71)

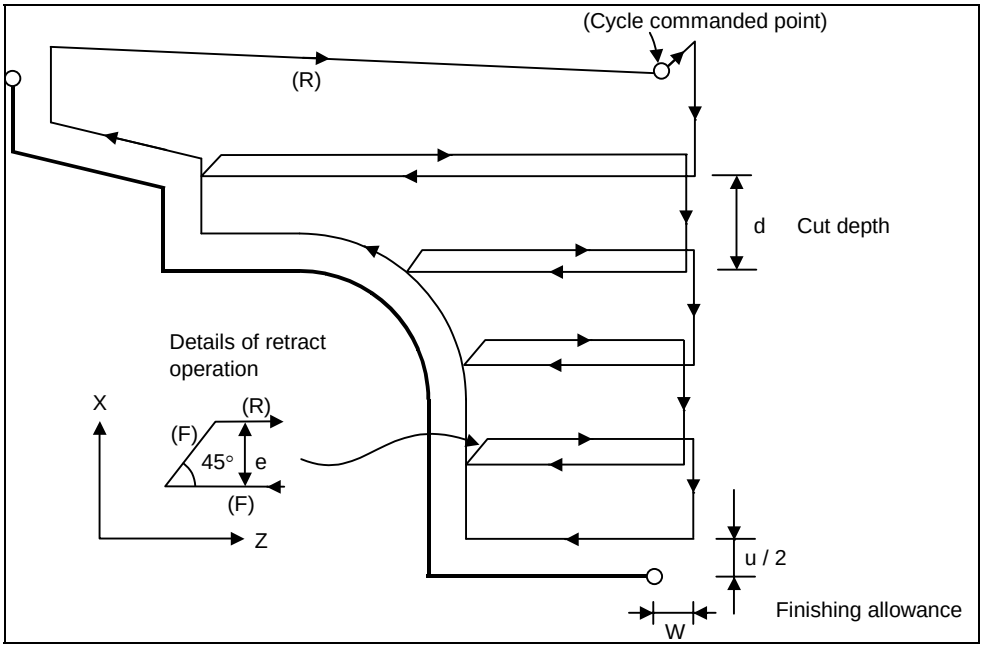
The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.
The machining program is commanded as follows.

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)

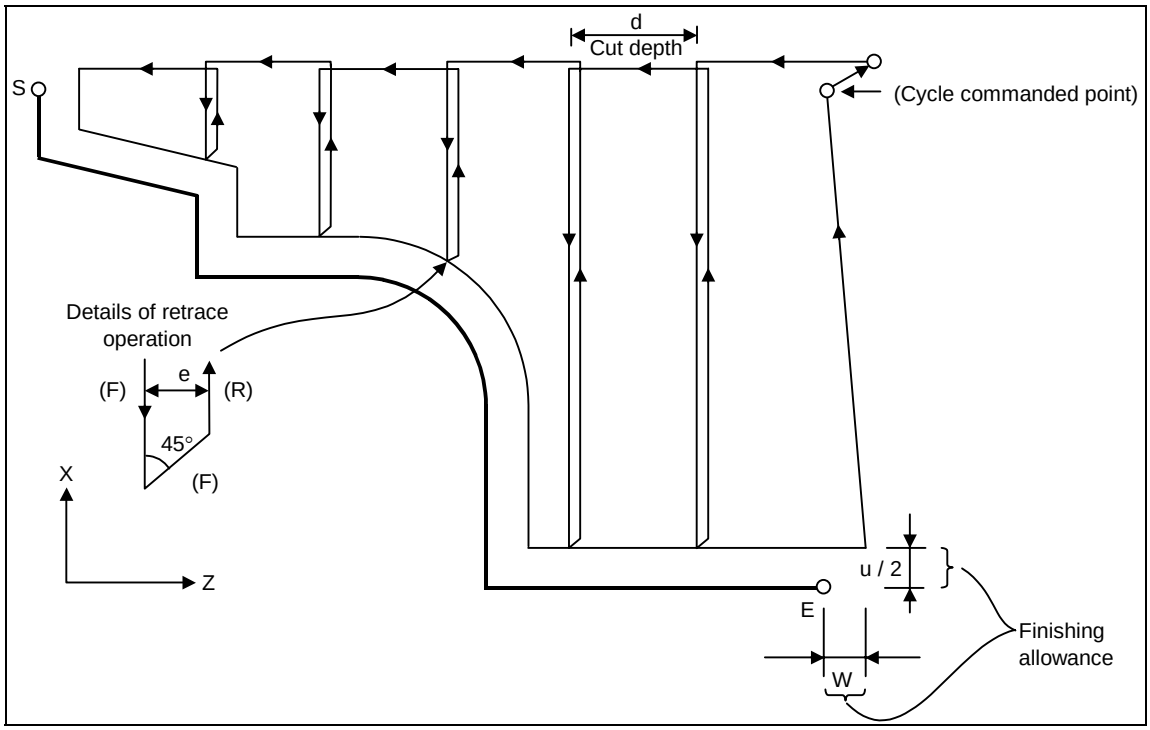
The finish shape program is called, and rough turning is performed in the end face direction while intermediate path is being calculated automatically.
The machining program is commanded as follows.

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.



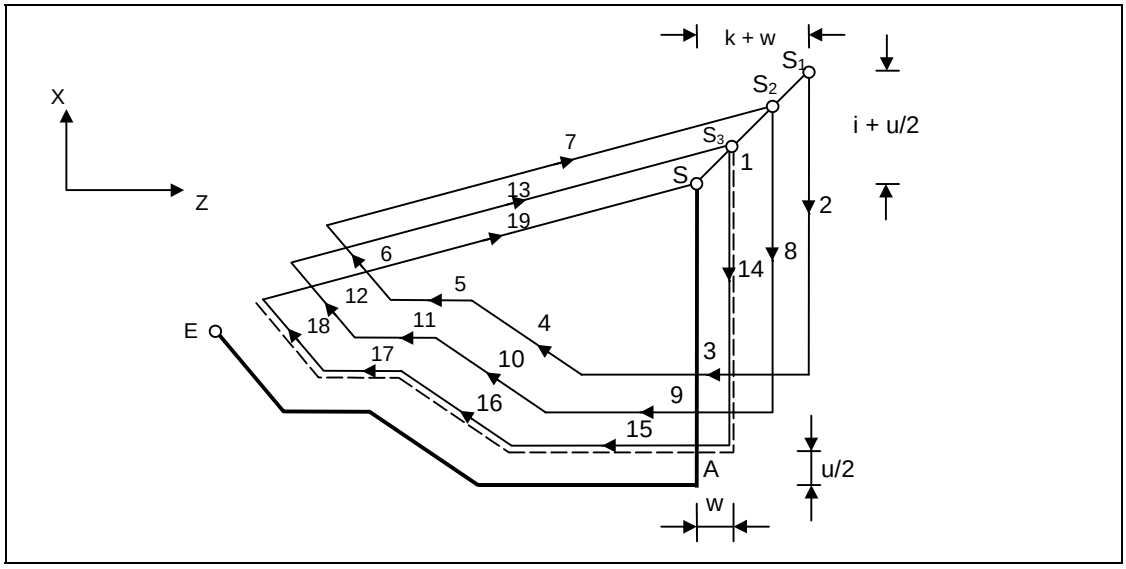
(c) Molding material in rough cutting cycle (G73)

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)

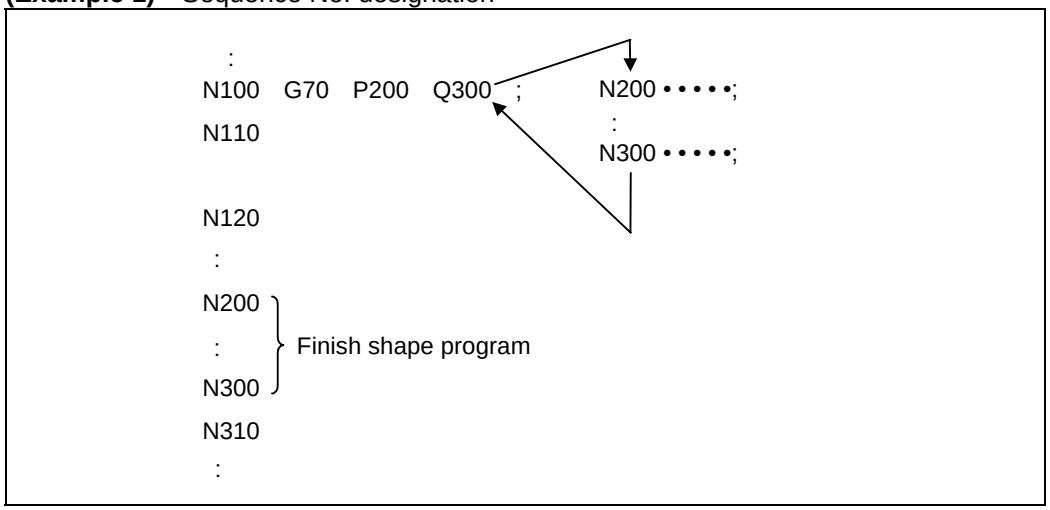
After rough cutting is performed by using G71 to G73, finish turning can be performed by using the G70 command.
The machining program is commanded as follows.

G70 A_ P_ Q_ ;

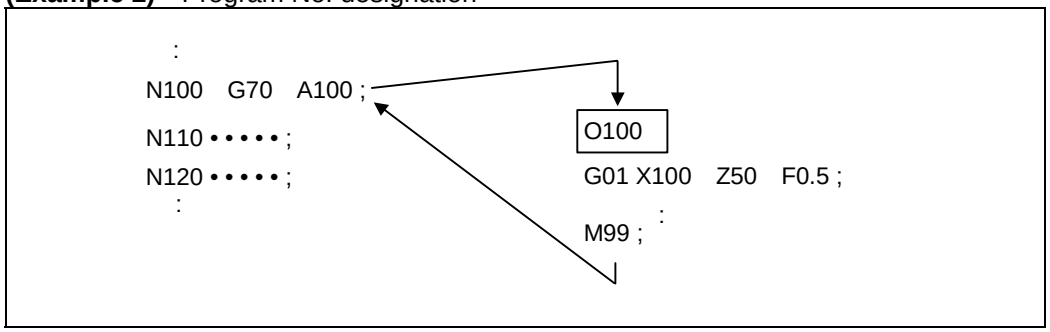
A : Finish shape program number. (If it is omitted, the program being executed is assumed to be designated.)
P : Finish shape start sequence number. (If it is omitted, the program top is assumed to be designated.)
Q : Finish shape end sequence number. (If it is omitted, the program end is assumed to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



(Example 2) Program No. designation



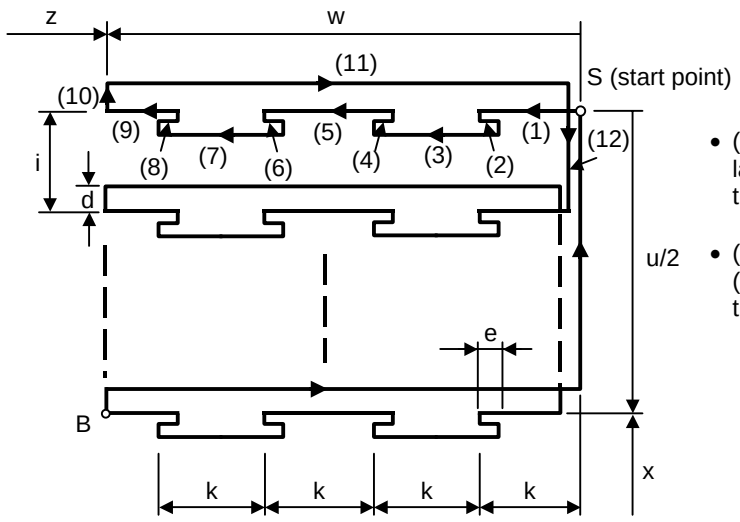
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

(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.

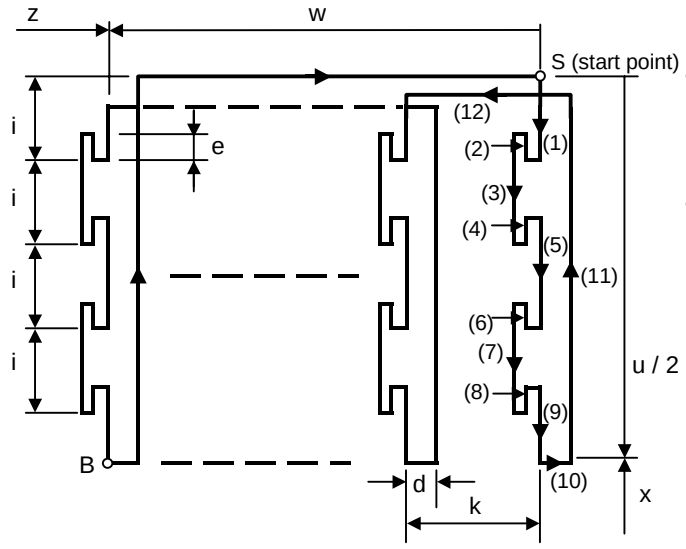
(f) Longitudinal cutting-off cycle (G75)

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



- (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 feedrate.

(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

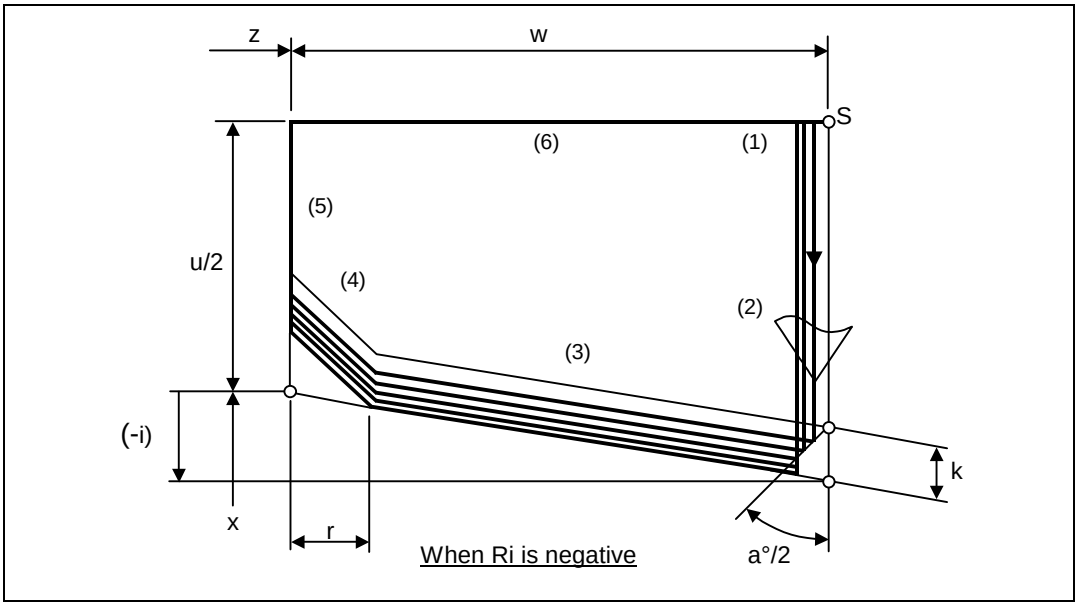
G76 Pmra Rd ;

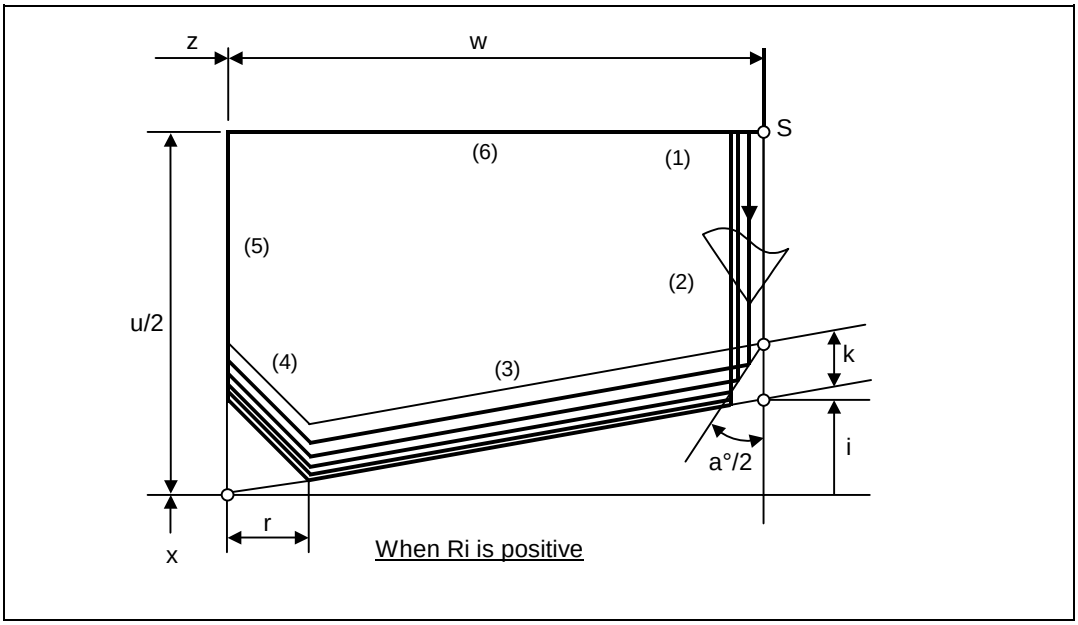
G76 X/U Z/W Ri Pk QΔd Fl ;

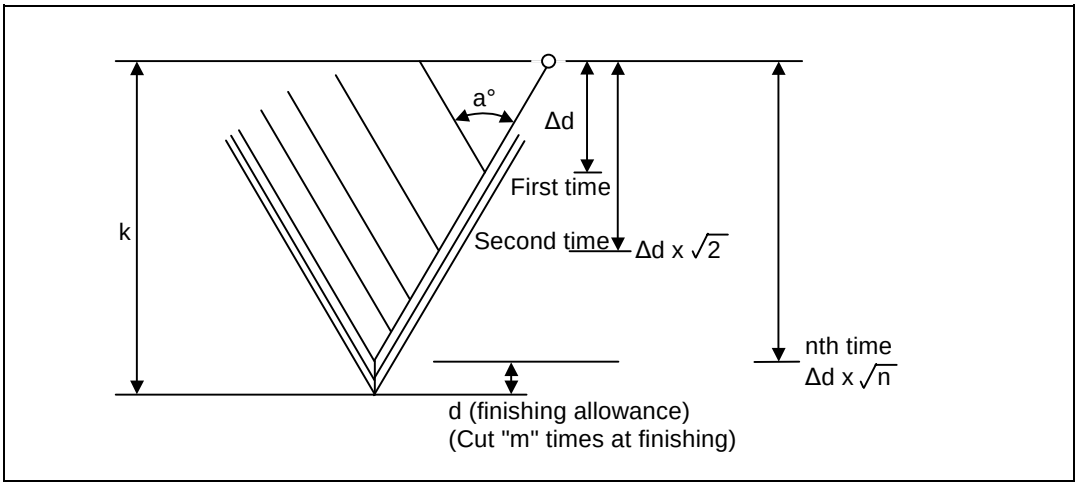
m	:	Cut count at finishing 01 to 99 (modal)
r	:	Chamfering amount 00 to 99 (modal). Set in 0.1-lead increments.
a	:	Nose angle (included angle of thread) 00 to 99 (modal) Set in 1-degree increments.
d	:	Finishing allowance (modal)
X/U	:	X axis end point coordinate of thread part. Designate the X coordinate 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
(4)
(3)
k
i
a°/2
r
x
When Ri is positive


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

12.1.4 Mirror Image

12.1.4.3 G Code Mirror Image

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

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.

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.

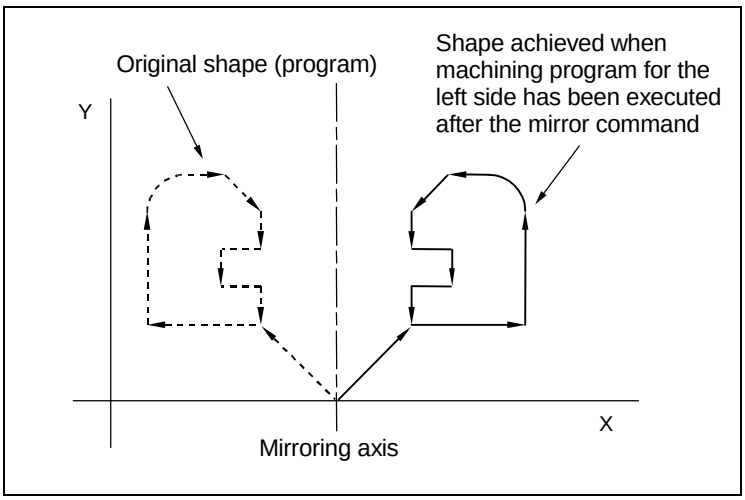
The program format for the G code mirror image cancel is shown below.

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



12.1.4.4 Mirror Image for Facing Tool Posts

C6		C64		
T system	L system	M system	L system	T system
–	Δ	–	Δ	–

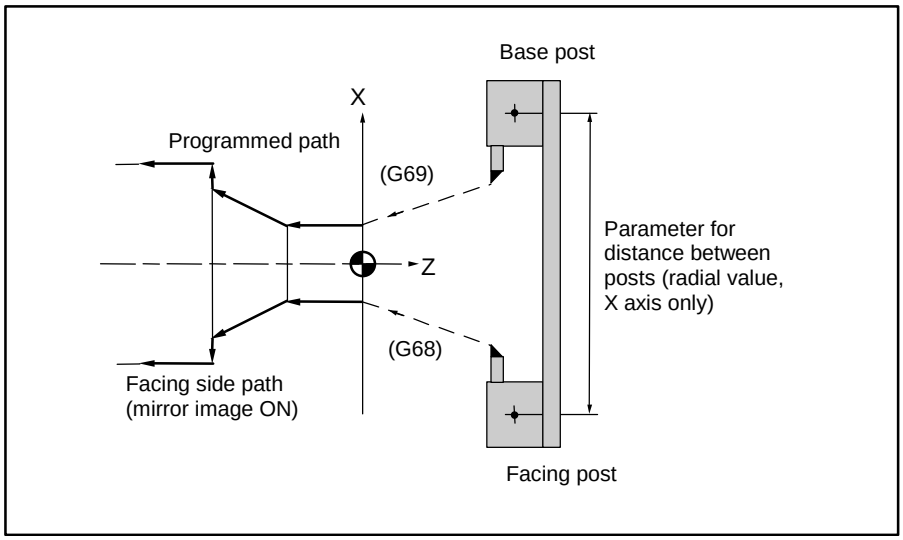
With machines in which the base tool post and facing tool post are integrated, this function enables the programs prepared for cutting at the base side to be executed by the tools on the facing side. The distance between the two posts is set beforehand with the parameter.

The command format is given below.

G68;	Facing tool post mirror image ON
G69;	Facing tool post mirror image OFF

When the G68 command is issued, the subsequent program coordinate systems are shifted to the facing side and the movement direction of the X axis is made the opposite of that commanded by the program.
When the G69 command is issued, the subsequent program coordinate systems are returned to the base side.

The facing tool post mirror image function can be set to ON or OFF automatically by means of T (tool) commands without assigning the G68 command.
A parameter is used to set ON or OFF for the facing tool post mirror image function corresponding to the T commands.



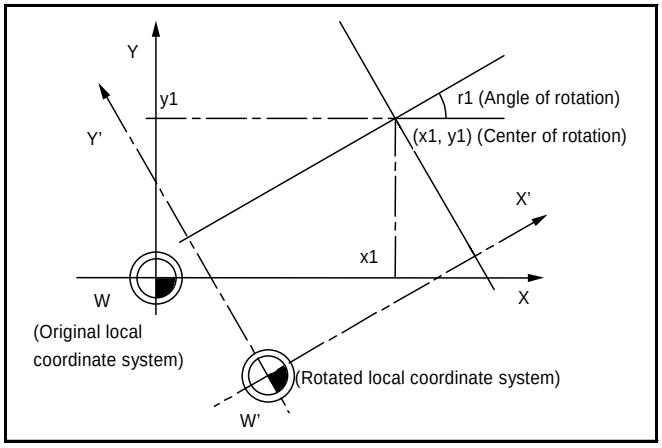
12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

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	



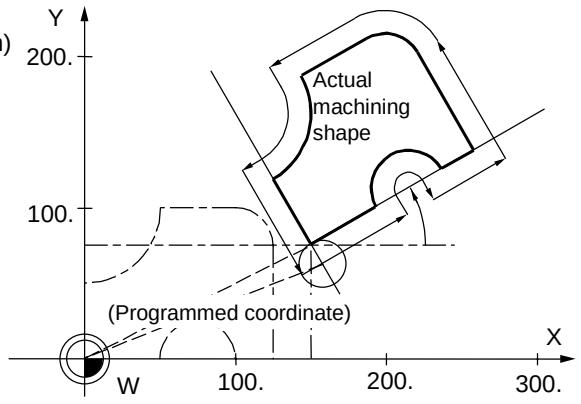
(1) Angle of rotation "r1" can be set in least input increment from –360° to 360°.
(2) The coordinates are rotated counterclockwise by an amount equivalent to the angle which is designated by angle of rotation "r1".
(3) The counter is indicated as the point on the coordinate system prior to rotation.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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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.

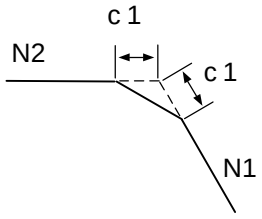
(1) Corner chamfering / Corner R I

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

- **Corner chamfering**

Example:

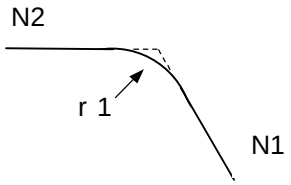
```
N1 G01 Xx1 Zz1,  Cc1  ;
N2 Zz2  ;
```



- **Corner R**

Example:

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



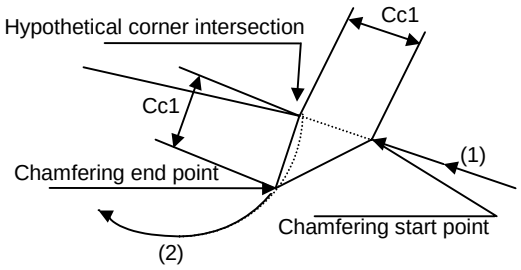
(Note 1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

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

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

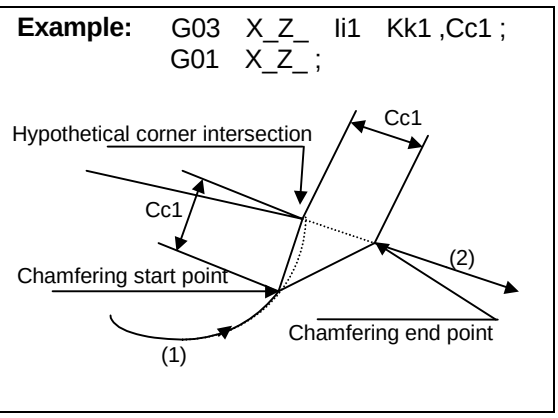
(a) Corner chamfering II (Linear – circular)

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



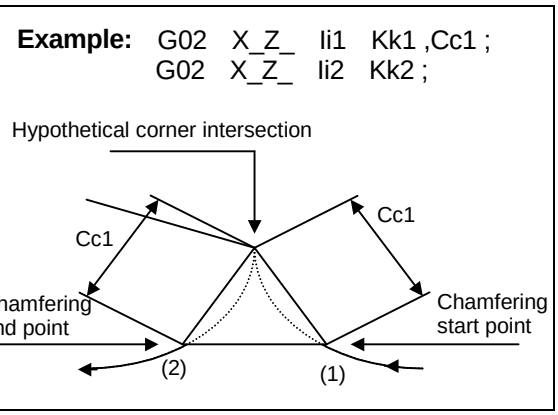
(b) Corner chamfering II (Circular - linear)

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

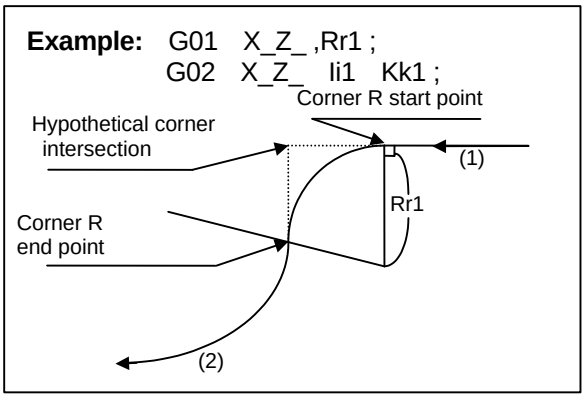


(c) Corner chamfering II (Circular - circular)

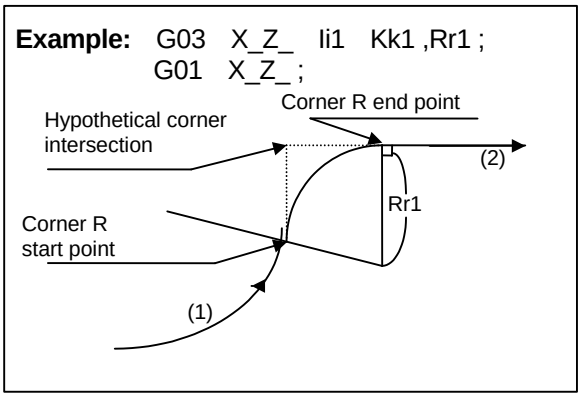
Example: G02 X_Z_ Ii1 Kk1 ,Cc1 ;
G02 X_Z_ Ii2 Kk2 ;



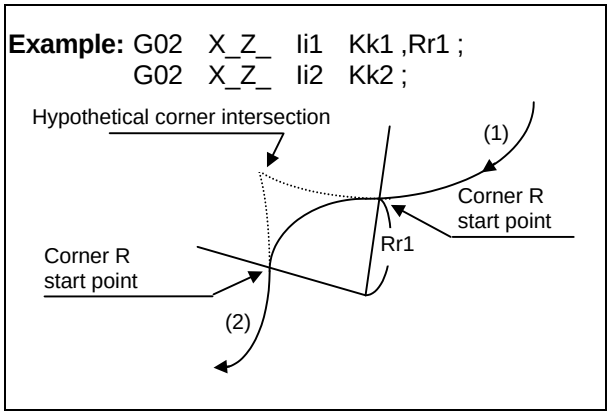
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular – linear)



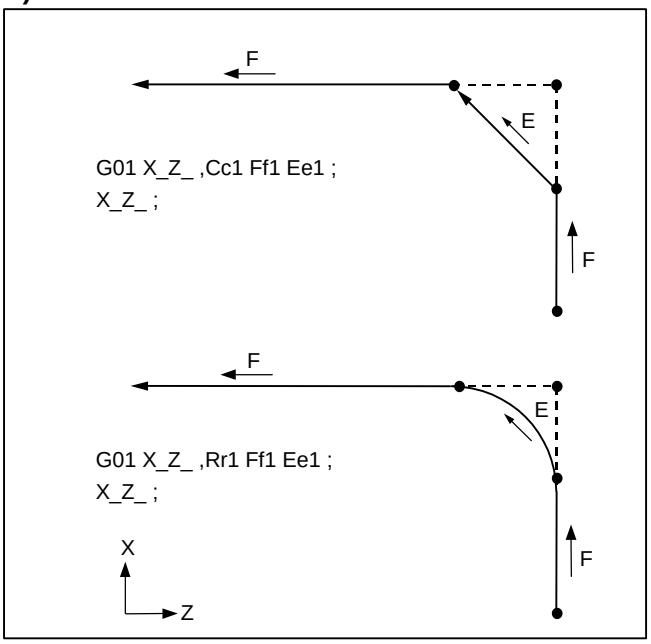
(f) Corner R II (Circular – circular)



(3) Specification of corner chamfering / corner R speed E

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

(Example)



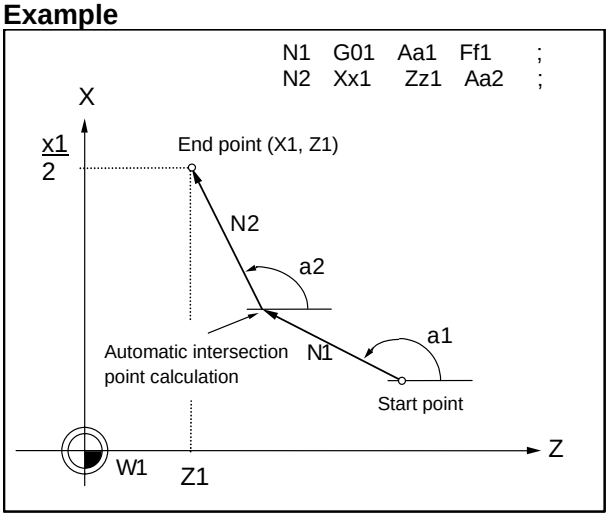
An E command is a modal and remains effective for feeding in next corner chamfering or corner R.
An E command has two separate modals: synchronous and asynchronous feed rate modals. The effective feed rate is determined by synchronous (G95) or asynchronous (G94) mode.
If an E command is specified in 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 corner R.
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).

12.1.6.3 Geometric Command

C6		C64		
T system	L system	M system	L system	T 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.

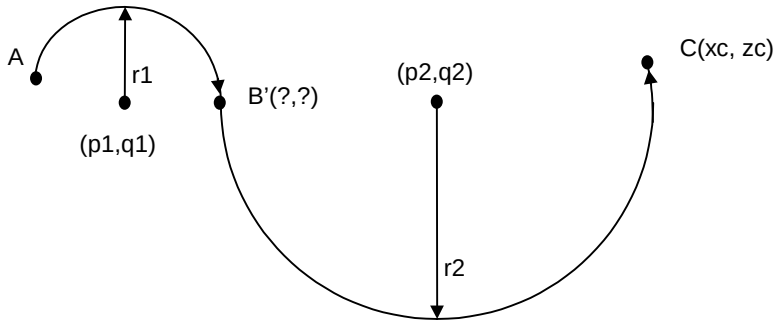
(Note 1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(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    ;
```



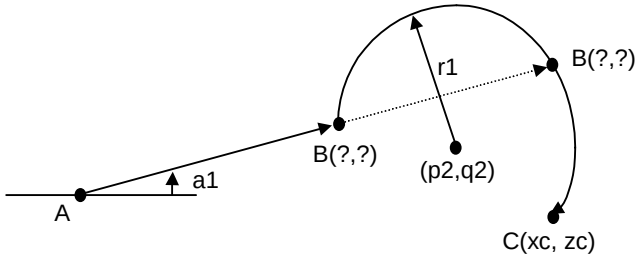
I and K are circular center coordinate incremental values; distances from the start point in the first block or distances from the end point in the second block. P and Q commands (X, Z absolute center coordinates of circular arc) can be given instead of I and K commands.

(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   ;
```



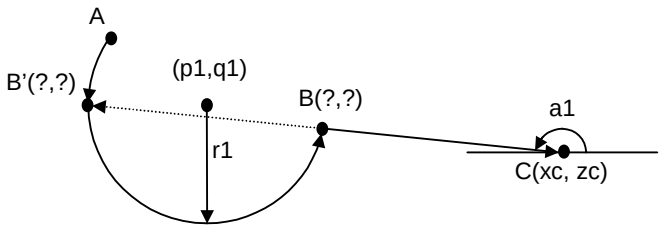
I and K : Incrimental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc
H = 0 : Intersection with shoter line
H = 1 : Intersection with longer line

(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  ;
```



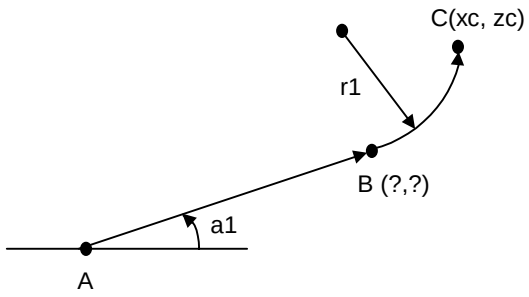
I and K : Incrimental coordinates from circular end point
P and Q : Absolute center coordinates of circular arc (L3 only)
H = 0 : Intersection with shoter line
H = 1 : Intersection with longer line

(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  ;
```

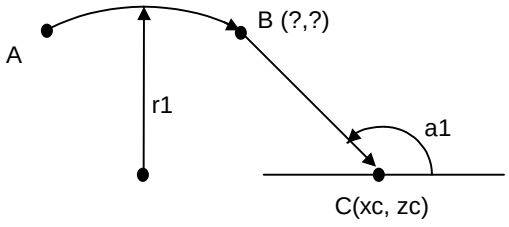


(5) Automatic calculation of arc-linear contact

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 ;
```



12.1.7 Axis Control

12.1.7.5 Circular Cutting

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

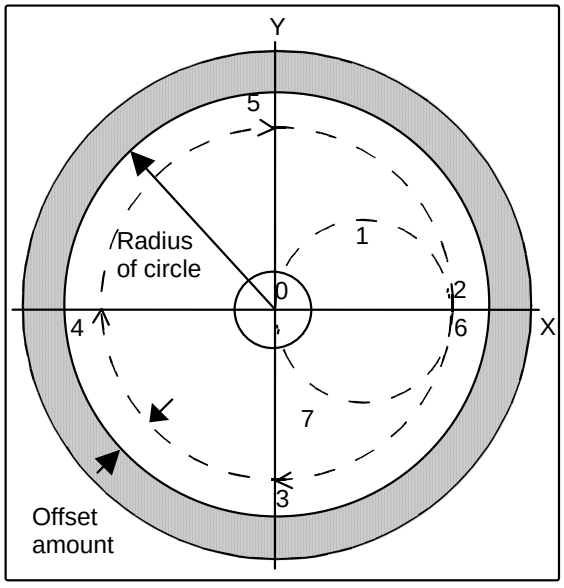
In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed 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)
$0 \rightarrow 1 \rightarrow 2 \rightarrow 3 \rightarrow 4 \rightarrow 5 \rightarrow 6 \rightarrow 7 \rightarrow 0$
When the G13 command is used
(path of tool center)
$0 \rightarrow 7 \rightarrow 6 \rightarrow 5 \rightarrow 4 \rightarrow 3 \rightarrow 2 \rightarrow 1 \rightarrow 0$

(Notes)

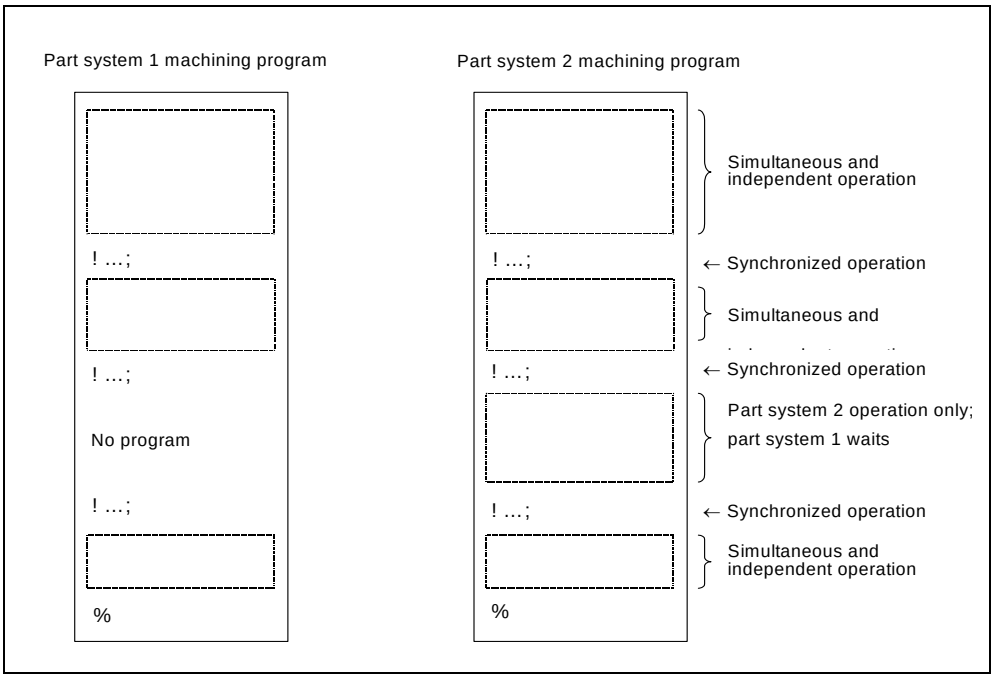
- Circular cutting is undertaken on the plane which has been currently selected (G17, G18 or G19).
- The (+) and (–) signs for the compensation amount denote reduction and expansion respectively.

12.1.8 Multi-part System Control

12.1.8.1 Synchronization between Part Systems

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The multi-axis, multi-part system compound control CNC system can simultaneously run multiple machining programs independently. This function is used in cases when, at some particular point during operation, the operations of different part systems are to be synchronized or in cases when the operation of only one part system is required.



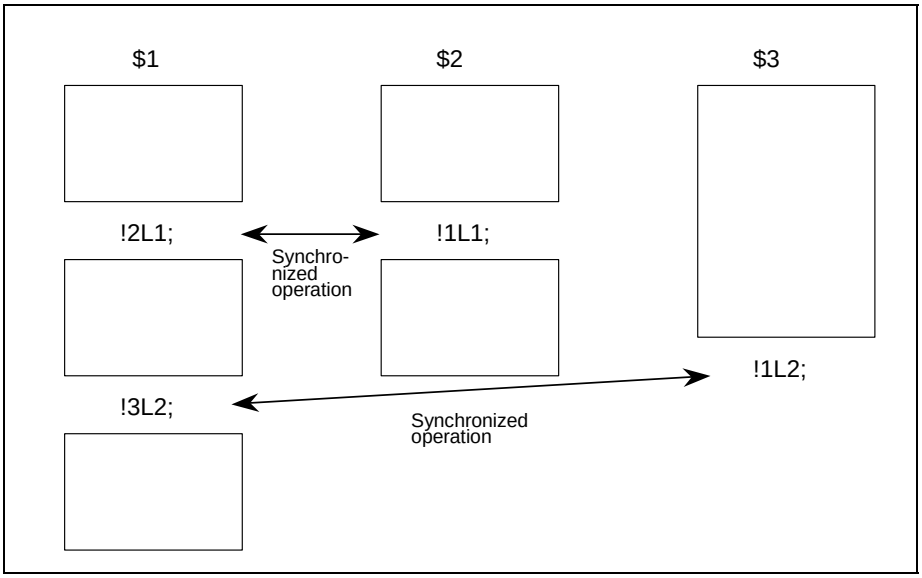
Command format

(1) Command for synchronizing with part system n

!nL1 ;

n : Part system number

1 : Synchronizing number 01 to 9999

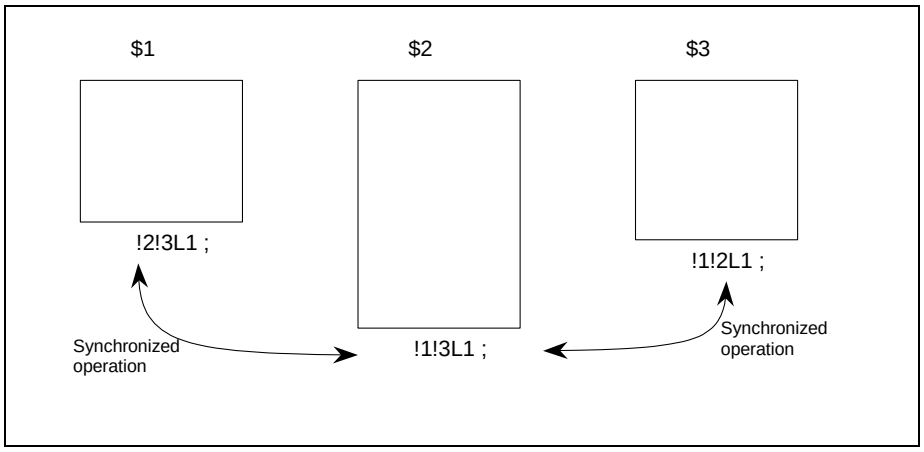


(2) Command for synchronizing among three part systems

!n!m…L1 ;

n, m: Part system number n ≠ m

1 : Synchronizing number 01 to 9999



12.1.8.2 Start Point Designation Synchronization

C6		C64		
T system	L system	M system	L system	T system
–	–	Δ	Δ	Δ

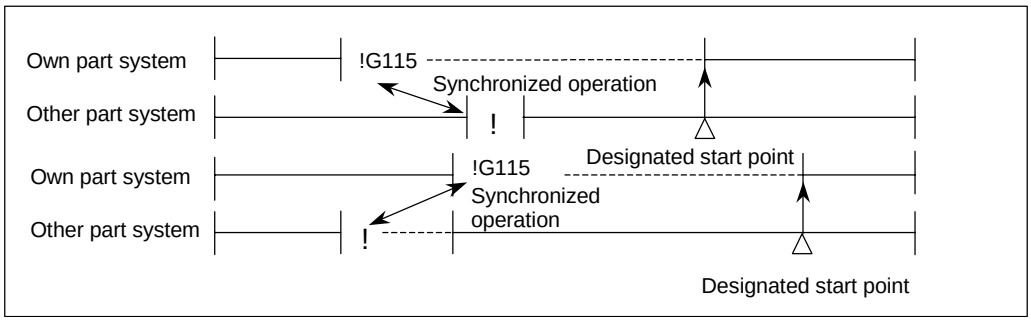
The synchronizing point can be placed in the middle of the block by designating the start point.

(1) Start point designation synchronization Type 1 (G115)

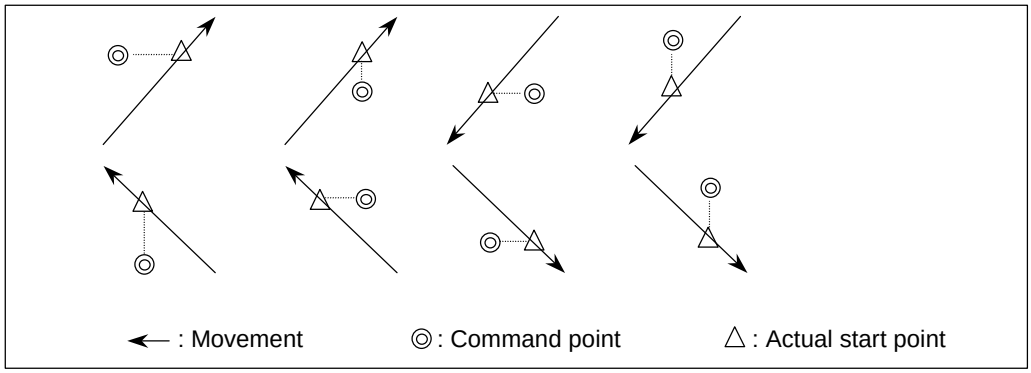
Command format

!LI G115 X_ Z_ ;	
!LI	: Synchronizing command
G115	: G command
X_, Z_	: Own start point (designate other part system's coordinate value)

(a) The other part system starts first when synchronizing is executed.
(b) The own part system waits for the other part system to move and reach the designated start point, and then starts.



(c) When the start point designated by G115 is not on the next block movement path of the other part system, the own part system starts once the other part system has reached all of the start point axis coordinates.

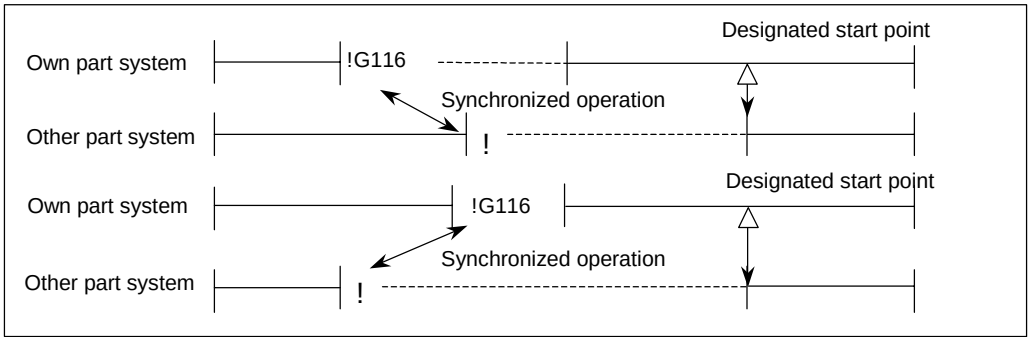


(2) Start point designation synchronization Type 2 (G116)

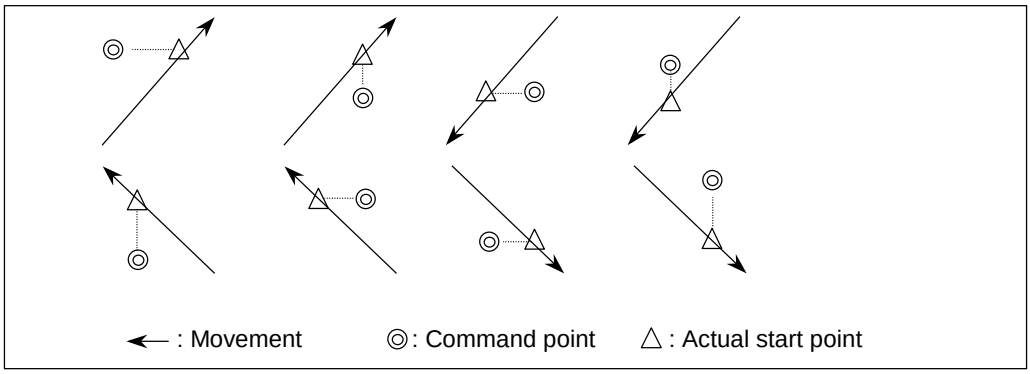
Command format

!Ll G116 X_ Z_ ;	
!Ll	: Synchronizing command
G116	: G command
X_, Z_	: Other start point (designate own part system's coordinate value)

(a) The own part system starts first when synchronizing is executed.
(b) The other part system waits for the own part system to move and reach the designated start point, and then starts.



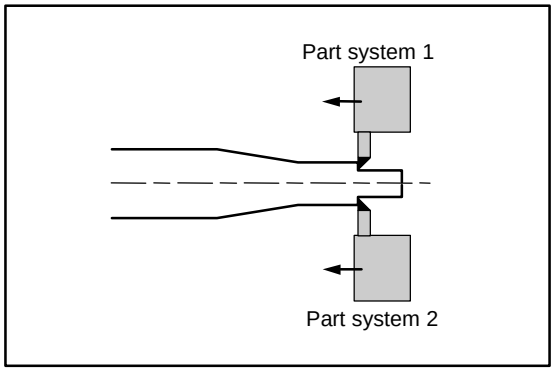
(c) When the start point designated by G116 is not on the next block movement path of the own part system, the other part system starts once the own part system has reached all of the start point axis coordinates.



12.1.8.6 Balance Cut; G14/G15

C6		C64		
T system	L system	M system	L system	T system
–	–	–	○	–

When workpiece that is relatively long and thin is machined on a lathe, deflection may result, making it impossible for the workpiece to be machined with any accuracy. In cases like this, the deflection can be minimized by holding tools simultaneously from both sides of the workpiece and using them in synchronization to machine the workpiece (balance cutting). This method has an additional advantage: since the workpiece is machined by two tools, the machining time is reduced.
The balance cutting function enables the movements of the tool rests belonging to part system 1 and part system 2 to be synchronized (at the block start timing) so that this kind of machining can easily be accomplished.



The command format is given below.

G14	Balance cut command OFF (modal)
G15	Balance cut command ON (modal)

G14 and G15 are modal commands. When the G15 command is assigned, the programmed operations of two part systems are synchronized (at the block start timing) for all blocks until the G14 command is assigned or until the modal information is cleared by the reset signal.

Part system 1 program

```
T0101;
G00 X_ Z_;
G15;
G01 Z_ F0.4;
      ⋮
```

Part system 2 program

```
T0102;
G00 X_ Z_;
G15;
G01 Z_ F0.4;
      ⋮
```

Whereas synchronization is possible only with the next block when using the code "!" of synchronization between part systems, the balance cutting function provides synchronization (at the block start timing) with multiple consecutive blocks.

12.1.8.8 2-part System Synchronous Thread Cutting; G76.1/G76.2

C6		C64		
T system	L system	M system	L system	T system
-	○	-	○	-

The 2-part system synchronous thread cutting cycle is the function which performs synchronous thread cutting for the same spindle by part systems 1 and 2.
The 2-part system synchronous thread cutting cycle is "2-part system synchronous thread cutting cycle I" (G76.1) for synchronous thread cutting of two screws or "2-part system synchronous thread cutting cycle II" (G76.2) for thread cutting of one screw.

(1) 2-part system synchronous thread cutting cycle (I)

Command format

G76. 1 X/U_ Z/W_ Ri Pk QΔd Fl ;

X/U : X axis end point coordinate of screw .... Designate the X coordinate of the end point at screw in an absolute or incremental value.

Z/W : Z axis end point coordinate of screw .... Designate the Z coordinate of the end point at screw in an absolute or incremental value.

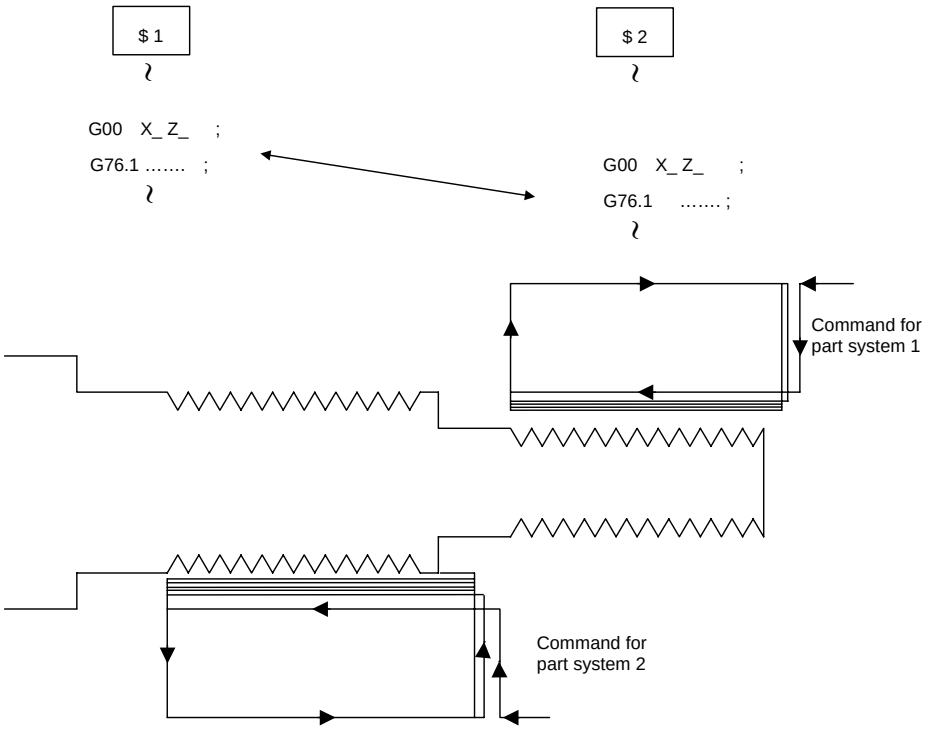
i : Height constituent of taper at screw (radius value) ... When i is 0, a straight screw is generated.

k : Screw 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

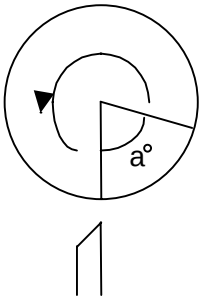
If G76.1 command is given in part system 1 or 2, a wait is made until G76.1 command is given in the other part system.
Once the G76.1 command exists in both part systems, the thread cutting cycle is started.



(2) 2-part system synchronous thread cutting cycle (II)

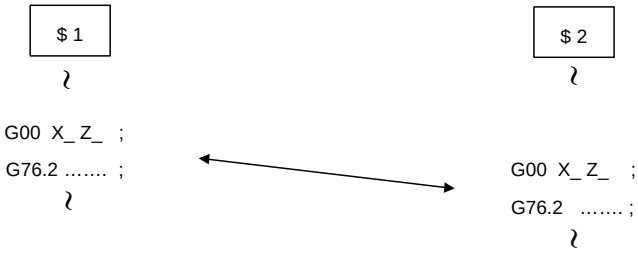
Command format

G76. 2 X/U_ Z/W_ Ri Pk QΔd Aa Fl ;
a : Thread cutting start shift angle

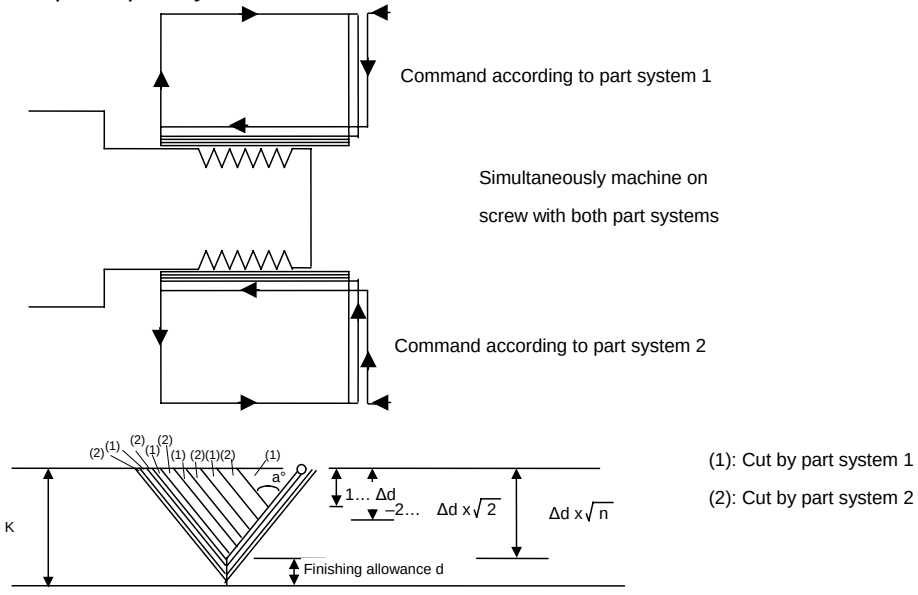


Thread cutting command waits for 1-revolution synchronizing signal of the spindle encoder and starts moving. The start point can be delayed by thread cutting start angle.

The address except A has the same meanings as those in 2-part system synchronous thread cutting cycle I.
If G76.2 command is given in part system 1 or 2, a wait is made until G76.2 command is given in the other part system. Once the G76.2 command exists in both part systems, the thread cutting cycle is started.



In the G76.2 cycle, the same screw is assumed to be cut, and it is cut deeply according to alternate cut depth in part systems 1 and 2.



12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The parameters set from the setting and display unit 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	;	Parameter settings in data setting mode
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. When using the multi-part system, the 1st axis in each part system is set as "1", the 2nd axis is set as "2", and so forth.
(Note 4) Command G10L50 and G11 in independent blocks. A program error will occur if not commanded in independent blocks.

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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(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)

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. [T system, M system]

(2) Tool offset input

The tool offset amounts, which have been set from the setting and display unit, can be input by program commands.
The command format differs between the [T system, M system] and the [L system]. The respective command format must be set by a parameter.

[T system, 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: G63

C6		C64		
T system	L system	M system	L system	T 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)
- Cutting mode (G64)
- High-accuracy control mode command (G61.1) [T system, M system]

The machine is in the cutting mode status when its power is turned on.

12.1.10.2 Cutting Mode; G64

C6		C64		
T system	L system	M system	L system	T 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)
- Tapping mode (G63)
- High-accuracy control mode command (G61.1) [T system, M system]

The machine is in the cutting mode status when its power is turned on.

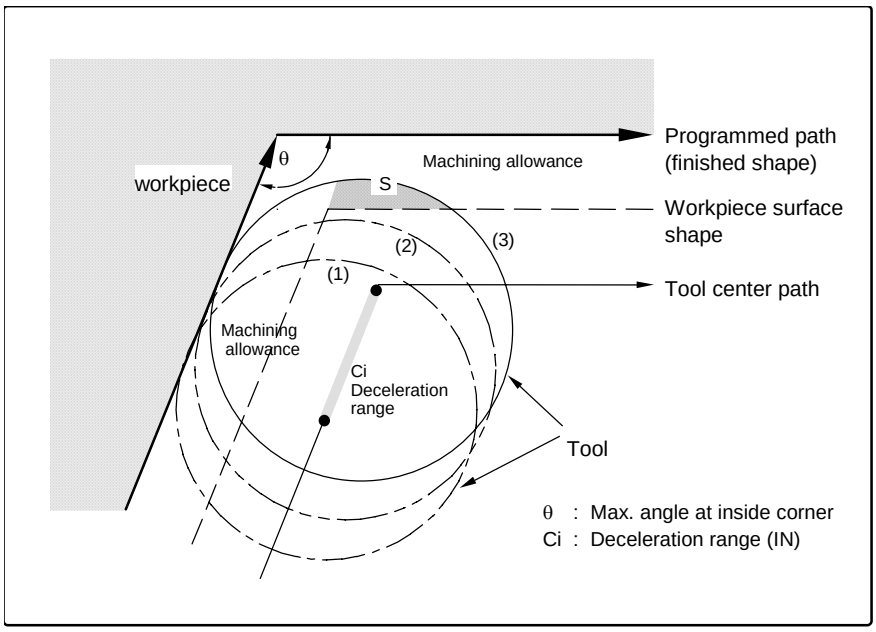
12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override; G62

C6		C64		
T system	L system	M system	L system	T 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 [T system, M system]



Operation

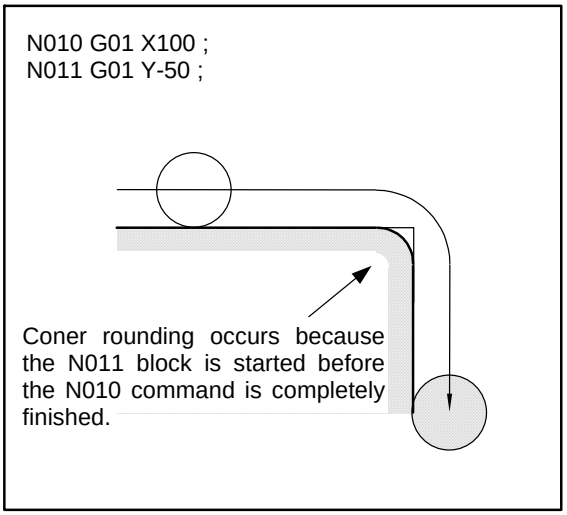
(a) When automatic corner override is not to be applied :
When the tool moves in the order of (1) → (2) → (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.

(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.

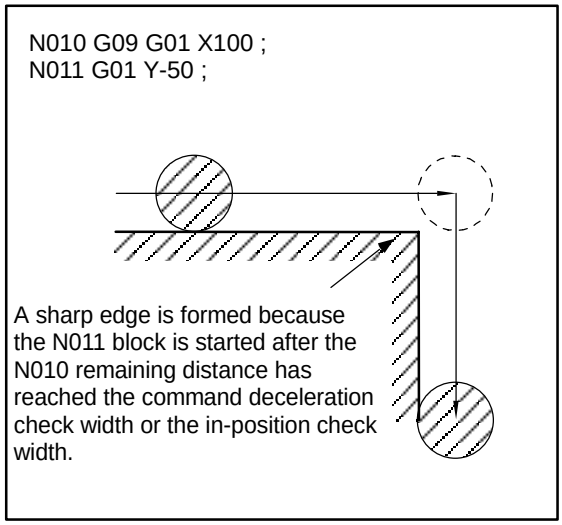
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another before executing the next block to alleviate the machine shock and to prevent the corner roundness that occurs 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.

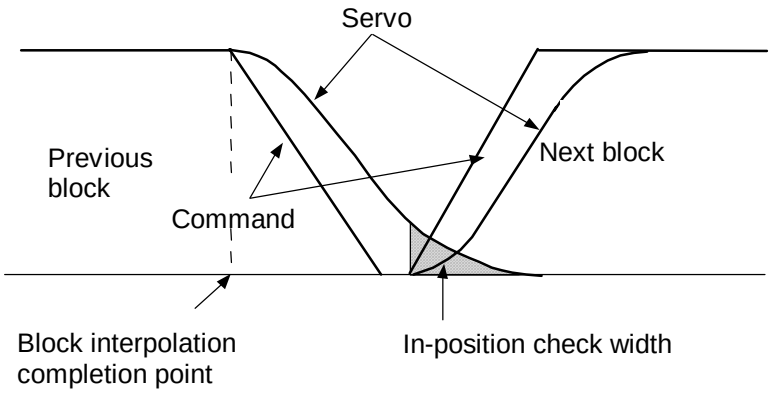
(2) Deceleration check in the cutting feed mode

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 Check Mode; G61

C6		C64		
T system	L system	M system	L system	T 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....High-accuracy control mode [T system, M system]

Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.2 Exact Stop Check; G09

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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; G61.1

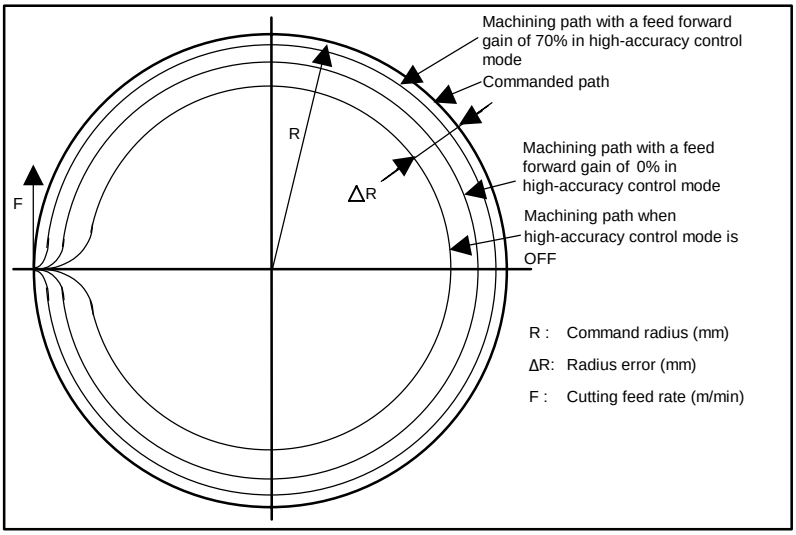
C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	–	Δ

This function controls the operation so the lag will be eliminated in control systems and servo systems. With this function, improved machining accuracy can be realized, especially during high-speed machining, and machining time can be reduced.

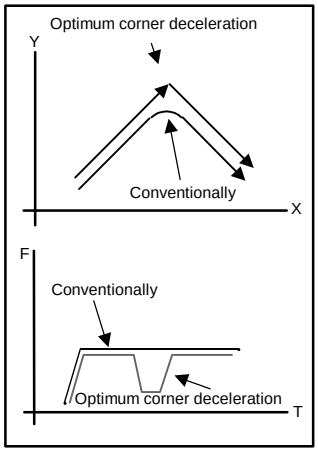
The high-accuracy control is commanded with ;

G61.1	**High-accuracy control ON**

Effects in G02/G03 circular interpolation



Neat machining of sharp corners without waste is realized with optimum linear acceleration/deceleration and corner judgement.



(1) Acceleration / deceleration before interpolation [T system, M system]

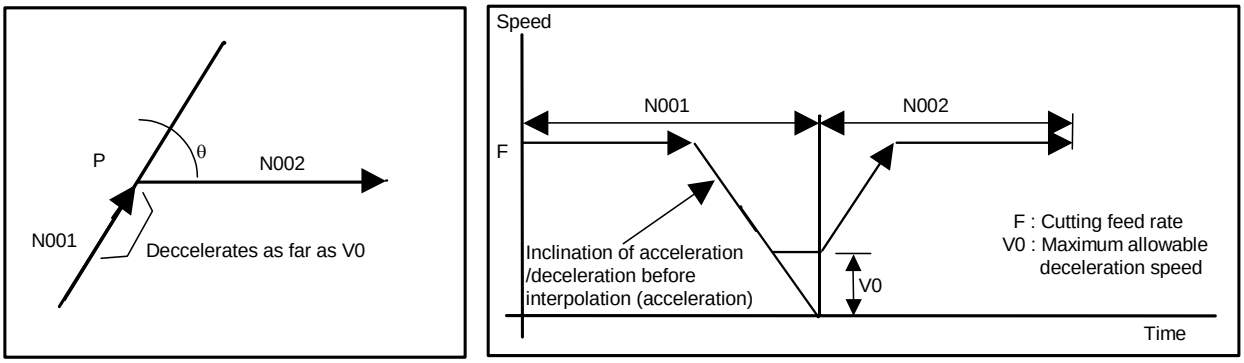
By accelerating /decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved.
With the arc commands, the radius reduction error can be significantly minimized.
Furthermore, since constant inclination acceleration/deceleration 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 corner deceleration [T system, M system]

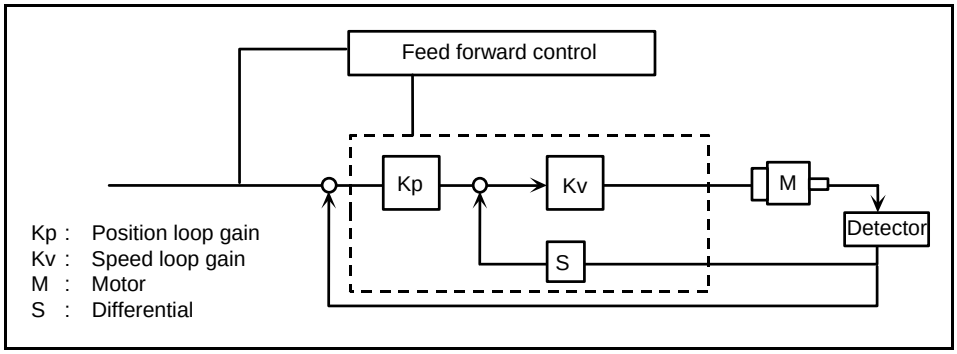
By determining the command vector in the machining program and thereby performing corner deceleration, it is possible to machine workpiece with a high-edge accuracy. The figure below shows the pattern of the deceleration speed at the corners.
(Optimum corner deceleration is a function of high-accuracy control mode.)

The speed change can be smoothed by the S-shape filter, the machine vibration can be suppressed, and the surface accuracy improved.
At the corner, the vector commanded in the machining program is automatically determined, and the speed is decelerated at the corner. A highly accurate edge can be machined by decelerating at the corner.



(3) Feed forward control

A stable servo control with an extremely small servo error can be realized using the feed forward control characteristic to this CNC system.



12.3 Programming Support Functions

12.3.2 Address Check

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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).

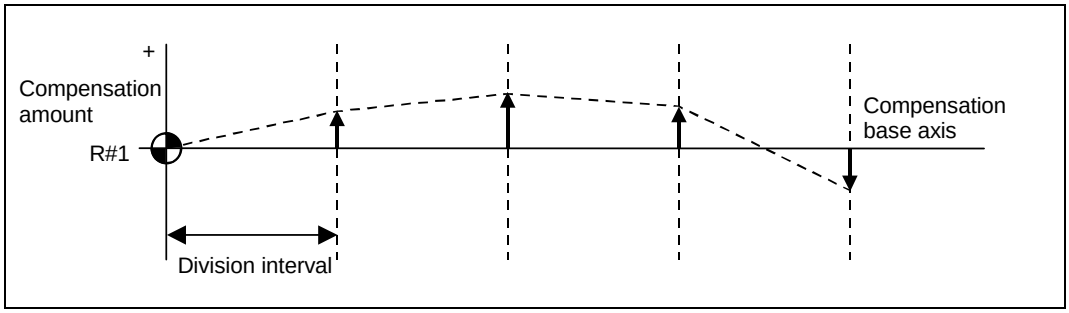
13.1.2 Memory-type Pitch Error Compensation

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 point serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed if return to reference point 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 (output unit)
4. No. of compensated axes : 10 axes (including number of axes for memory type pitch error compensation.)

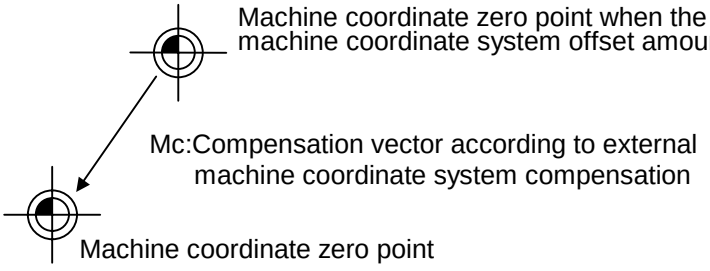
(1) The compensation position is set for the compensation axis whose reference point serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed if return to reference point is not made for the compensation base axis or compensation execution axis after the controller 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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.

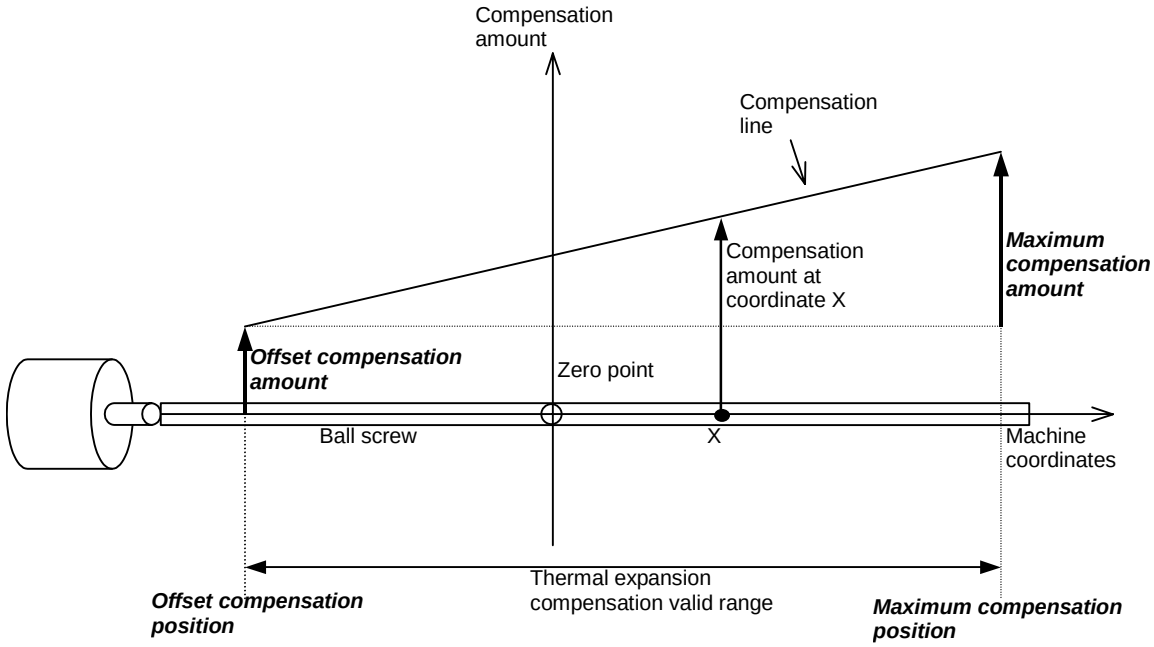


13.1.6 Ball Screw Thermal Expansion Compensation

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(1) Outline

The error in the axis feed caused by the thermal expansion of the ball screw is compensated with the value set in PLC I/F.



The offset compensation amount and maximum compensation amount are set from the PLC.
The compensation amount based on the offset compensation amount is set as the maximum compensation amount.
The offset compensation amount and maximum compensation amount are set beforehand in the parameters.

(2) Compensation operation

The offset compensation position and maximum compensation position are connected with a straight line following the designated compensation amount, and the compensation amount to the current coordinates is obtained and compensated. The compensation amount changes immediately when the offset compensation amount or maximum compensation amount changes.
The thermal expansion compensation is valid only between the offset compensation amount and maximum compensation position, and is "0" outside of this range.
The compensation amount is not included in the coordinate value display.

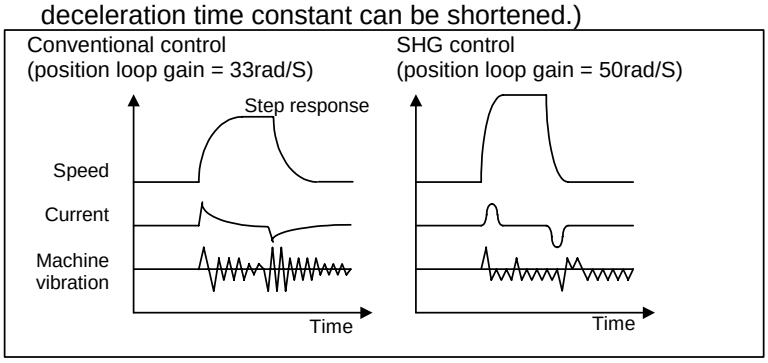
13.2 Dynamic Accuracy Compensation

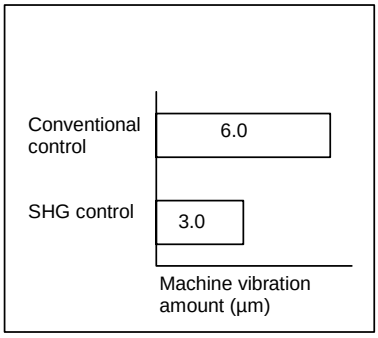
13.2.1 Smooth High-gain Control (SHG Control)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

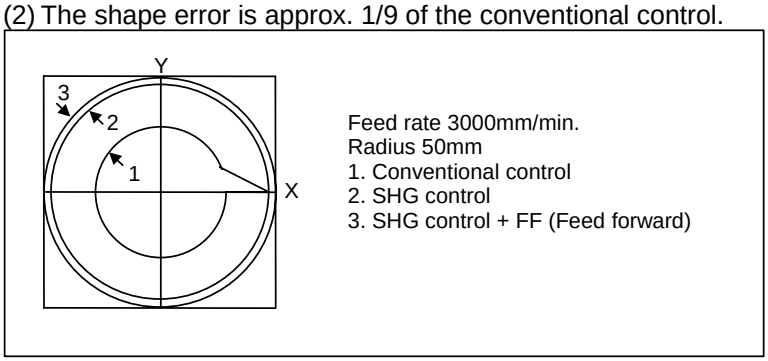
This is a high-response and stable position control method using the servo system (MDS-□-V□/SVJ2). This SHG control realizes an approximately three-fold position loop gain equally compared to the conventional control method.
The features of the SHG control are as follows.

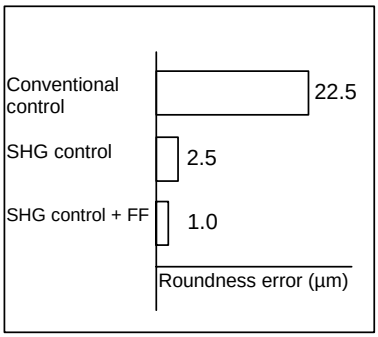
(1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/deceleration time constant can be shortened.)



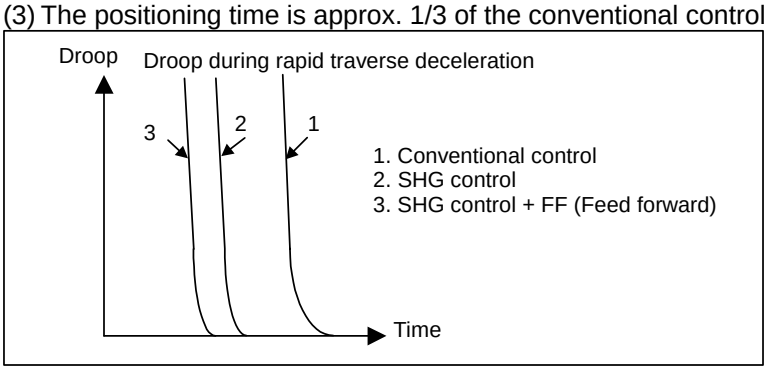


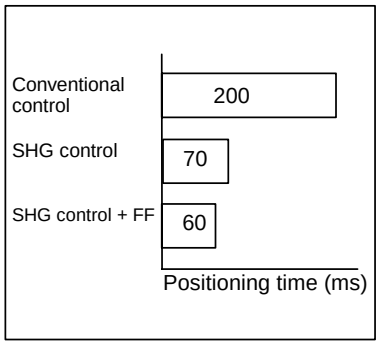
(2) The shape error is approx. 1/9 of the conventional control.





(3) The positioning time is approx. 1/3 of the conventional control.

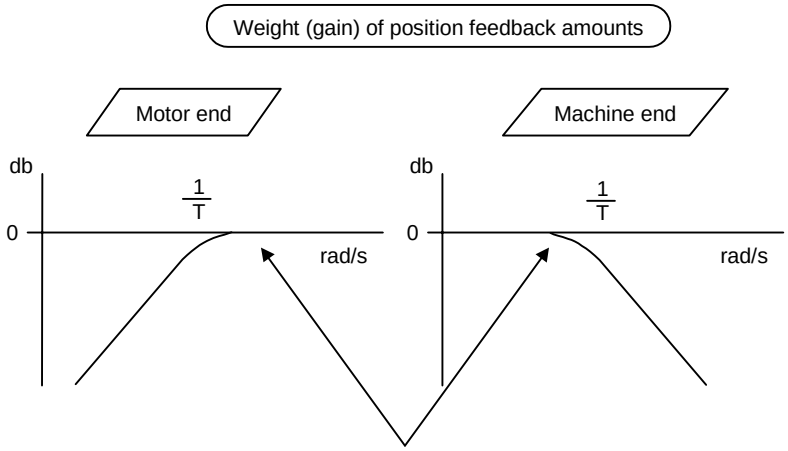




13.2.2 Dual Feedback

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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.



Time constant T here is adjusted using a parameter.

13.2.3 Lost Motion Compensation

C6		C64		
T system	L system	M system	L system	T 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

14.1.1 External Search

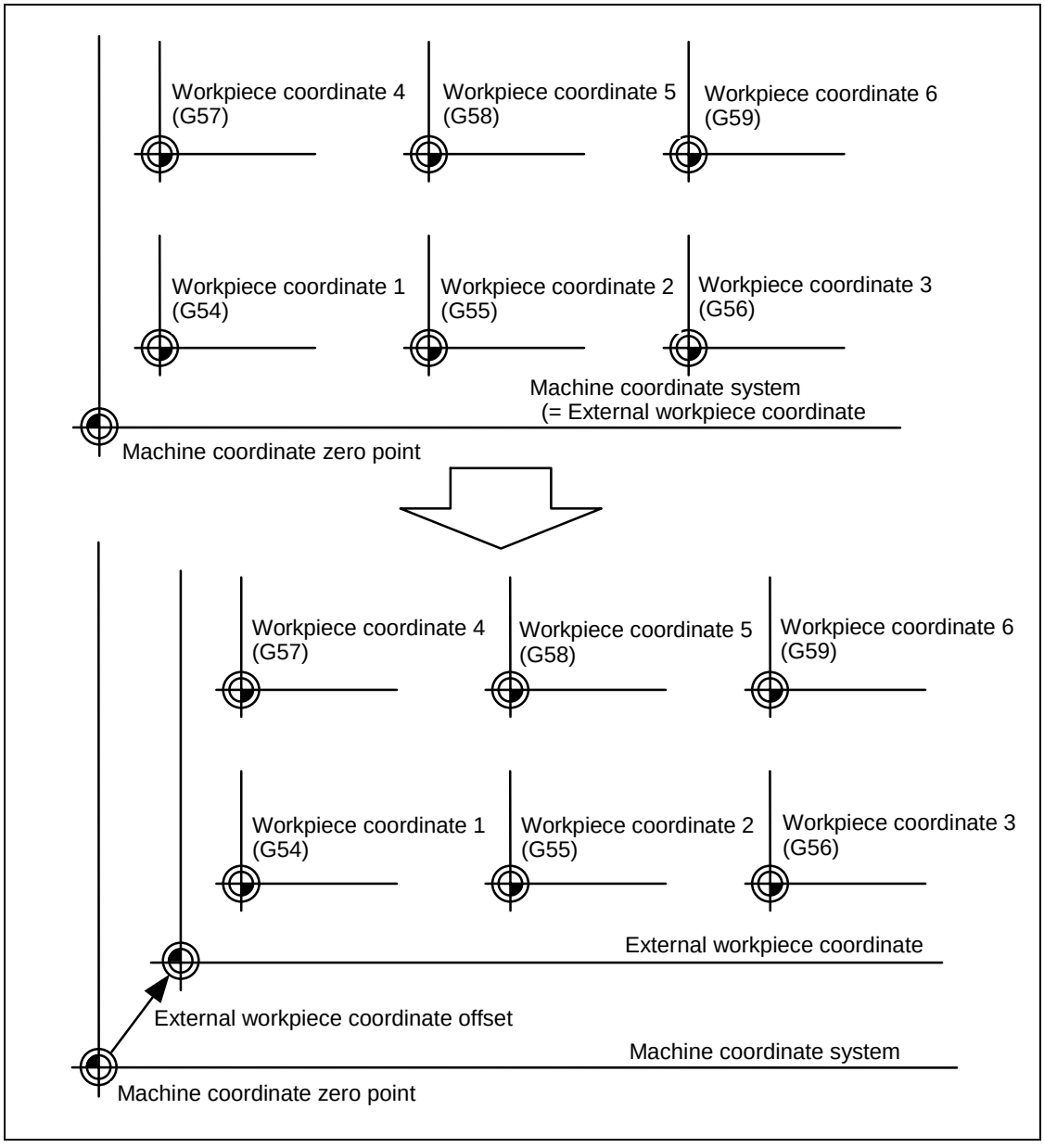
C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

This function enables the program numbers, sequence numbers and block numbers of machining programs, which are to be used in automatic operation, to be searched from the memory using the user PLC.
When a number is to be searched, the storage location of the program to be searched can be specified as the device number.
The currently searched contents (device number, program number, sequence number, block number) can be read from the PLC.

14.1.2 External Workpiece Coordinate Offset

C6		C64		
T system	L system	M system	L system	T 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.



14.2 Measurement; G31, G37

14.2.1 Skip

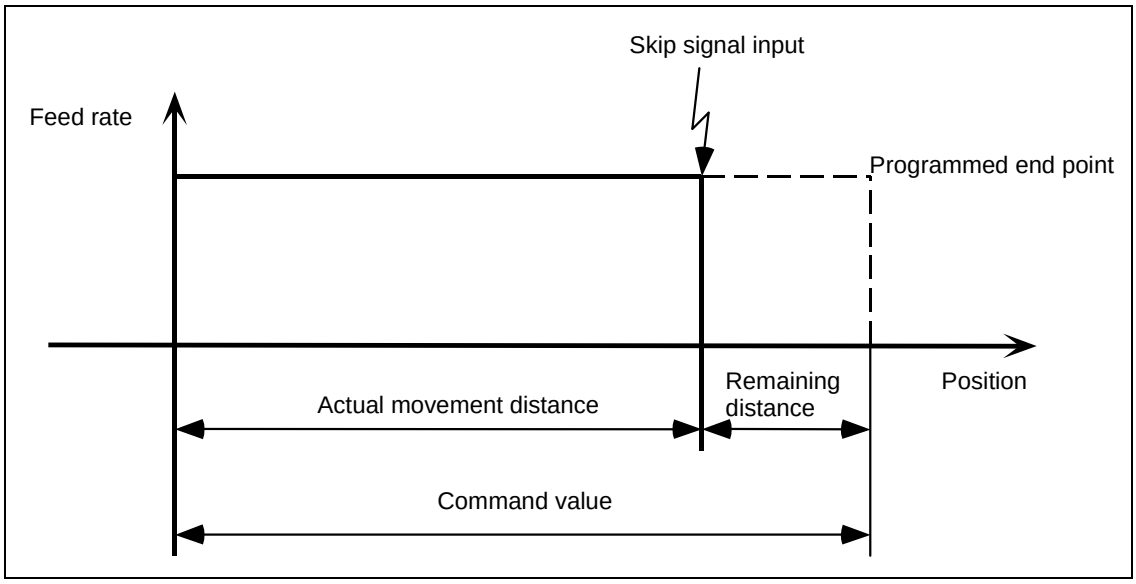
14.2.1.1 Skip

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 ;

G31 : 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(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 ;

G31.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 feedrate 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.
(c) When the skip signals are commanded in combination, the skip operation takes place with OR result of those signals.

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	○	○	○	○

14.2.5 Automatic Tool Length Measurement

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(1) Automatic Tool Length Measurement (T system, M system)

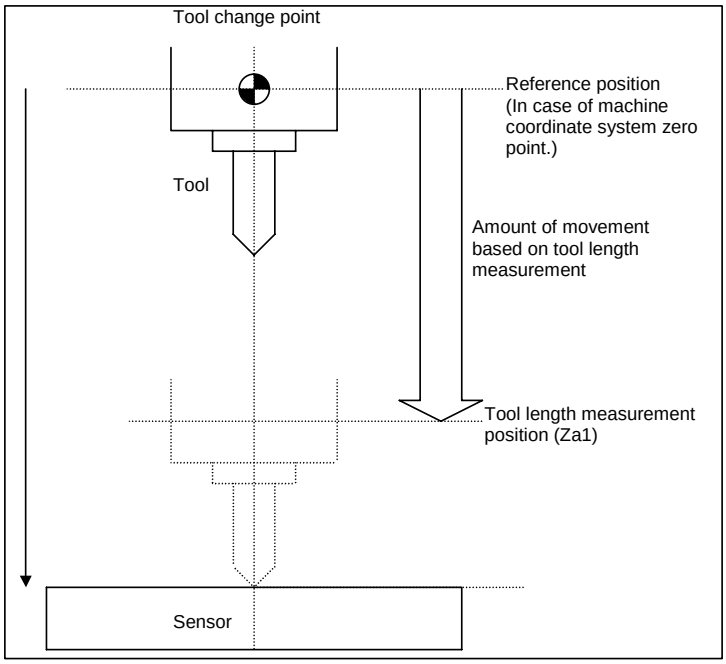
This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped 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, Y, Z, α (where α is an optional 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.



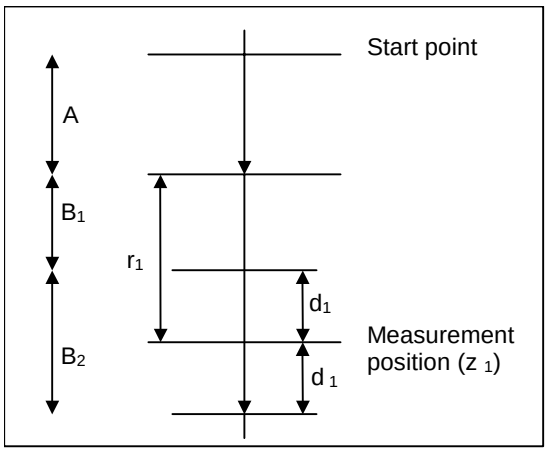
At this time, the tool length offset amount has a minus ("–") value.

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 which is then registered in the tool offset table.

(Note 1) The measurement position arrival signal (sensor signal) is also used as the skip signal.



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)

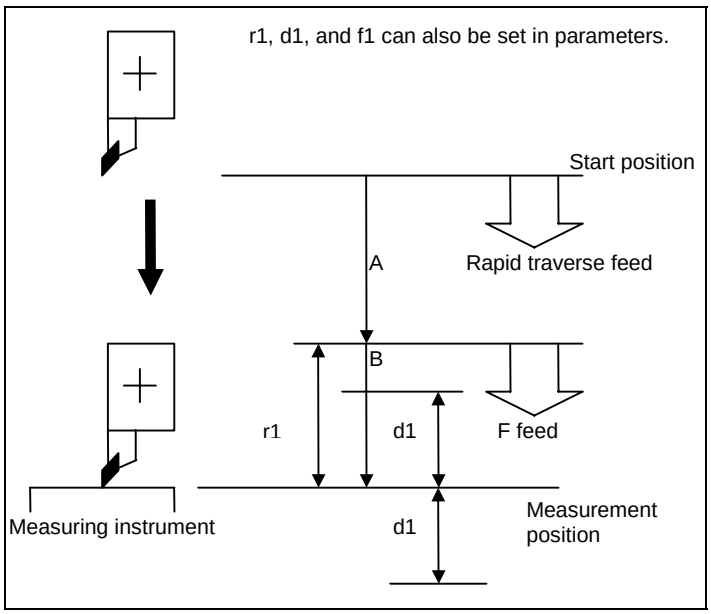
This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped 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.

(Note 1) The measurement position arrival signal (sensor signal) is also used as the skip signal.

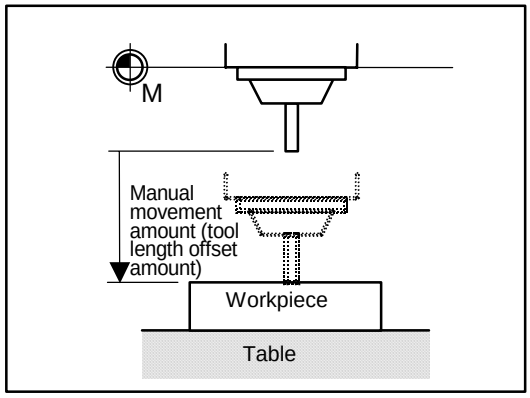
(Note 2) This is valid for the G code lists 2 and 3.

14.2.6 Manual Tool Length Measurement 1

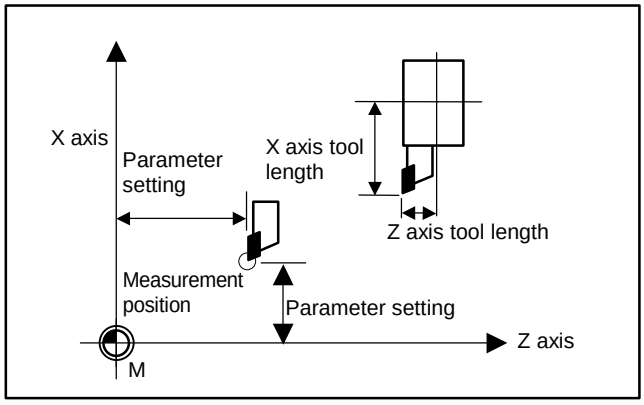
C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Simple measurement of the tool length is done without a sensor.

(1) Manual tool length measurement I [T system, M system]
When the tool is at the reference point, 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 measurement I [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.

14.3 Monitoring

14.3.1 Tool Life Management

14.3.1.2 Tool Life Management II

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

(1) T system, M system

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 system

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: Max. 40 sets (each part system)/ For 1 part system: 80 sets
- No. of tools in group: Max. 16 tools

14.3.2 Number of Tool Life Management Sets

The number of tools that can be managed for their lives are shown below. (These are fixed by the No. of part systems according to the model.)

20/40/80 sets

C6		C64		
T system	L system	M system	L system	T system
–	Δ80	–	Δ80	–

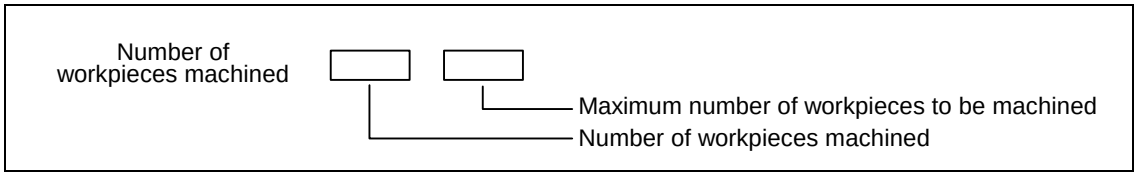
100/200 sets

C6		C64		
T system	L system	M system	L system	T system
Δ100	–	Δ100	–	Δ100

14.3.3 Display of Number of Parts

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The number of machined parts is counted up each time a part is machined, and displayed .



14.3.4 Load Meter

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Using the user PLC, this function displays the spindle load, Z-axis load, etc. in the form of bar graphs.

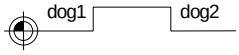
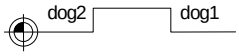
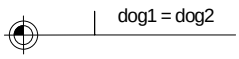
14.3.5 Position Switch

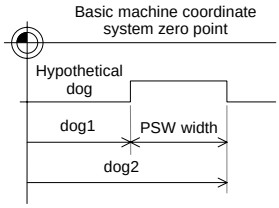
C6		C64		
T system	L system	M system	L system	T system
○16	○16	○16	○16	○16

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.
There can be a maximum of 16 switches for each part system.

Example of dog1, dog2 settings and execution

dog1, dog2 settings	dog1, dog2 positions	Description
dog1 < dog2		Signal is output between dog1 and dog2
dog1 > dog2		Signal is output between dog2 and dog1
dog1 = dog2		Signal is output at the dog1 (dog2) position



14.5 Others

14.5.1 Programmable Current Limitation

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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".

14.5.4 Automatic Restart

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The controller can be reset and the program started again from the head when the automatic restart signal is turned ON during program running.

15. Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

C6		C64		
T system	L system	M system	L system	T 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 on the setting and display unit goes OFF and the servo ready signal is turned OFF.

15.1.2 Data Protection Key

C6		C64		
T system	L system	M system	L system	T 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 setting and display unit.
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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The warnings which 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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The alarms which 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.
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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The stop cause of automatic operation is displayed on the setting and display unit.

15.2.4 Emergency Stop Cause

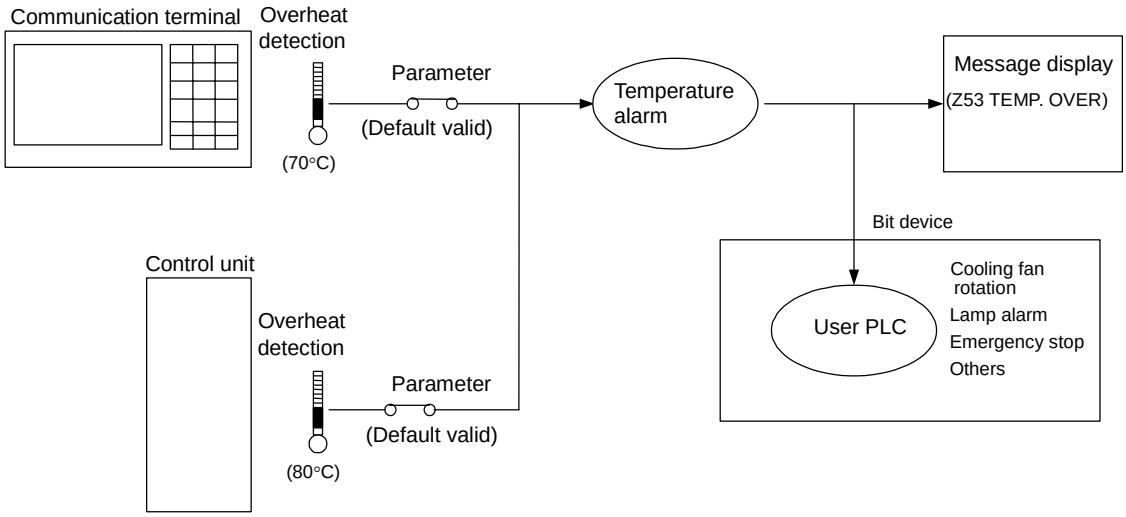
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

When "EMG" (emergency stop) message is displayed in the operation status display area of the setting and display unit, the emergency stop cause can be confirmed.

15.2.5 Temperature Detection

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

When overheating is detected in the control unit or the communication terminal, an overheat signal is output at the same time as the alarm is displayed. If the system is in auto run at the time, run is continued, but it cannot be started after reset or M02/M30 run ends. (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 for the control unit or 70°C or more for the communication terminal.



(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)

C6		C64		
T system	L system	M system	L system	T 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.

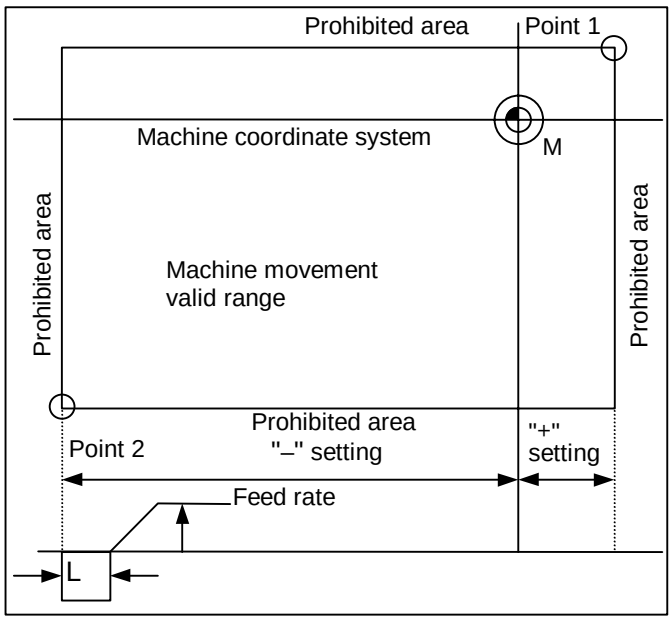
Type	Prohibited range	Explanation
I	Outside	•Set by the machine maker. •When used with II, the narrow range designated by the two types becomes the movement valid range. •Can be rewritten with DDB.
II	Outside	•Set by the user. •The change or function of parameter can be turned OFF/ON with the program command. •Select II or IIB with the parameters. •Can be rewritten with DDB.
IIB	Inside	
IB	Inside	•Set by the machine maker.
IC	Outside	•Set by the machine maker. •Can be rewritten with DDB.

15.3.2.1 Stored Stroke Limit I/II

C6		C64		
T system	L system	M system	L system	T 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 point 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.

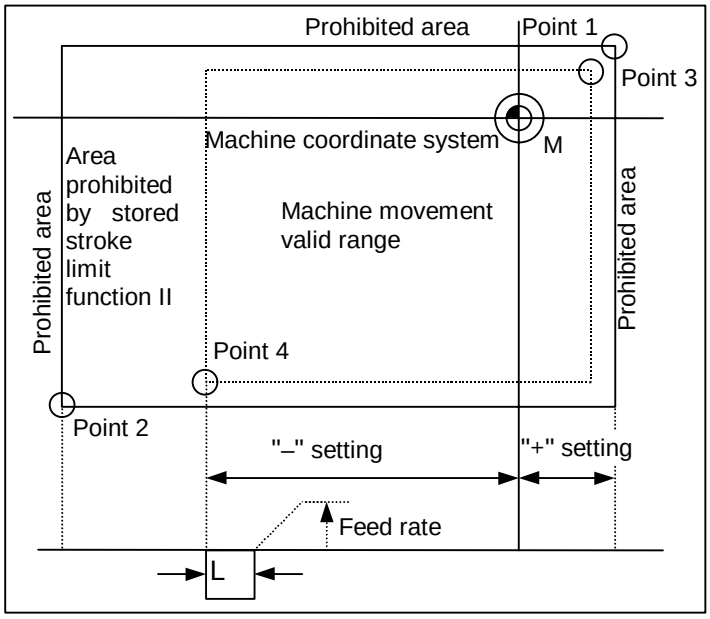
(2) Stored Stroke Limit II

This is the stroke limit function which 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.

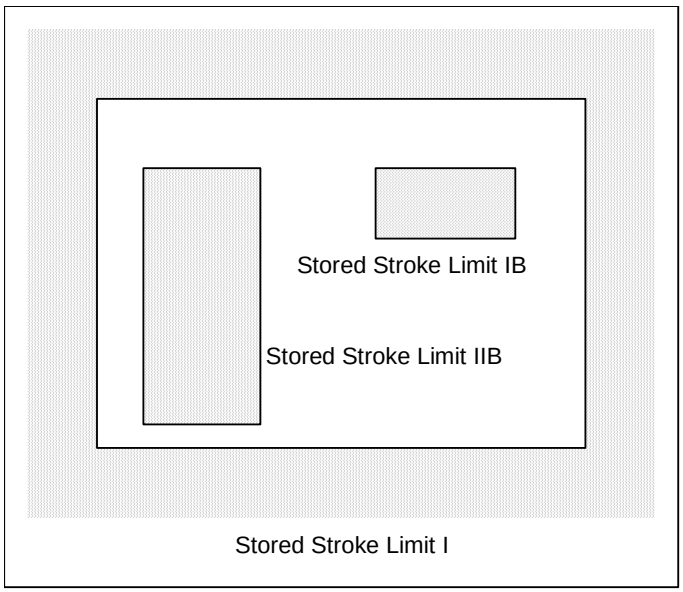
The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

The stored stroke limit II function can also be invalidated with the parameter settings.

15.3.2.2 Stored Stroke Limit IB

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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.
This function is an option.

Precautions

- 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

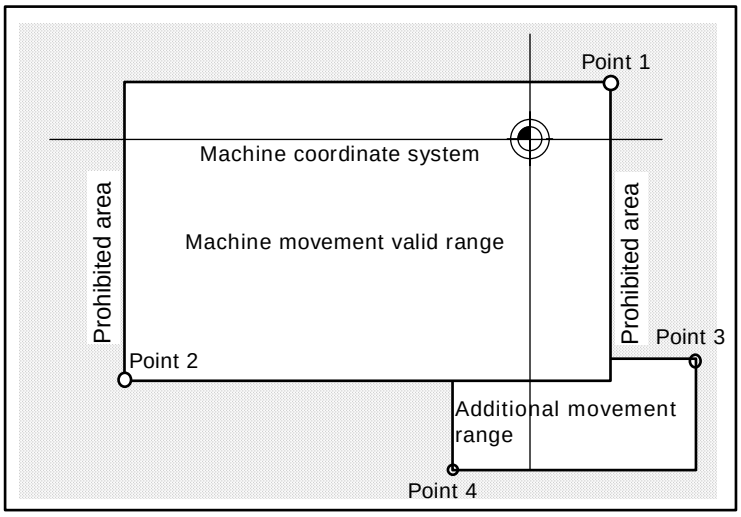
C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 soft limit IB.



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.

15.3.3 Stroke Check Before Movement

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

By assigning commands in the program to designate the boundaries beyond which machine entry is prohibited using the coordinate values in the machine coordinate system, this function ensures that machine entry inside these boundaries is prohibited.
Whereas the regular stored stroke limit function stops the machine immediately in front of the set prohibited area, the stroke check before movement function raises a program alarm before the machine initiates the movement in a block containing a command which calls for the machine to move beyond the movement enabled range.

15.3.4 Chuck/Tail Stock Barrier Check; G22/G23

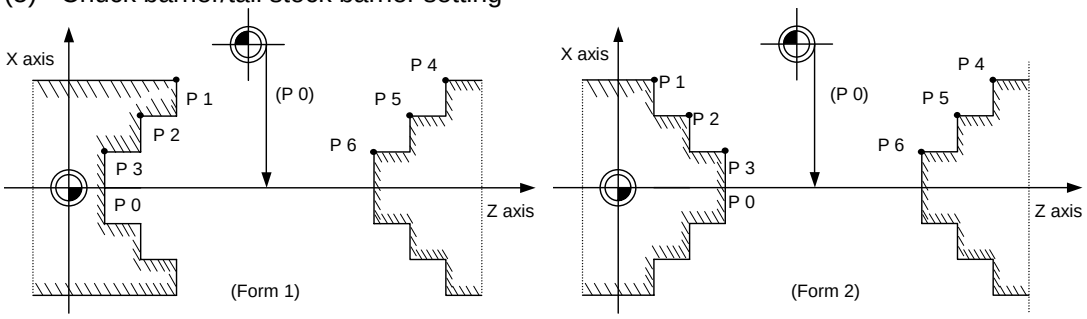
C6		C64		
T system	L system	M system	L system	T system
–	○	–	○	–

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 point return.
(4) The chuck barrier/tail stock barrier can be set independently for part system 1 and part system 2.
(5) 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

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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 drive units 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

C6		C64		
T system	L system	M system	L system	T 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 drive units 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.

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

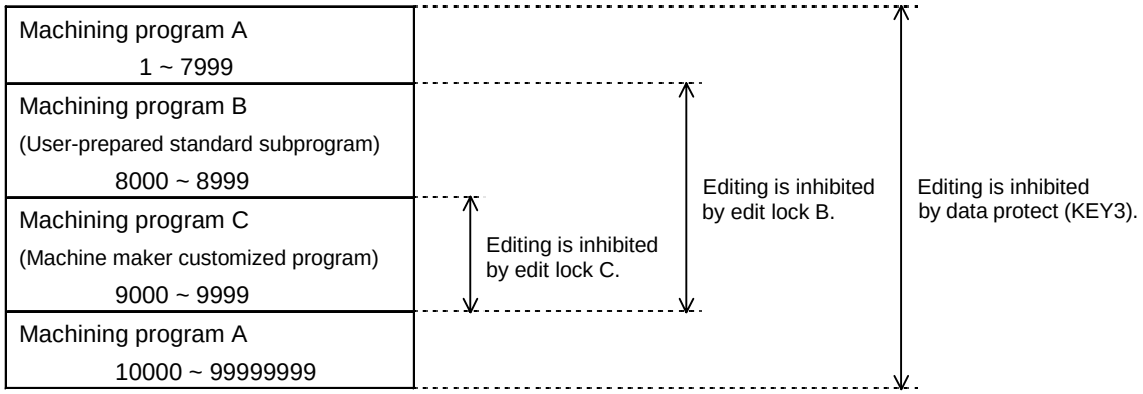
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

This function is used to prohibit changing the setup parameter.

15.3.10 Program Protect (Edit Lock B, C)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The edit lock function B or C inhibits machining program B or C (group with machining program numbers) from being edited or erased when these programs require to be protected.



15.3.11 Program Display Lock

C6		C64		
T system	L system	M system	L system	T 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

C6		C64		
T system	L system	M system	L system	T 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.
The part system information is displayed as the alarm histories.
For instance, "$1" denotes the first part system, and "$2" the second part system.
The history data containing the most recent operation history and event occurrence times (2,068 sets) are displayed on the "Operation history" screen. The most recent history data appears at the top of the screen, and the older data is displayed in sequence below.

(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

C6		C64		
T system	L system	M system	L system	T system
○ monitor	○ monitor	○ monitor	○ monitor	○ monitor

The information on the servos (NC axes), spindles, PLC axes and power supplies appears on the setting and display unit.
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

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Sampling of the servo and spindle data for which an alarm occurrence is a stop condition is performed all the time.

15.4.5 Machine Operation History Monitor

C6		C64		
T system	L system	M system	L system	T 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

This function serves to back up the parameters and other data of the NC control unit.
The data can also be restored.

(1) RS-232C

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

[Backup target]

Machining programs, parameters, workpiece offset data, common variables, tool compensation data, tool life control data
Ladders (ladder, message)

(2) IC card

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

[Backup target]

Machining programs, parameters, common variables, tool compensation data, tool life control data
Ladders (ladder, message)

15.4.7 PLC I/F Diagnosis

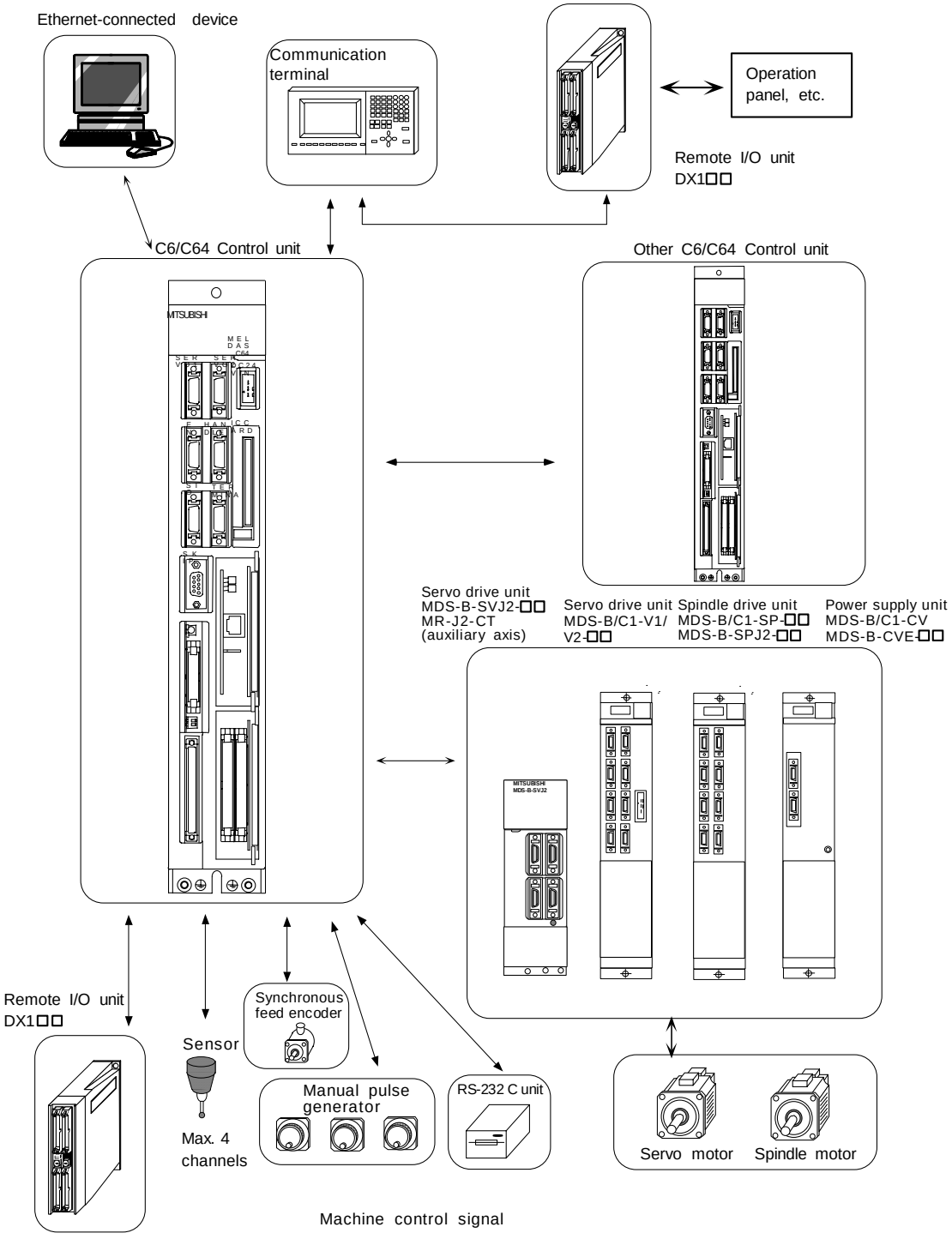
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

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.

16. Cabinet and Installation

16.1 Cabinet Construction

The configuration of the unit used by the MELDAS C6/C64 series is shown below.
Refer to the Connection / Maintenance Manual for details.



List of configuration units

(1) Control unit

Type		Configuration element	Details
FCU6-MU043 FCU6-MU042	C6 Control unit C64 Control unit	HR851 card	Main card
		HR891 card	Back panel
		HR899 card	IC card interface

(2) Extension unit

Type		Configuration element	Details
FCU6-EX871	DeviceNet (Master)	HR871 card	Expansion card
FCU6-EX872	DeviceNet (Slave)	HR872 card	Expansion card
FCU6-EX873	FL-net	HR873 card	Expansion card
FCU6-EX875	Ethernet	HR875/876 card	Expansion card, Use as set
FCU6-EX878	MELSECNET/10 (Coaxial interface)	HR877/878 card	Expansion card, Use as set
FCU6-EX879	MELSECNET/10 (Optical interface)	HR877/879 card	Expansion card, Use as set
FCU6-HR865	CC-Link	HR865 card	Expansion card
FCU6-EX871-40	DeviceNet	HR871 card	Expansion card
FCU6-HR881	Extension DIO (Sink type)	HR881 card	Expansion card
FCU6-HR882	Extension DIO (Sink type, with AO)	HR882 card	Expansion card
FCU6-HR883	Extension DIO (Source type)	HR883 card	Expansion card
FCU6-HR884	Extension DIO (Source type, with AO)	HR884 card	Expansion card
FCU6-HR893	External extension unit	HR893 card	Extension back panel, a set of metal plates

(3) Communication terminal (Display unit/ NC keyboard)

Type		Configuration element	Details
FCUA-LD100	7.2-type monochrome LCD with integrated keyboard (Integrated type/machining system sheet)	7.2- type monochrome LCD	Control card 24VDC input
		RX213 card	
		Key switch / escutcheon	
FCUA-LD10	7.2- type monochrome LCD with display unit (Keyboard separated type)	7.2- type monochrome LCD	Use as set with FCUA-KB20 Control card 24VDC input
		Escutcheon	
		RX213 card	
FCU6-DUT32	10.4- type monochrome LCD with display unit (Keyboard separated type)	10.4- type monochrome LCD	Use as set with FCUA-KB20 Control card 24VDC input
		Escutcheon	
		RX215 card	
FCUA-CT100	Keyboard integrated type with 9- type CRT (Integrated type/machining system sheet)	9- type CRT	Control card 24VDC input CRT 100VAC input
		RX211 card	
		Key switch / escutcheon	
FCUA-CT120	Keyboard integrated type with 9- type CRT (Integrated type/lathe system sheet)	9- type CRT	Control card 24VDC input CRT 100VAC input
		RX211 card	
		Key switch / escutcheon	
FCUA-CR10	Display unit with 9- type CRT (Keyboard separated type)	9- type CRT	Use as set with FCUA-KB10 Control card 24VDC input CRT 100VAC input
		Escutcheon	
FCUA-KB10	Keyboard (Separated type/machining system sheet)	Key switch	Use as set with FCUA-CR10
		RX211 card	
FCUA-KB20	Keyboard (Separated type/machining system sheet)	Key switch	Use as set with FCUA-LD10 or FCU6-DUT32
FCU6-KB021	Keyboard (Separated type/machining system sheet)	Key switch	Use as set with FCU6-DUT32 (FCUA-KB20 with changed outline dimensions)
FCUA-KB30	Keyboard (Separated type/lathe system sheet)	Key switch	Use as set with FCUA-LD10 or FCU6-DUT32
FCU6-KB031	Keyboard (Separated type/lathe system sheet)	Key switch	Use as set with FCU6-DUT32 (FCUA-KB30 with changed outline dimensions)

(4) Peripheral device

Type		Configuration element	Details
HD60	Manual pulse generator		Without MELDAS logo
HD60-1	Manual pulse generator		With MELDAS logo
Ground plate D			Grounding plate D, one set
Ground plate E			Grounding plate E, one set

(5) Remote I/O unit

Type		Configuration element	Details
FCUA-DX100	DI (sink/source)/DO (sink) = 32/32	RX311	Base PCB : DI (sink/source)/ DO (sink) = 32/32
		Case	
FCUA-DX110	DI (sink/source)/DO (sink) = 64/48	RX311	Base PCB : DI (sink/source)/ DO (sink) = 32/32
		RX321-1	Add-on PCB : DI (sink/source)/ DO (sink) = 32/16
		Case	
FCUA-DX120	DI (sink/source)/DO (sink) = 64/48 Analog output 1 point	RX311	Base PCB : DI (sink/source)/ DO (sink) = 32/32
		RX321	Add-on PCB : DI (sink/source)/ DO (sink) = 32/16 analog output 1 point
		Case	
FCUA-DX130	DI (sink/source)/DO (sink) = 32/32 Manual pulse 2ch	RX311	Base PCB : DI (sink/source)/ DO (sink) = 32/32
		RX331	Add-on PCB : Manual pulse generator 2ch
		Case	
FCUA-DX140	DI (sink/source)/DO (sink) = 32/32 Analog input 4 points Analog output 1 point	RX311	Base PCB : DI (sink/source)/ DO (sink) = 32/32
		RX341	Add-on PCB : Analog input 4 points, analog output 1 point
		Case	
FCUA-DX101	DI (sink/source)/ DO (source) = 32/32	RX312	Base PCB : DI (sink/source)/ DO (source) = 32/32
		Case	
FCUA-DX111	DI (sink/source)/ DO (source) = 64/48	RX312	Base PCB : DI (sink/source)/ DO (source) = 32/32
		RX322-1	Add-on PCB : DI (sink/source)/ DO (source) = 32/16
		Case	
FCUA-DX121	DI (sink/source)/ DO (source) = 64/48 Analog output 1 point	RX312	Base PCB : DI (sink/source)/ DO (source) = 32/32
		RX322	Add-on PCB : DI (sink/source)/ DO (source) = 32/16 analog output 1 point
		Case	
FCUA-DX131	DI (sink/source)/ DO (source) = 32/32 Manual pulse 2ch	RX312	Base PCB : DI (sink/source)/ DO (source) = 32/32
		RX331	Add-on PCB : Manual pulse generator 2ch
		Case	
FCUA-DX141	DI (sink/source)/ DO (source) = 32/32 Analog input 4 points, analog output 1 point	RX312	Base PCB : DI (sink/source)/ DO (source) = 32/32
		RX341	Add-on PCB : Analog input 4 points, analog output 1 point
		Case	

16.2 Power Supply, Environment and Installation Conditions

⚠ 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

Unit name			Control unit
Type			FCU6-MU043/MU042/MU041
General specifications	Ambient temperature	During operation	0 to 55°C
		During storage	–20 to 60°C
	Ambient humidity	During operation	Long term, Up to 75% RH (with no dew condensation) Short term (Within 1 month), Up to 95% RH (with no dew condensation)
		During storage	Up to 75% RH (with no dew condensation)
	Vibration resistance		4.9m/s^2 or less (during operation)
	Shock resistance		29.4m/s^2 or less (during operation)
	Working atmosphere		No corrosive gases, dust or oil mist
	Power noise		1kV (P-P)
Power specifications	Power voltage		24VDC±5% Ripple ±5% (P-P)
	Instantaneous stop tolerance time		2.1ms (during 24VDC line cutting)
	Current consumption		3A (max.)
Heating value			70W (during full option)
Mass			1.6kg
Unit size			Refer to Appendix.

(2) Communication terminal

Unit name			Communication terminal		
Type			FCUA-LD100/ FCUA-LD10+KB20	FCU6-DUT32 +KB021	FCUA-CT100/ FCUA-CR10+KB10
General specifications	Ambient temperature	During operation	0 to 50°C		0 to 55°C
		During storage	–20 to 60°C		–20 to 65°C
	Ambient humidity	During operation	Long term, Up to 75% RH (with no dew condensation) Short term (Within 1 month), Up to 95% RH (with no dew condensation)		
		During storage	Up to 75% RH (with no dew condensation)		
	Vibration resistance		4.9m/s^2 or less (during operation)		
	Shock resistance		29.4m/s^2 or less (during operation)		
	Working atmosphere		No corrosive gases, dust or oil mist		
	Power noise		1kV (P-P)		
Power specifications	Power voltage		24VDC±5% Ripple ±5% (P-P)		Single phase 100 to 115VAC –15%+10% 50/60Hz±5%
					24VDC±5% Ripple ±5% (P-P)
	Instantaneous stop tolerance time		Follows specifications of 24VDC power supply being used		
	Current consumption		24V, 0.9A		100V, 0.4A 24V, 0.6A
Heating value			20W		55W
Mass			1600g	2200g	4800g
Unit size			Refer to Appendix.		

(3) Remote I/O unit

Unit name			Remote I/O unit				
Type			FCUA-DX10□	FCUA-DX11□	FCUA-DX12□	FCUA-DX13□	FCUA-DX14□
General specifications	Ambient temperature	During operation	0 to 55°C				
		During storage	–20 to 65°C				
	Ambient humidity	During operation	Long term, Up to 75% RH (with no dew condensation) Short term (Within 1 month), Up to 95% RH (with no dew condensation)				
		During storage	Up to 75% RH (with no dew condensation)				
	Vibration resistance		4.9m/s^2 or less (during operation)				
	Shock resistance		29.4m/s^2 or less (during operation)				
	Working atmosphere		No corrosive gases, dust or oil mist				
	Power noise		1kV (P-P)				
Power specifications	Power voltage		24VDC±5% Ripple ±5% (P-P)				
	Instantaneous stop tolerance time		–				
	Current consumption		24V, 0.7A **(Note 1)**	24V, 1.5A **(Note 1)**		24V, 0.7A **(Note 1)**	
Heating value			25W **(Note 2)**	30W **(Note 2)**		30W **(Note 2)**	
Mass			470g	570g	590g	550g	
Unit size			Refer to Appendix.				

(Note 1) Only the amount consumed by the control circuit.
(Note 2) When all points of the machine input/output interface circuit are operating.

(4) Servo / Spindle

Refer to the following manuals for details on the servo and spindle system.

MDS-C1 Series	Specification Manual	(BNP-C3040)
MDS-B-SVJ2 Series	Specifications and Instruction Manual	(BNP-B3937)
MDS-B-SPJ2 Series	Specification and Instruction Manual	(BNP-B2164)
MDS-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	Specification Manual	(BNP-C3040)
MDS-B-SVJ2 Series	Specifications and Instruction Manual	(BNP-B3937)
MDS-B-SPJ2 Series	Specification and Instruction Manual	(BNP-B2164)
MDS-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: HC□□-A51/E51 (1000kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	–	☐	–	☐

(2) Servo motor: HC□□-A42/E42 (100kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.1.4 MDS-B-SVJ2 (Compact and Small Capacity)

(1) Servo motor: HC□□-A42/E42 (100kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(2) Servo motor: HC□□-A47 (100kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(3) Servo motor: HC□□-A33/E33 (25kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.1.6 MDS-R-V1/R-V2 (200V Compact and Small Capacity)

(1) Servo motor: HF□□-A51/E51 (1000kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	–	☐	–	☐

(2) Servo motor: HF□□-A42/E42 (100kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(3) Servo motor: HF□□-A47 (100kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.2 Spindle

17.2.1 MDS-C1-SP/C1-SPM/B-SP (200V)

(1) Spindle motor: SJ/SJ-V

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.2.3 MDS-B-SPJ2 (Compact and Small Capacity)

(1) Spindle motor: SJ-P/SJ-PF

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.3 Auxiliary Axis

17.3.1 Index/Positioning Servo: MR-J2-CT

(1) Servomotor: HC-SF/HC-RF (16kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

(2) Servomotor: HA-FF/HC-MF (8kp/rev)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.4 Power Supply

17.4.1 Power Supply: MDS-C1-CV/B-CVE

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.4.2 AC Reactor for Power Supply

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

17.4.3 Ground Plate

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

17.4.4 Power Supply: MDS-A-CR (Resistance Regeneration)

C6		C64		
T system	L system	M system	L system	T system
☐	☐	☐	☐	☐

18. Machine Support Functions

18.1 PLC

18.1.1 PLC Basic Function

18.1.1.1 Built-in PLC Basic Function

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

(1) Ladder commands

Basic commands (bit processing commands)
c LD, LDI, OR, ORI, AND, ANI, OUT, PLS, etc.
Function commands
192 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
5 commands including ATC control
Tool life management
12 types of network related commands

(2) Devices

The device number for devices X, Y, B, W and H are expressed with a hexadecimal. All other device numbers are expressed as decimals.

Device	Device range		Units	Details
X*	X0 to XAFF	2816 points	1-bit	Input signals to the PLC. Machine input, etc.
Y*	Y0 to YE7F	3712 points	1-bit	Output signals from the PLC. Machine output, etc.
M	M0 to M8191	8192 points	1-bit	For temporary memory
L	L0 to L255	256 points	1-bit	Latch relay (Backup memory)
F	F0 to F127	128 points	1-bit	For temporary memory. Alarm message interface
SB	SB0 to SB1FF	512 points	1-bit	Special relay for links
B	B0 to B1FFF	8192 points	1-bit	Link relay
SM*	SM0 to SM127	128 points	1-bit	Special relay
V	V0 to V255	256 points	1-bit	Edge relay
SW	SW0 to SW1FF	512 points	16-bit	Special register for links
SD	SD0 to SD127	128 points	16-bit	Special register
T	T0 to T15	16 points	1-bit/16-bit	10ms unit timer
	T16 to T95	80 points	1-bit/16-bit	100ms unit timer
	T96 to T103	8 points	1-bit/16-bit	100ms incremented timer
	T104 to T143	40 points	1-bit/16-bit	10ms unit timer (Fixed timers)
	T144 to T239	96 points	1-bit/16-bit	100ms unit timer (Fixed timers)
	T240 to T255	16 points	1-bit/16-bit	100ms incremented timer (Fixed timers)
	T0000 to T0255	256 points	1-bit	T1: Timer coil
	T1000 to T1255	256 points	1-bit	T0: Timer contact
	T2000 to T2255	256 points	16-bit	TS: Timer setting value
	T3000 to T3255	256 points	16-bit	TA: Timer current value
C	C0 to C23	24 points	1-bit/16-bit	Counter
	C24 to C127	104 points	1-bit/16-bit	Counter (Fixed counters)
	C0000 to C0127	128 points	1-bit	C1: Counter coil
	C1000 to C1127	128 points	1-bit	C0: Counter contact
	C2000 to C2127	128 points	16-bit	CS: Counter setting value
	C3000 to C3127	128 points	16-bit	CA: Counter current value
D	D0 to D8191	8192 points	16-bit/32-bit	Data register
R*	R0 to R8191	8192 points	16-bit/32-bit	File register. CNC word I/F
W	W0 to W1FFF	8192 points	16-bit/32-bit	Link register
Z	Z0 to Z13	14 points	16-bit	Address index
N	N0 to N7			Master control's nesting level
P*	P0 to P255 P360 to P379			Conditional jump, subroutine call label
K	K-32768 to K32767			Decimal constant for 16-bit command
	K-2147483647 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 1) Devices with an asterisk in the device field have sections with predetermined applications. Do not use these devices for other applications.

(Note 2) 8192 points of D device are available on the S/W version D or higher.

(3) External alarm messages

The contents of the alarms which have occurred during sequence (user PLC) processing can be displayed on the setting and display unit.

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 setting and display unit 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 setting and display unit.

(6) Load meter display

A load meter can be displayed on the setting and display unit.

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 setting and display unit.

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.

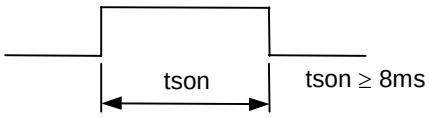
(11) Workpiece counter display (parts counter)

The number of parts can be set and displayed when continuously machining parts.
The M code to be count, the current number of machined parts 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 parts 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.

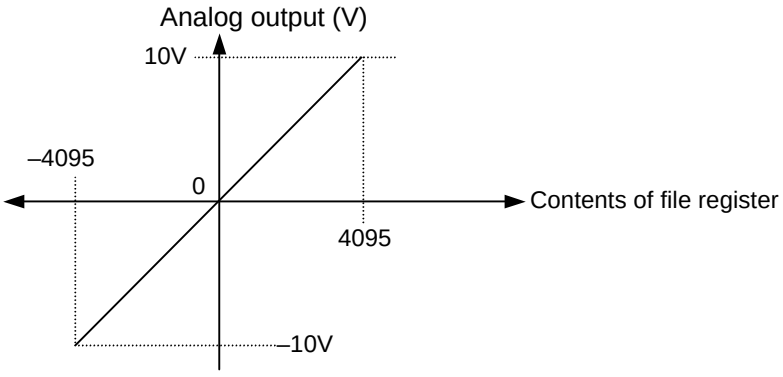
(c) The signals used for high-speed processing are assigned with the parameters. Assignment is possible in a continuous 16-point unit.

(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 DX120/DX121 is 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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)

C6		C64		
T system	L system	M system	L system	T system
○32000	○32000	○32000	○32000	○32000

There are four bytes for each step.

18.1.4 Machine Contact Input/Output I/F

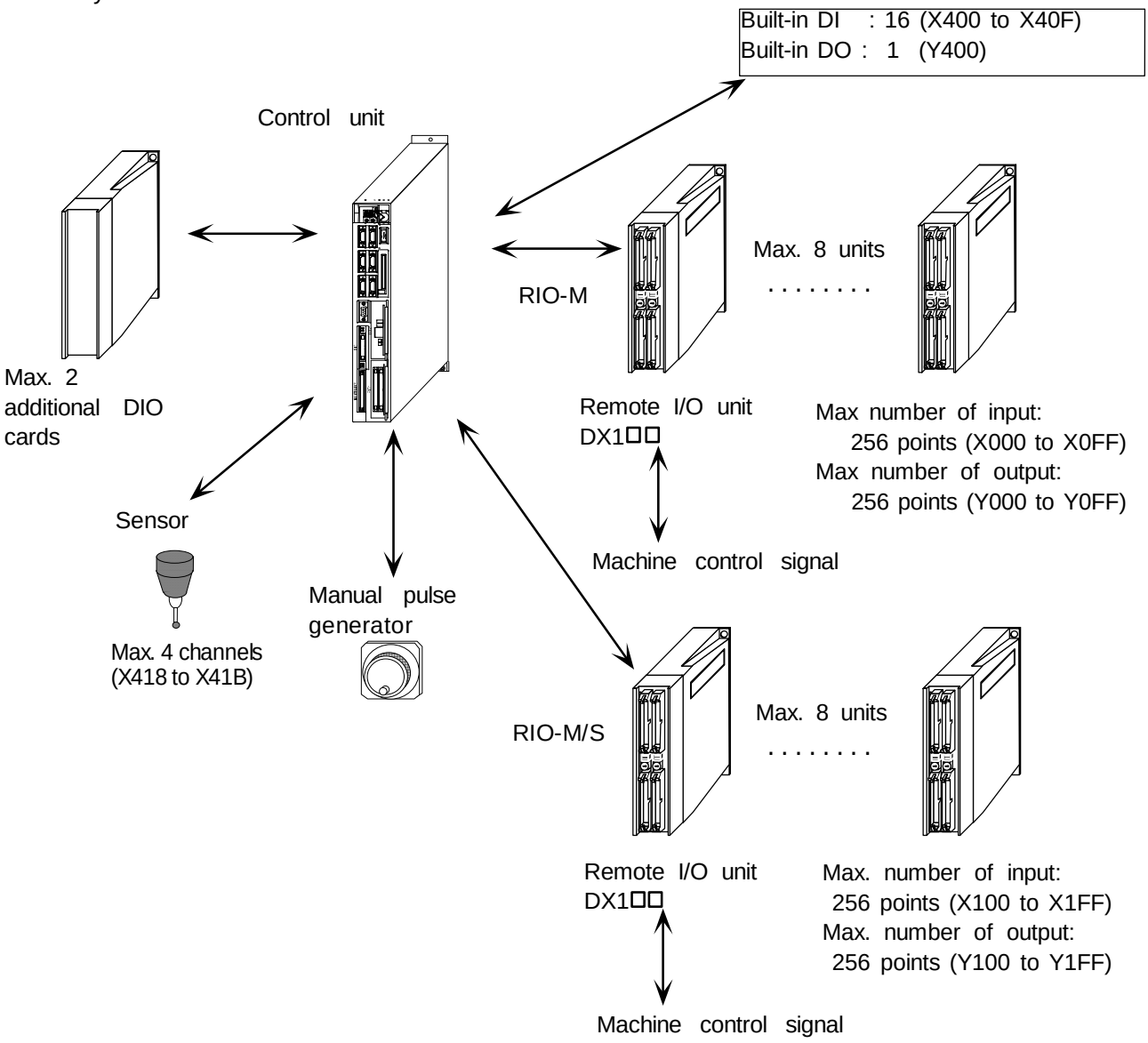
C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

⚠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 the MELDAS C6/C64 Connection/Maintenance Manual for details.

The machine contacts can be input or output using the internal DI/O and remote I/O, as shown in the figure below.
There are two kinds of DI/O, the sink type and source type. A 24V power supply must be provided externally for this DI/O.



Refer to the Connection Manual for details.

(1) Types of remote I/O units

The remote I/O units (FCUA-DX□□□) are 10 shown in the remote I/O unit list according to the types of signals that can be input/output and the no. of contacts. There are 10 types, and are used as a control unit.

Multiple remote I/O units can be combined for use if the total of possessed channel during the serial link connection is less than eight.

Remote I/O unit list

Unit model	Compatible machine control signals	No. of channels possessed by serial link
FCUA-DX100	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Sink type	1
FCUA-DX101	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Source type	1
FCUA-DX110	Digital input signal (DI) : 64 points (insulation) Common for sink/source Digital output signal (DO): 48 points (non-insulated) Sink type	2
FCUA-DX111	Digital input signal (DI) : 64 points (insulation) Common for sink/source Digital output signal (DO): 48 points (non-insulated) Source type	2
FCUA-DX120	Digital input signal (DI) : 64 points (insulation) Common for sink/source Digital output signal (DO): 48 points (non-insulated) Sink type Analog output (AO) : 1 point	2
FCUA-DX121	Digital input signal (DI) : 64 points (insulation) Common for sink/source Digital output signal (DO): 48 points (non-insulated) Source type Analog output (AO) : 1 point	2
FCUA-DX130	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Sink type Handle input : 2 handles	2
FCUA-DX131	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Source type Handle input : 2 handles	2
FCUA-DX140	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Sink type Analog input : 4 points Analog output : 1 point	2
FCUA-DX141	Digital input signal (DI) : 32 points (insulation) Common for sink/source Digital output signal (DO): 32 points (non-insulated) Source type Analog input : 4 points Analog output : 1 point	2

(Note) The power for the input/output signal drive unit and receiver must be prepared by the machine maker.

Interface specifications

Input specifications

	Sink type	Source type
Input voltage when ON	0 to 6V	18 to 24V
Input voltage when OFF	20 to 24V	0 to 4V

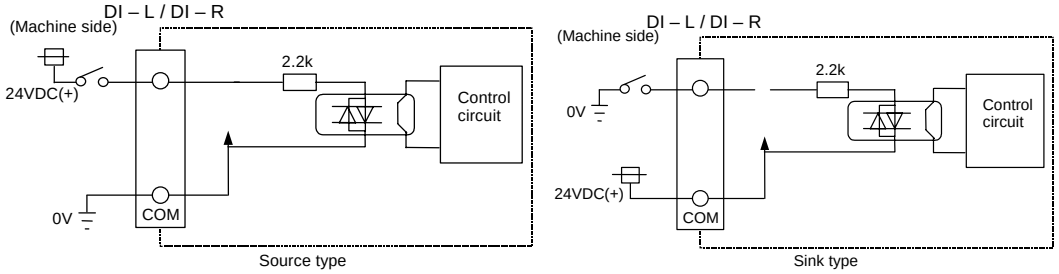
Output specifications

Rated load voltage	24VDC
Maximum output current	60mA

(2) 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.

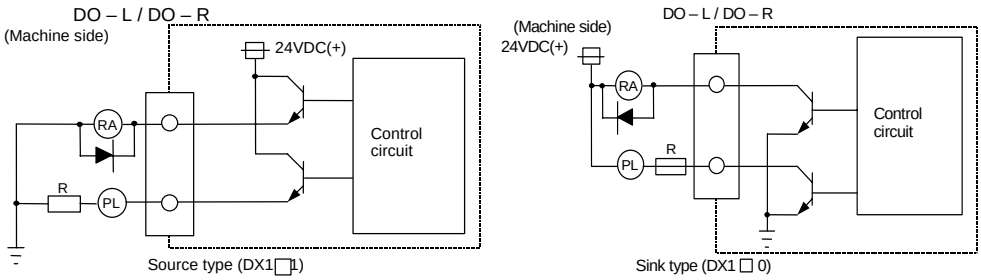
Input circuit



(3) Outline of digital signal output circuit

There is a sink type (DX1□0) and source type (DX1□1) digital signal output circuit. Use within the range of the specifications given below.

Output circuit



Output conditions

Insulation method	Non-insulated
Rated load voltage	+24VDC
Max. output current	60mA
Output delay time	40μs

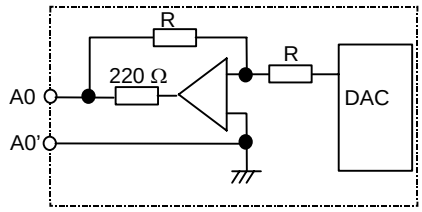
<Caution>

* When using an inductive load such as a relay, always connect a diode (withstand voltage 100V or more, 100mA or more) in parallel with the load. The diode should be inserted as close to the load (within 20cm) as possible.
* When using a capacitive load such as a lamp, connect a protective resistor (R=150) in serial with the load to limit the rush current. (Make sure that the current is lower than the above tolerable current, including momentary current.)

(4) Outline of analog signal output circuit

The analog signal output circuit can be used only with the FCUA-DX120/DX121.

Output circuit



Output conditions

Output voltage	0V~ ±10V (±5%)
Resolution	12bit (±10V×n/4095) (Note)
Load conditions	10 kΩ load resistance
Output impedance	220 Ω

(Note) n = (2^0 ~ 2^{11})

(5) Input signal conditions

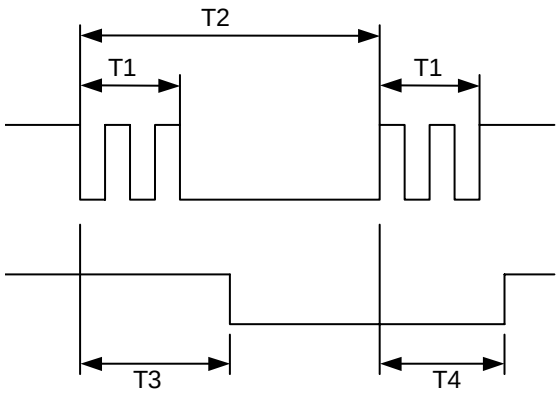
The input signals must be used within the ranges of the following conditions.

Source type <Contact common + 24V>

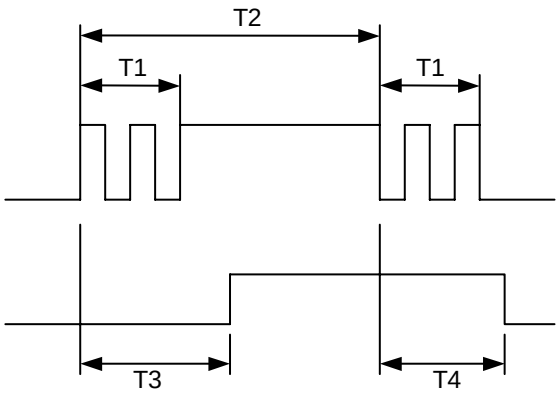
Input voltage when external contact is ON	18V or more, 25.2V or less
Input current when external contact is ON	9mA or more
Input voltage when external contact is OFF	4V or less
Input current when external contact is OFF	2mA or less
Tolerable chattering time	3ms or less (Refer to T_1 below)
Input signal hold time	40ms or more (Refer to T_2 below)
Input circuit operation delay time	3ms $\leqq T_3 \fallingdotseq T_4 \leqq$ 20ms
Machine side contact capacity	+30V or more, 16mA or more

Sink type <Contact common grounding (RG)>

Input voltage when external contact is ON	6V or more
Input current when external contact is ON	9mA or more
Input voltage when external contact is OFF	20V or less
Input current when external contact is OFF	2mA or less
Tolerable chattering time	3ms or less (Refer to T_1 below)
Input signal hold time	40ms or more (Refer to T_2 below)
Input circuit operation delay time	3ms $\leqq T_3 \fallingdotseq T_4 \leqq$ 20ms
Machine side contact capacity	DC30V or more, 16mA or more



Constantly closed contact



Constantly open contact

18.1.6 PLC Development

18.1.6.2 MELSEC Development Tool

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

The GX Developer installed in a personal computer (OS: Windows) can be used.

18.1.7 C Language Function

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

PLC subprograms prepared in C language can be called from PLC ladders.

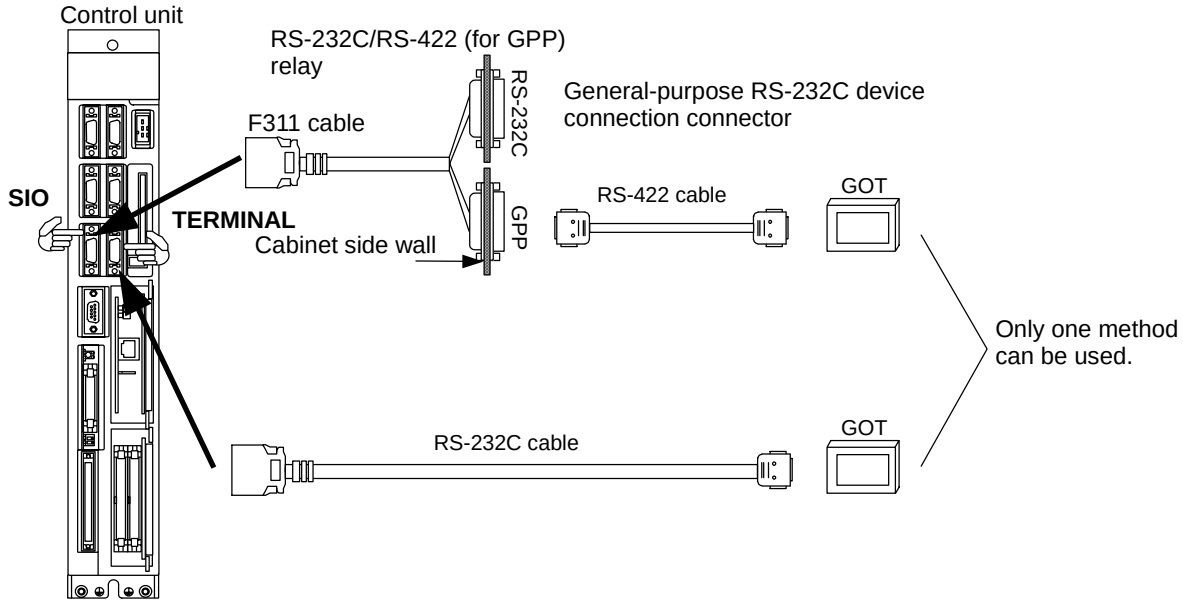
18.1.12 GOT Connection

This function connects a Mitsubishi graphic operation terminal (GOT) with the C6/C64 so it can be used as a machine operation panel, etc.
The information displayed on the GOT includes all of the PLC devices in the C6/C64, and the various monitor information. The C6/C64 dedicated setting and display screen and circuit monitor can also be displayed.
The following methods can be used to connect the C6/C64 and GOT.
A communication unit is required on each unit for either connection method. When using the CPU direct connection, an additional unit is not required on the C6/C64 side.

18.1.12.1 CPU Direct Connection (RS-422/RS-232C)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

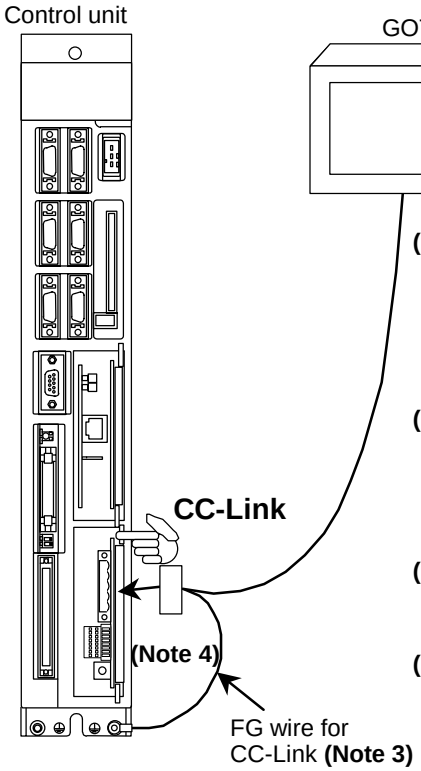
Connecting the C6/C64 and GOT with an RS-422 or RS-232C cable is the most cost efficient method.
When connecting with RS-422, the GOT is connected to the GPP connector side of the F311 cable connected to the SIO connector on the G64 control unit.
When connecting with RS-232C, the GOT is connected to the TERMINAL connector on the C64 control unit.



18.1.12.2 CC-Link Connection (Remote Device)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

C6/C64 functions as the CC-Link system's intelligent device station and remote device station, and can be remotely operated over a network.
To connect with CC-Link, the CC-Link unit (FCU6-HR865) must be mounted in the extension slot on the control unit.
Use a dedicated cable for the CC-Link cable, and connect to the CC-Link unit (FCU6-HR865) terminal block.
Always attach a resistor (enclosed) onto the unit which is the final station.



(Note 1) With the CC-Link system, the performance will not be guaranteed if a cable other than the CC-Link dedicated cable is used. Refer to the CC-Link Association web site (http://cc-link.org) for information on the CC-Link dedicated cable specifications. (Information is given in the section "Partner Association".

(Note 2) Always use the enclosed terminator.
The terminating resistance value differs according to the cable in use. The CC-Link dedicated cable is 110Ω, and the CC-Link dedicated high-performance cable is 130Ω.

(Note 3) Connect the FG wire from the FG terminal on the C64 control unit's CC-Link terminal block to the FG terminal at the bottom of the control unit.

(Note 4) For the C64 control unit's channel No. setting rotary switch and baud rate setting rotary switch, pull out the CC-Link unit from the control unit and set the switches.

Refer to section"18.6.4 CC-Link" for details on the CC-Link specifications for the MELDAS C6/C64.

Refer to the "GOT-A900 Series User's Manual (GT Works2 Version1/GT Designer2 Version 1 compatible connection section) and other related documents for details on GOT.

18.1.12.3 CC-Link Connection (Intelligent Terminal)

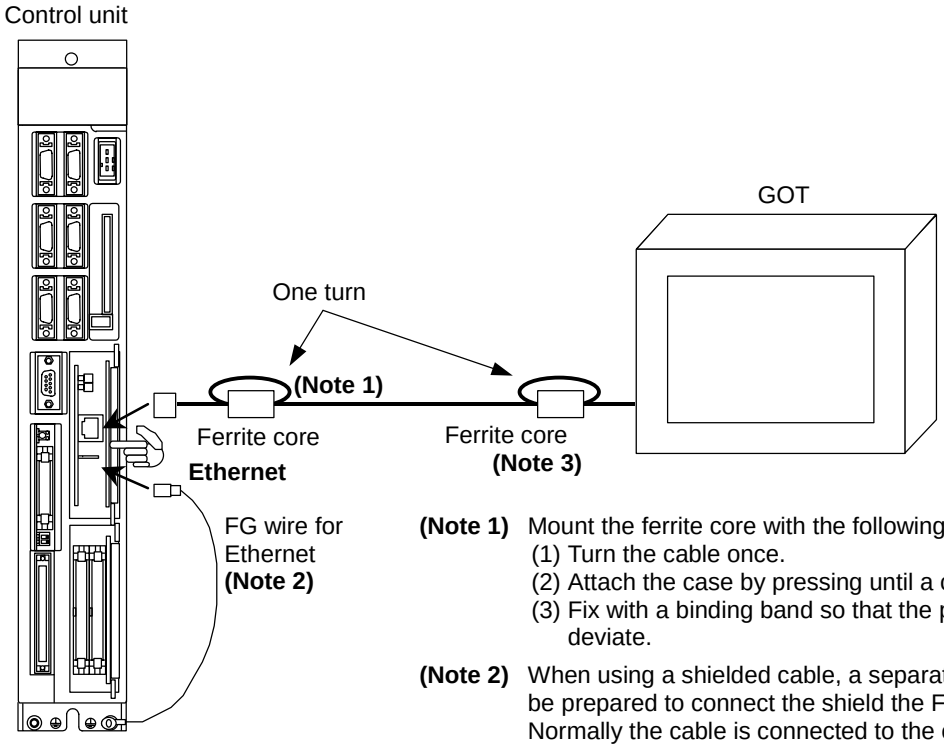
C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Refer to section "18.1.12.2 CC-Link Connection (Remote Device)" for details.

18.1.12.5 Ethernet Connection

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

When assembled in an Ethernet system, the C6/C64 can be remotely operated over a network.
To connect with Ethernet, the Ethernet module (FCU6-EX875) must be mounted in the extension slot on the control unit.
The Ethernet cable (10BASE-T cable) is connected to the Ethernet module's modular jack.
The Ethernet cable is easily affected by noise, so separate it from the drive and power cables, and mount the ferrite core (enclosed) on the control unit side.
Use of a shielded cable is recommended when using in a poor environment, or when compliance with EMC Directives is required.



(Note 1) Mount the ferrite core with the following procedures.
(1) Turn the cable once.
(2) Attach the case by pressing until a click is heard.
(3) Fix with a binding band so that the position does not deviate.

(Note 2) When using a shielded cable, a separate FG cable must be prepared to connect the shield the FG.
Normally the cable is connected to the control unit's FG terminal, but if the position is near the grounding plate, connect directly to that plate.

(Note 3) To comply with the EMC Directives, a ferrite core must also be mounted on the GOT side.

18.1.13 PLC Message

18.1.13.1 Japanese

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

18.1.13.2 English

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

18.1.13.13 Polish

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

18.2 Machine Construction

18.2.1 Servo OFF

C6		C64		
T system	L system	M system	L system	T 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.

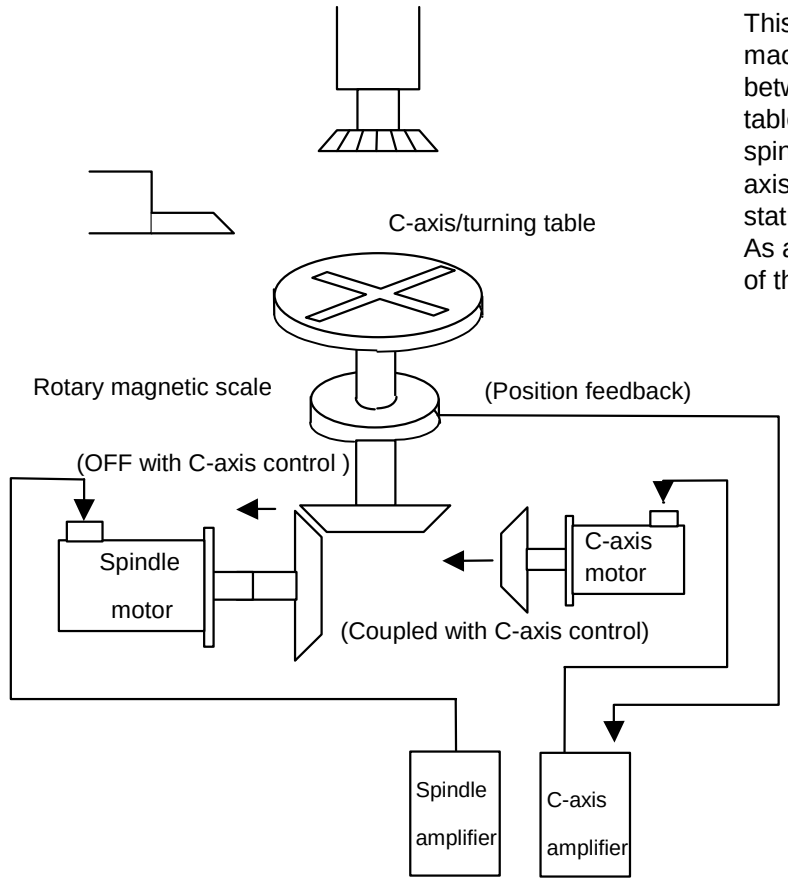
When follow up is designated, the movement amount will be compensated even in the emergency stop state.
The axis is simultaneously set with servo OFF to the interlock state.

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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

This function enables the control axis to be freed from control. Conversely, an axis which 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 axis detach command has been released completes its dog-type reference point 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   1 2 3 . 4 5 6
     Z     0 . 0 0 0 #1
     C   3 4 5 . 6 7 8 ><
```

The detached status > < is indicated on the right of the current position 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.

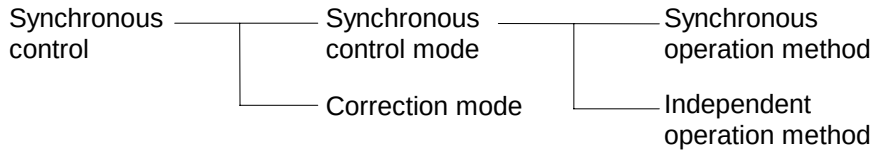
(Note) Axis detach can be executed even for the absolute position detection specifications axis, but when the axis is reinstalled, the zero point must be set.

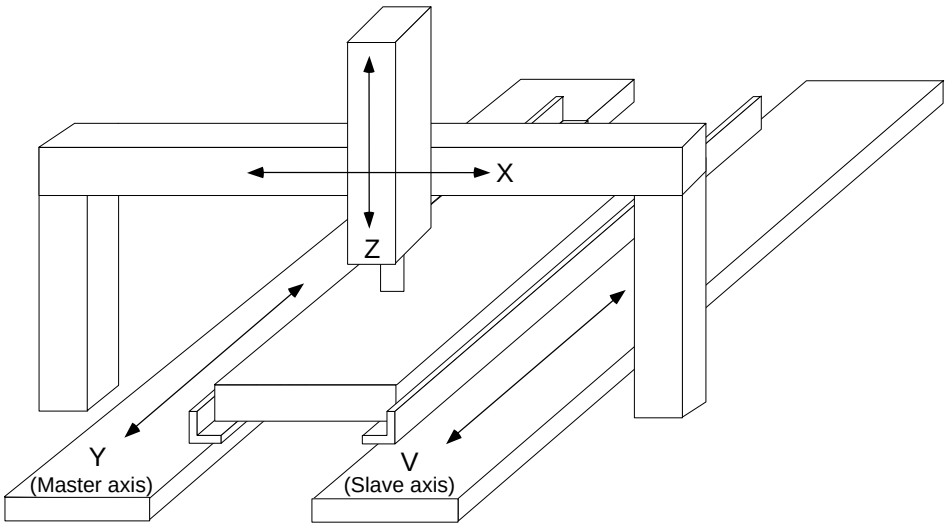
18.2.3 Synchronous Control

18.2.3.1 Position Tandem

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	Δ	Δ

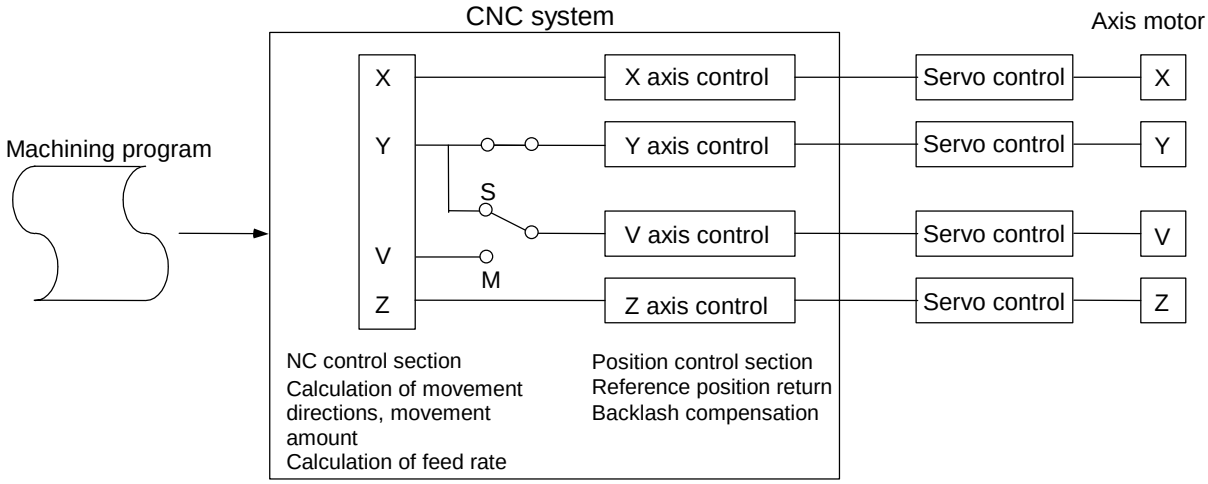
The synchronous control is a control method that both master and slave axes are controlled with the same movement command by designated the movement command for the master axis also to the slave axis. This function is assumed to be used in the large machine tool, etc. which drives one axis with two servo motors.
The axis for the base of the synchronization is called the master axis, and the axis according to the master axis is called the slave axis.
The axis detach function cannot be added to the axes used in the synchronous control.
• The slave axis is controlled with the movement command for the master axis.
• One slave axis can be set to one master axis.
• Two sets are applied for the master and slave axes





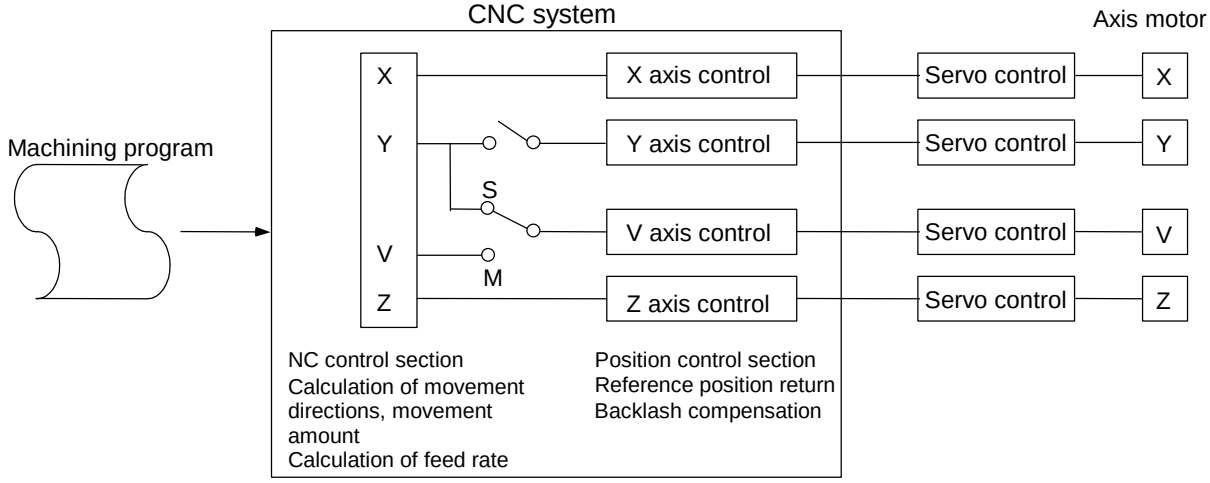
(1) Synchronous control mode
The following two operation methods are available in the synchronous control mode.

(a) Synchronous operation
This is a method that both master and slave axes are moved simultaneously with the movement command for the master axis.



There is a function that checks the correlation between the positions of the master axis and slave axis at all times while the synchronous operation method is selected to stop the feed as alarm when the allowable synchronization error value set in the parameter is exceeded. However, when the zero point is not established, the synchronous error is not checked.

(b) Independent operation
This is a method that either the master or slave axis is moved with the movement command for the master axis.



(2) Correction mode
The synchronization is temporary canceled to adjust the balance of the master and slave axes during the synchronous control mode in the machine adjustment. Each axis can be moved separately with the manual handle feed or the arbitrary feed in manual mode. If the operation mode other than the manual handle feed and arbitrary feed in manual mode is applied during the correction mode, the operation error will occur.

18.2.3.2 Speed Tandem

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	Δ	Δ

This function is used to drive in parallel while matching the position and speed.
In addition to the NC's synchronous control function, the master axis and slave axis speed command can be set to the same command by making the master axis and slave axis position feedback signal the same using the servo drive unit.
The speed command synchronization control cannot be used unless the NC setting and servo drive unit settings are changed.
The speed loop and current loop are controlled using the feedback signals for the respective axis.

18.2.3.3 Torque Tandem

C6		C64		
T system	L system	M system	L system	T system
Δ	–	Δ	Δ	Δ

This function is used to drive in parallel while matching the position, speed and current when the machine rigidity is high.
In addition to the NC's synchronous control function, the master axis and slave axis speed command can be set to the same command by making the master axis and slave axis position feedback signal and the speed feedback signal the same using the servo drive unit.
The current loop is controlled using the feedback signals for the respective axis.

18.2.7 Auxiliary Axis Control (J2-CT)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	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 point return mode
  In this mode, the axis is positioned at the coordinate reference point. Two methods are used: one method is based on a dog switch and the other method is to carry out positioning to the reference point which 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

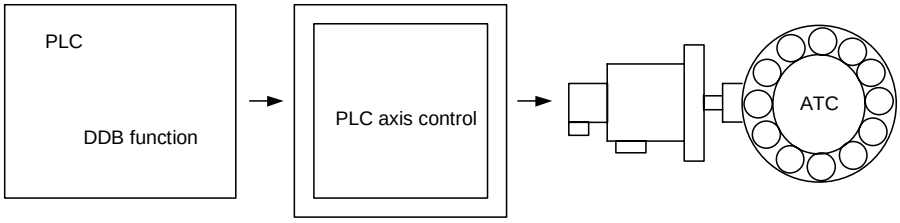
C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	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
Number of control axes	Max. 7 axes
Simultaneously controlled axes	PLC control axis is controlled independently from NC control axes. A multiple number of PLC axes can be started simultaneously.
Command increment	Least command increment 0.001mm (0.0001 inch) 0.0001mm (0.00001 inch) (Same as command increment for NC control axes)
Feed rate	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)
Operation modes	Rapid traverse, cutting feed, jog feed (+) (–), reference point return feed (+) (–), handle feed
Acceleration/deceleration	Rapid traverse, jog feed, reference point 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.)

18.4 PLC Interface

18.4.1 CNC Control Signal

C6		C64		
T system	L system	M system	L system	T 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 [T system, 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

C6		C64		
T system	L system	M system	L system	T 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 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

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

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 capability of referencing read data or write data in the next step just after a DDBA instruction is executed.

18.5 Machine Contact I/O

Standard DI/DO (DI:16/DO:1)

C6		C64		
T system	L system	M system	L system	T system
○	○	○	○	○

Operation board IO DI:32/DO:32

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Operation board IO DI:64/DO:48

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Remote IO 32/32

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Remote IO 64/48

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

Additional built-in DI/DO (DI:32/DO:32)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

18.6 External PLC Link

18.6.4 CC-Link

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

NC unit can be directly connected to the network to serve as the master/local station of the MELSEC CC-Link. To enable this connection, the CC-Link master/local units (HR865) must be installed in the expansion slots. Up to two communication units can be mounted.
Refer to the "MELSEC CC-Link System Master/Local Unit User's Manual" for details on CC-Link.

(1) Performance specifications

Item	CC-Link master/local unit (HR865)
Baud rates	156kbps/625kbps/2.5Mbps/5Mbps/10Mbps can be selected.
Max. transmission distance	The followings are obtained by the baud rate described above. 1200m/600m/200m/150m•110m/100m•80m•50m
Max. number of connection units	64 units Note that the following conditions must be satisfied. $\{(1 \times a)+(2 \times b)+(3 \times c)+(4 \times d)\} \leq 64$ a: Number of units that occupy station 1 b: Number of units that occupy station 2 c: Number of units that occupy station 3 d: Number of units that occupy station 4 $\{(16 \times A)+(54 \times B)+(88 \times C)\} \leq 2304$ A: Number of remote I/O stations $\leq$ 64 units B: Number of remote device stations $\leq$ 42 units C: Local station, Standby master station, Number of intelligent device stations $\leq$ 26 units
Number of occupied stations (Number of local stations)	Station 1 to station 4 (Changing over with DIP switch)
(Note 1) Max. number of link points per one system	Remote input/output (RX, RY) : Input/output each 2048 points Remote register (RWw) : 256 points (Master station → remote/local station) Remote register (RWw) : 256 points (Remote/local station → master station)
Number of link points per one remote station/local station	Remote input/output (RX, RY) : 32points (Local station is 30 points) Remote register (RWw) : 4 points (Mater station → remote/local station) Remote register (RWw) : 4 points (Remote/local station → master station)
Communication method	Polling method
Synchronization method	Flame synchronization method
Encode method	NRZI method
Transmission path method	Bus (RS485)
Transmission format	HDLC standard satisfied
Illegal control method	CRC ($X^{16} + X^{12} + X^5 + 1$)
Connection cable	Twist pair cable with shield
RAS function	• Automatic link refresh function • Sub-station isolation function • Link special relay/error detection by register
Number of Input/output occupied points	32 points

(Note 1) When assigning the CC-Link master station to the C64, the maximum number of remote input/output points may decrease depending on the number of device points that can be secured on the C64 side.

(2) Usable functions

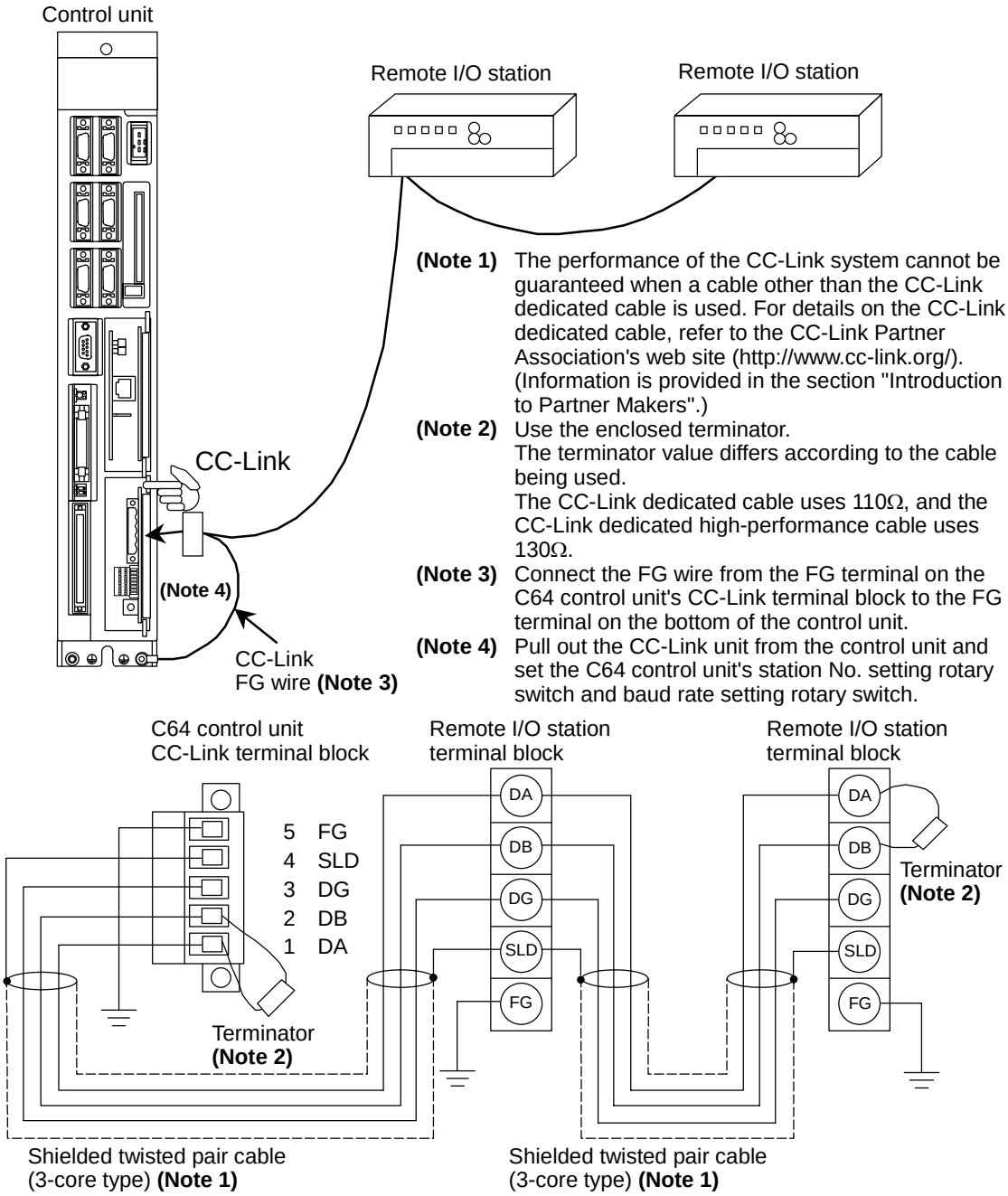
In the CC-Link functions, the ones listed in the table below can be used by the NC.

Function item			MELSEC	MELDAS C6/C64
Method	Ver.1		○	○
	Ver.2		○	×
Master function	Communication between master station and remote I/O station		○	○
	Communication between master station and remote device station		○	○
	Communication between master station and local station		○	○
	Mixed system communication		○	○
	Reserved station function		○	○
	Error cancel station function		○	○
	Setting of data link status when trouble occurs in CPU of master station		○	○
	Registration of parameters in EEPROM		○	○
	Setting of input data status from data link trouble station		○	○
	Unit resetting by sequence program		○	○
	Data link stop/restart		○	○
	Parameter registration function		○	○
	Automatic refresh function		○	○
	Scan synchronization function	Synchronous mode	○	○
		Asynchronous mode	○	○
	Local station		○	○
Setting and display functions	LED diagnosis status		16-point display (A1SJ61QBT11)	16-point display
	Station number setting		Unit front panel switches	Setting switches on card
	Baud rate setting			
	Mode setting switch			Card front panel switches
	Condition setting			
RAS functions	Automatic link refresh function		○	○
	Sub-station isolation function		○	○
	Data link status check (SB/SW)		○	○ To SB/SW Automatic refresh
	Off-line test		○	○
	On-line test		○	○
	Monitor diagnosis		○	×
	Standby master function		○	○
	Temporary error cancel station designation function		○	○
Dedicated commands	READ command/SREAD command		○	○
	WRITE command/SWRITE command		○	○
	RIRD command/RIWT command (Note 1)		○	○

(Note 1) Transient operation following these commands is applicable from software version D and following.

(3) Connection

The CC-Link unit (FCU6-HR865) must be mounted in the control unit's extension slot to connect IO devices using CC-Link.
Connect a dedicated CC-Link cable to the CC-Link unit (FCU6-HR865) terminal block.
Always install the enclosed terminator on the final station.
This unit functions as the CC-Link system's master and local station. Refer to the MELSEC A1SJ61QBT11 type CC-Link System Master/Local Unit's User Manual, etc., for details on the CC-Link system.



(Note 1) The performance of the CC-Link system cannot be guaranteed when a cable other than the CC-Link dedicated cable is used. For details on the CC-Link dedicated cable, refer to the CC-Link Partner Association's web site (http://www.cc-link.org/). (Information is provided in the section "Introduction to Partner Makers".)

(Note 2) Use the enclosed terminator.
The terminator value differs according to the cable being used.
The CC-Link dedicated cable uses 110Ω, and the CC-Link dedicated high-performance cable uses 130Ω.

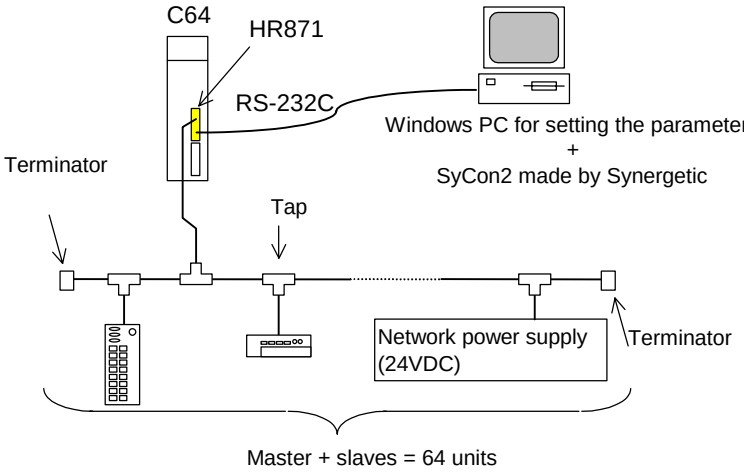
(Note 3) Connect the FG wire from the FG terminal on the C64 control unit's CC-Link terminal block to the FG terminal on the bottom of the control unit.

(Note 4) Pull out the CC-Link unit from the control unit and set the C64 control unit's station No. setting rotary switch and baud rate setting rotary switch.

18.6.6 DeviceNet (Master/Slave)

C6		C64		
T system	L system	M system	L system	T system
Δ master	Δ master	Δ master	Δ master	Δ master

This function is for connecting MELDAS C6/C64 with DeviceNet as the master station.
The HR871 dedicated interface card is required for this function.



Features

- DeviceNet complies with the revised version 2.0 of the written DeviceNet standards.
- C6/C64 operates as a Group2-only client of DeviceNet, and it communicates with the Group2-only server.
- I/O communication involves 256 bytes (2048 points) each for the input and output.

Restrictions

(1) The HR871 interface card enables C6/C64 to operate as the Group2-only client, but no communication is performed with other masters. In other words, communication with the configurator in the network is not supported, and dynamic establishment of connections is not supported either.
(2) The communication circuit board is made by Hilsher of Germany and, as such, when the network analyzer is installed, it will appear to be a Hilsher product (since Hilsher's vendor ID is recognized).
(3) The DeviceNet communication parameters must be set (configured) using either the configurator SyCon Ver.2.0 made by Synergetic and running in Windows or the PLC program.

18.6.7 MELSEC-Q Series Input/Output/Intelligent Function Unit Connection

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The MELSEC-Q Series input/output/intelligent function unit can be connected to the NC (MELDAS C6/C64).
Connections with the following specifications are possible when the Q bus bridge card HR863 is added. Only one Q bus bridge card can be mounted, and the extension space for up to two stages can be connected to the Q bus bridge card. There is a maximum of 24 slots (number of units).

Basic specifications for MELSEC I/O connection

Item	Basic specifications
Number of input/output points	Maximum input points: 512 points Maximum output points: 512 points
Access of intelligent unit's buffer memory	A maximum of 12k words can be accessed per scan of the intelligent unit's buffer memory using the FROM/TO commands issued from the C6/C64's built-in PLC.

Connectable MELSEC units
I/O unit

Part		Type	Outline
Input unit	AD	QX10	100 to 120VAC/7 to 8mA, 16 points, response time: 20ms, terminal block
		QX28	24VDC , 8 points, terminal block
	DC	QX40	240VDC/4mA, plus common, 16 points, response time: 1/5/10/20/70ms, terminal block
		QX40-S1	24VDC plus common input, 16 points, terminal block, for high-speed input (response time can be designated as 0.1ms)
		QX41	24VDC/4mA, plus common, 32 points, response time: 1/5/10/20/70ms, connector
		QX42	24VDC/4mA, plus common, 64 points, response time: 1/5/10/20/70ms, connector
		QX80	24VDC/4mA, minus common, 16 points, response time: 1/5/10/20/70ms, terminal block
		QX81	24VDC/4mA, minus common, 32 points, response time: 1/5/10/20/70ms, connector
Output unit	Contact	QY10	240VAC/24VDC, 2A/point, 8A/common, 16 points (16 points/common), output delay: 12ms, no fuse, terminal block
		QY18A	240VAC/24VDC, 2A, 8-point independent contact output, terminal block, no fuse
	AC Triac	QY22	240VAC/0.6A, 16 points, terminal block, no fuse
	Transistor	QY40P	12/24VDC, 0.1A/point, 1.6A/common, 16 points (16 points/common), output delay: 1ms, terminal block, with short-circuit protection function
		QY41P	12/24VDC, 0.1A/point, 2A/common, 32 points (32 points/common), output delay: 1ms, terminal block, with short-circuit protection function
		QY42P	12/24VDC, 0.1A/point, 2A/common, 64 points (32 points/common), output delay: 1ms, connector, with short-circuit protection function
		QY50	12/24VDC, 0.5A/point, 4A/common, 16 points (16 points/common), output delay: 1ms, with fuse, terminal block
	Transistor (sink)	QY68A	5-24VDC, 2A/point, 8A/unit, 8 points, all points independent, sink/source, terminal block, no fuse
	TTL CMOS (sink)	QY70	5/12VDC, 16mA/point, 16 points (16 points/common), output delay: 0.3ms, with fuse, terminal block
		QY71	5/12VDC, 16mA/point, 32 points (32 points/common), output delay: 0.3ms, with fuse, connector
	Transistor (source)	QY80	12/24VDC, 0.5A/point, 4A/common, 16 points (16 points/common), output delay: 1ms, with fuse, terminal block
		QY81P	12/24VDC, 0.1A/point, 2A/common, 32points (32points/common), output delay: 1ms, connector, with short-circuit protection function

Intelligent unit

Part	Type	Outline
FL-net (OPCN-2) unit	QJ71FL71-T-F01	
	QJ71FL71-B5-F01	
	QJ71FL71-B2-F01	
AS-i master unit	QJ71AS92	AS-i Standard Ver. 2.11 compatible master

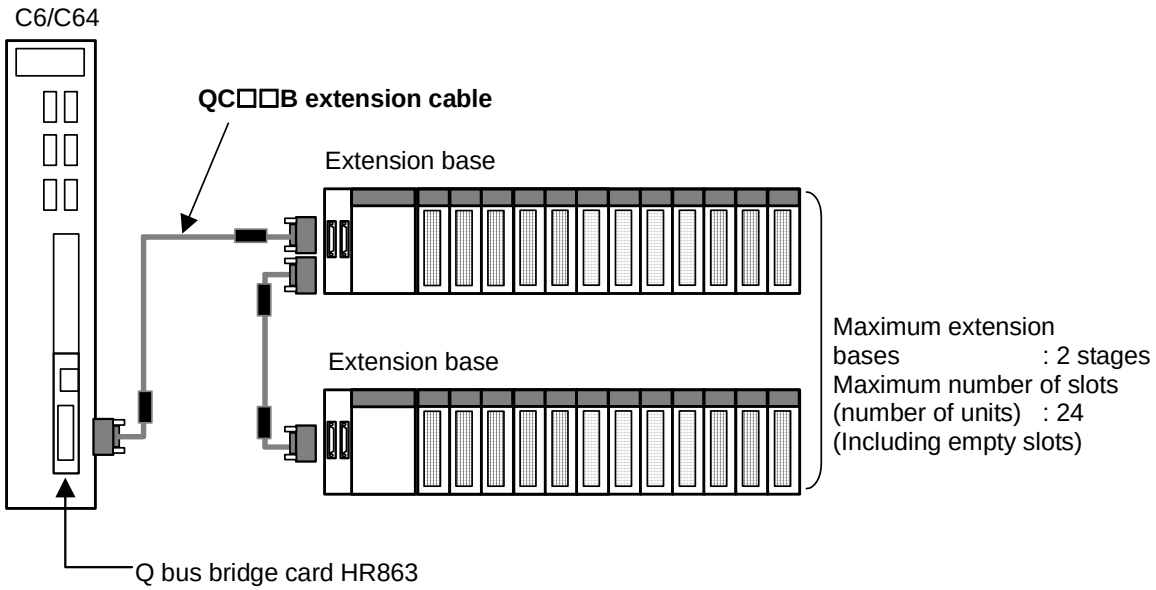
Others

Part	Type	Outline
Extension base	Q63B	Power supply + 3-I/O slots, for mounting Q Series units
	Q65B	Power supply + 5-I/O slots, for mounting Q Series units
	Q68B	Power supply + 8-I/O slots, for mounting Q Series units
	Q612B	Power supply + 12-I/O slots, for mounting Q Series units
Power supply unit	Q61P-A1	100-120VAC input/5VDC 6A output
	Q61P-A2	200-240VAC input/5VDC 6A output
	Q62P	100-240VAC input/5VDC 3A, 24VDC/0.6A output
	Q63P	24VDC input/5VDC 6A output
	Q64P	100-120/200-240VAC input, 5VDC 8.5A output

(Note 1) Up to two stages of extension bases can be connected.
(Note 2) The extension base with no power supply cannot be used.

The MELSEC units are connected in the following manner.

MELSEC unit connection



18.6.9 MELSECNET/10

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The coaxial bus type and optical loop type networks can be used between the controllers in the MELSECNET/10 data link system. When using the coaxial bus type, the FCU6-EX878 MELSECNET/10 unit must be mounted in the control unit's extension slot, and when using the optical loop type, the FCU6-EX879 MELSECNET/10 unit must be mounted.
This unit functions as the control station and normal station of the MELSECNET/10 data link system. Refer to the AJ71QLP21 (S1)/AJ71QBR11 type MELSECNET/10 Network Unit User's Manual (Hardware Section) for details on MELSECNET/10.

(1) Performance specifications

Item		Optical loop system (HR879)	Coaxial bus system (HR878)	
Maximum number of links per network	LX/LY	8192 points		
	LB	8192 points		
	LW	8192 points		
Maximum number of links per station		$\left\{\frac{B + Y}{8}\right\}$ + (2 × W) ≤ 2000 byte		
Maximum ring devices in NC	B	8192 points		
	W	8192 points		
Communication speed		10MBPS (equivalent to 20MBPS during multiplex transmission)	10MBPS	
Communication method		Token ring method	Token bus method	
Synchronization method		Frame synchronization		
Coding method		NRZI (Non Return to Zero Inverted)	Manchester coding	
Transmission path format		Double loop	Single bus	
Transmission format		HDLC compliant (frame type)		
Maximum number of networks		255		
Maximum number of groups		9		
Number of connected stations per network		64 stations (Control station 1, normal station: 63)	32 stations (Control station 1, normal station: 31)	
Overall distance per network		30km (500mm between stations)	3C-2V	5C-2V
			300m (300mm between stations)	500m (500mm between stations)
Error control method		Retry with CRC ($X^{16}+X^{12}+X^{5}+1$) and overtime		
RAS functions		Loop back at error detection and cable disconnection (only optical loop system) Diagnosis of local station's number of link check System down prevention with control station transfer Error detection with special relays and special registers, etc. Network monitor, various diagnosis functions		
Transient transmission		N:N communication (monitor, program upload, download, etc.) ZNRD/ZNWR (N:N)		
Connection cable		SI-200/250	3C-2V, 5C-2V or equivalent	
Applicable connector		2-core connector plug CA7003	BNC-P-3-Ni-CAU, BNC-P-5-Ni-CAU (DDK) or equivalent	
Cable transmission loss		12db/Km or less	JIS C 3501 compliant	

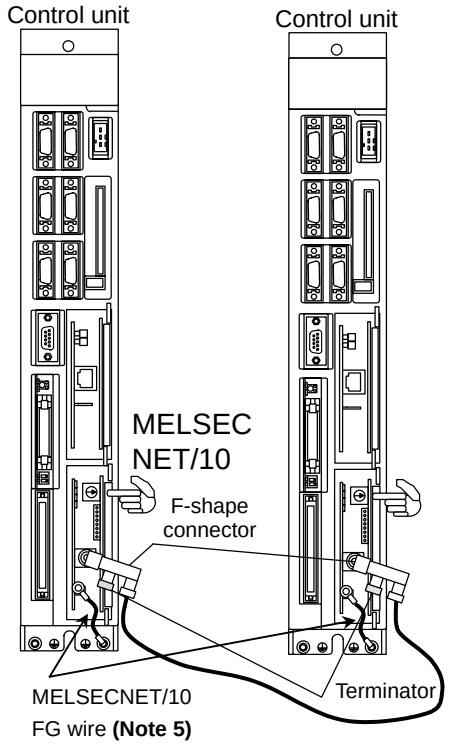
(2) Usable functions

The MELDAS C6/C64 can use the following MELSECNET/10 network functions.

Function item			MELSEC	MELDAS C6/C64
Network function	Control station function		○	○
	Control station transfer function		○	○
	Cyclic transmission	Communication with B/W (1:N)	○	○
		Communication with X/Y (1:1)	○	○
		Constant link scan function	○	○
		Data link stop/restart	○	○
		Transmission between data links	○	○
		Station parameters	○	×
	Transient transmission	N:N communication	○	○
		Routing function	○	○
		Group function	○	×
	RAS function	Automatic return function	○	○
		Loopback function	○	○
		Station cutoff function	○	○
		Data link status detection function	○	○
	Remote I/O network		○	×
	Multiple transmission function		○ (only optical loop system)	○ (only optical loop system)
	Reserved station function		○	○
Setting and display functions	LED diagnosis function		22-point display	4-point (coaxial) or 7-point (optical loop) display
	Network No. setting		Switch on front of unit	Setting switch on card
	Group No. setting			
	Station No. setting			
	Condition setting			
	Mode setting switch			Switch on front of card
	Display changeover switch			×
Self-diagnosis function	Hardware test		○	×
	Internal self-loopback test		○	×
	Self-loopback test		○	×
	Station-to-station test		○	×
	Main/sub-loop test		○ (only optical loop system)	×
	Loop test		○	×
	Setting switch confirmation		○	×
	Station check order		○	×
	Line monitor		○	×
	Status monitor		○	×
	Error history monitor		○	×
	Network test		○	×
Dedicated commands	READ/SREAD		○	○
	WRITE/SWRITE		○	○

(3) Connecting the coaxial bus type MELSECNET/10

Connect a dedicated coaxial cable to the MELSECNET/10 unit (FCU6-EX878) connector.
Use the enclosed F-shape connector, and always install the terminator A6RCON (optional) on the final unit.



(Note 1) Use a high-frequency coaxial cable 3C-2V or 5C-2V (compliant with JIS-C-3501).
The BNC-P-☐-Ni-CAU (DDK) is recommended.

(Note 2) Lay the coaxial cable at least 100mm away from the other drive lines and control cables.
When using in an adverse environment, or when compliance to EMC Directives is required, use a double shielded coaxial cable (Mitsubishi Wire 5C-2V-CCY, etc.). Connect the outer shield to the FG using the shield clamp fitting.

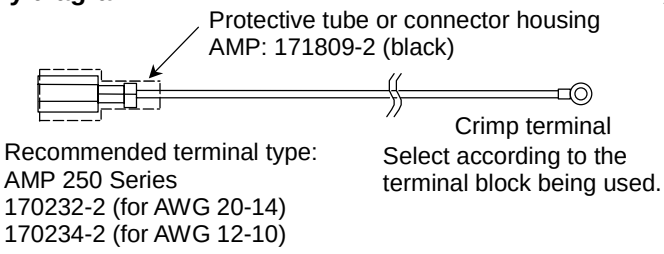
(Note 3) Use the following length of coaxial cable according to the total number of stations.

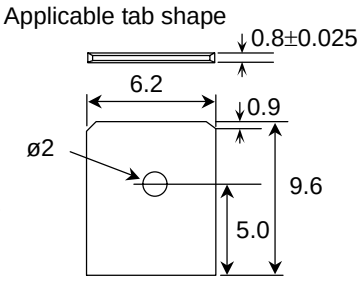
Total number of stations	Distance between stations
1 to 9 stations	1 to 500m
10 to 32 stations	1 to 5m 13 to 17m 25 to 500m

(Note 4) The BNC-TMP-05 (75) (Hirose Electric) terminator can be used instead of the A6RCON-R75 (optional).

(Note 5) Connect the FG wire from the FG terminal on the front of the MELSECNET/10 unit (FCU6-EX878) to the FG terminal on the bottom of the control unit.

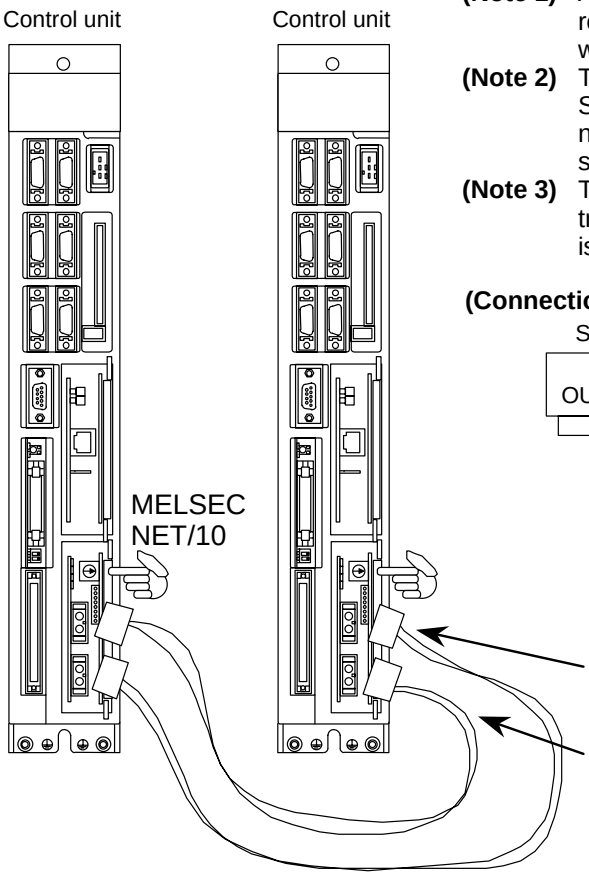
FG cable assembly diagram





(4) Connecting the optical loop type MELSECNET/10

Connect a dedicated optical fiber cable to the optical connector on the MELSECNET/10 unit (FCU6-EX879).

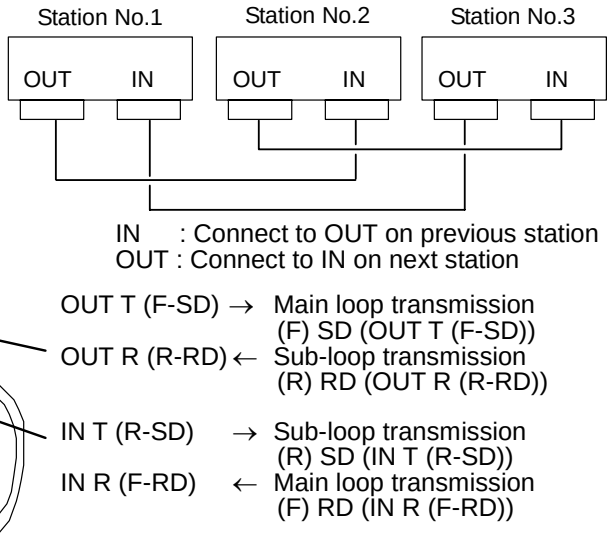


(Note 1) An indoor standard cable AS-2P-5M-A, etc., is recommended for the optical fiber cable. Consult with Mitsubishi Electric System Service.

(Note 2) The optical loop system's optical module follows SI specifications. The total distance within one network is 30km, and the distance between stations is 500m.

(Note 3) The optical loop system is a double loop transmission path method. The following system is used to connect the optical fiber cables.

(Connection example)

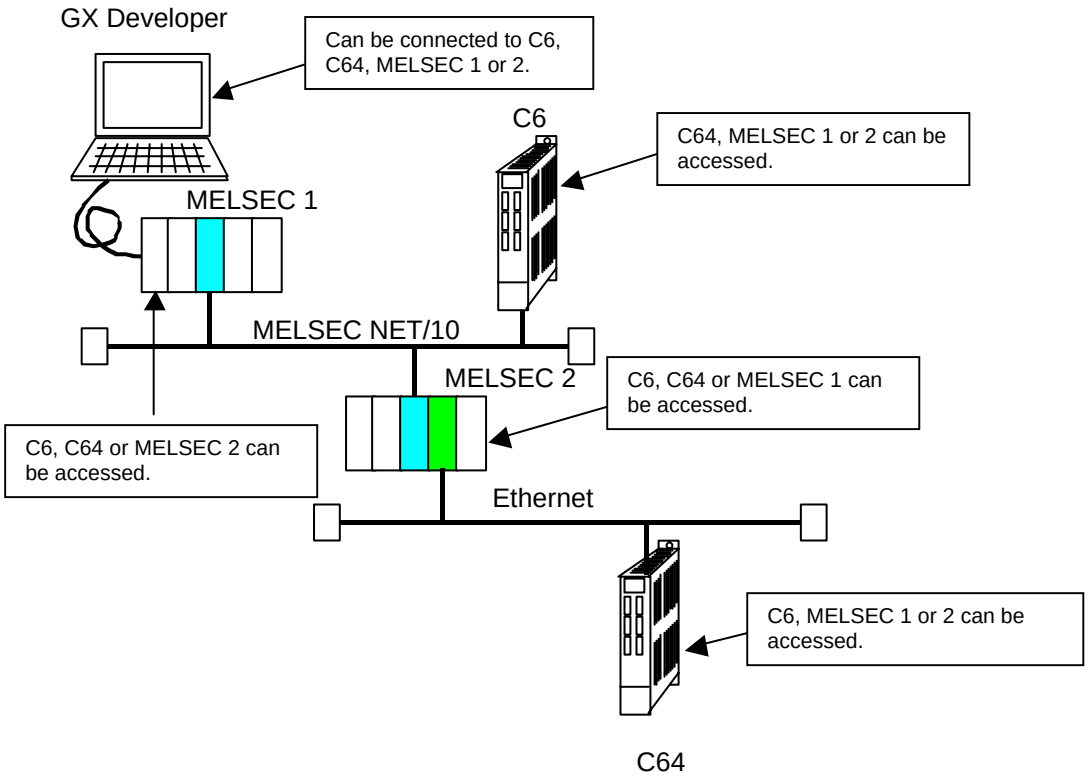


18.6.10 Ethernet I/F (MELSEC Communication Protocol)

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

MELSEC communication protocol (hereinafter, MC protocol) is the name of the MELSEC communication method used to read/write the data in the MELSEC CPU.
By using this protocol, the sequence programs and data in the C6/C64 can be accessed from an MELSEC peripheral device, etc., connected with Ethernet.
In this explanation, the C6/C64 and MELSEC CPU are collectively called the "PLC CPU".

On the PLC side, the Ethernet unit sends and receives data based on the instructions from the client device. Thus, a sequence program for exchanging data is not required on the PLC CPU side.



18.7 Installing S/W for Machine Tools

Software other than the built-in PLC can be installed in order to implement the machine tool builder's own functions (customized release). The customized release function consists of the following items.

(1) Screen release interface function : Change of CNC standard screen, preparation of inherent screen
(2) DDB interface function : Read/write CNC data
(3) Machine control interface function : Set/reset PLC device
(4) File release interface function : Preparation, modification, registration, etc. of user files using file system of CNC system

18.7.1 APLC

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

The screens are released by pressing the "F0" function key (nothing is displayed on the screen of the NC unit). This enables the machine tool builder to display its own screens from its customized software.
Using the APLC libraries, the customized software enables screen displays (characters, graphics), key loading, file read/write, NC unit internal information read/write, and exchanges of R register and other information with PLC ladders.
Customized software is described using C language and developed using a commercial compiler.

18.7.6 EZSocket I/F

C6		C64		
T system	L system	M system	L system	T system
Δ	Δ	Δ	Δ	Δ

This middleware makes it easy to develop applications having a Windows interface.
The various functions of the NC unit can be used from a Windows application using VC++ language, VB language and VBA macro language.

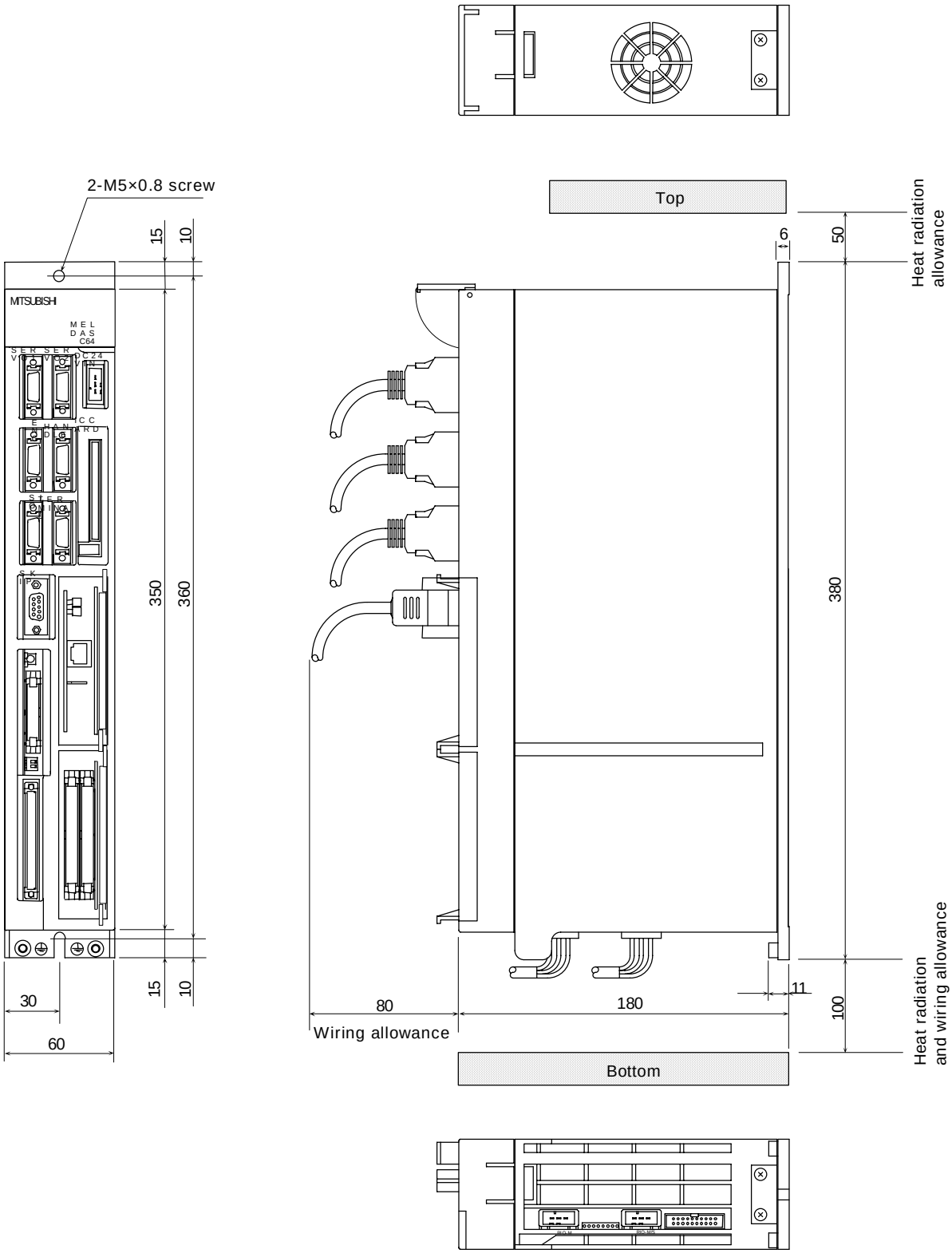
Appendix 1. List of Specifications

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.

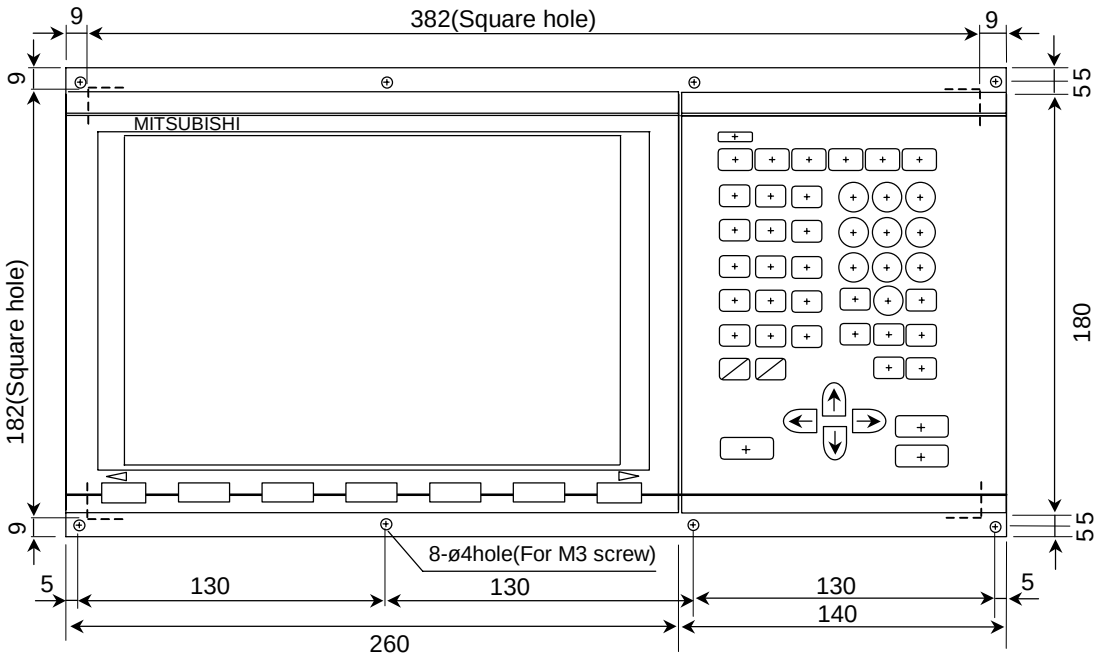
Appendix 2. Outline and Installation Dimension Drawings of Units

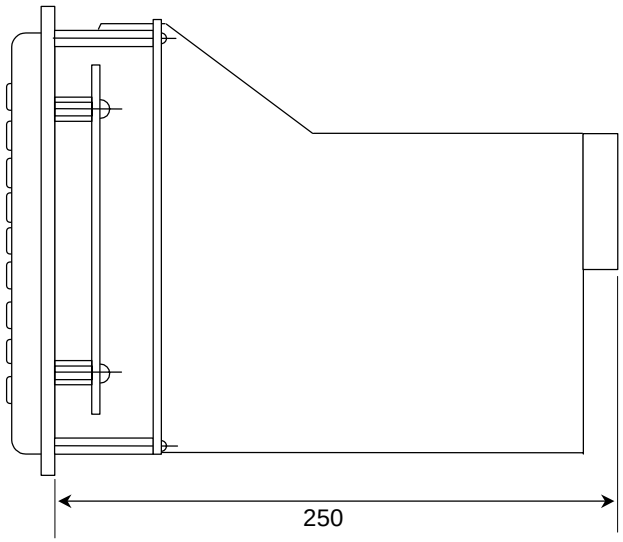
Appendix 2.1 Outline Drawing of Control Unit

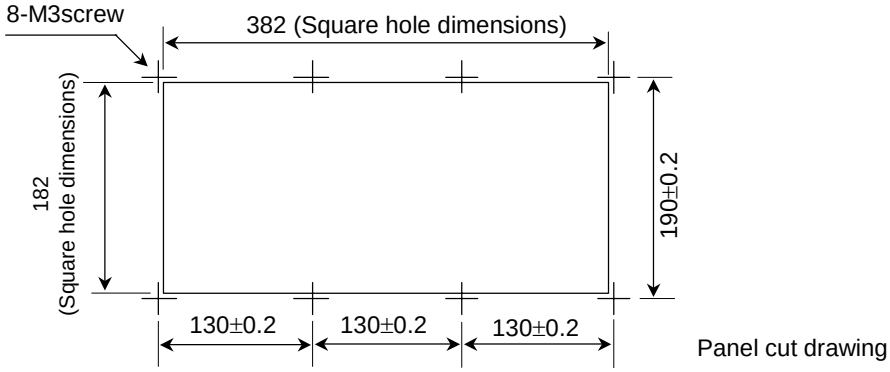


Appendix 2.2 Outline Drawing of Communication Terminal

Appendix 2.2.1 FCUA-CT100

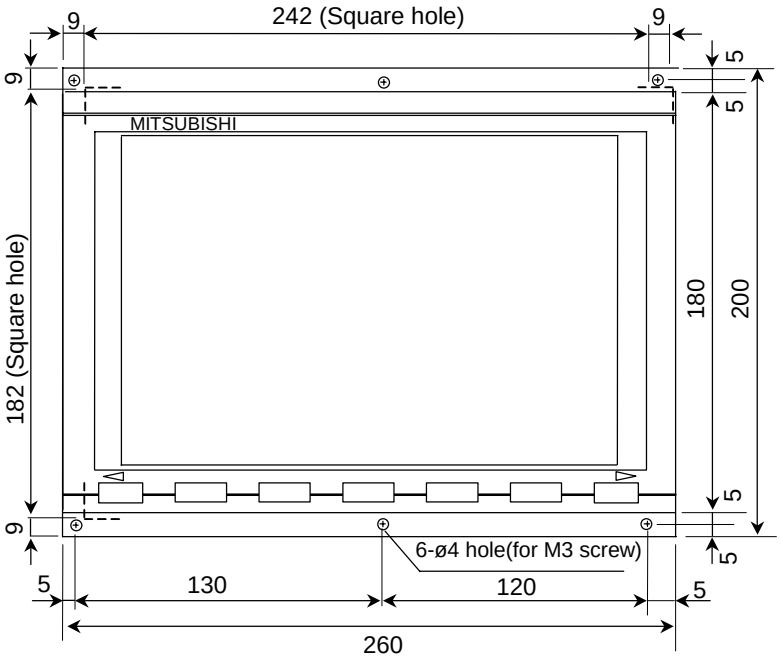


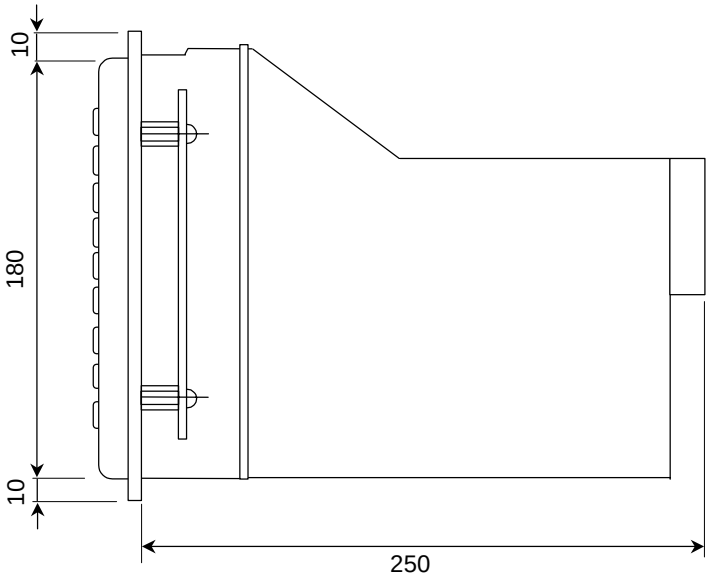


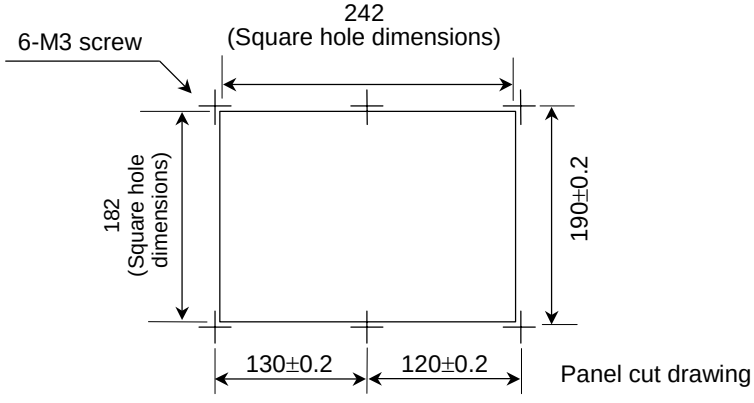


Panel cut drawing

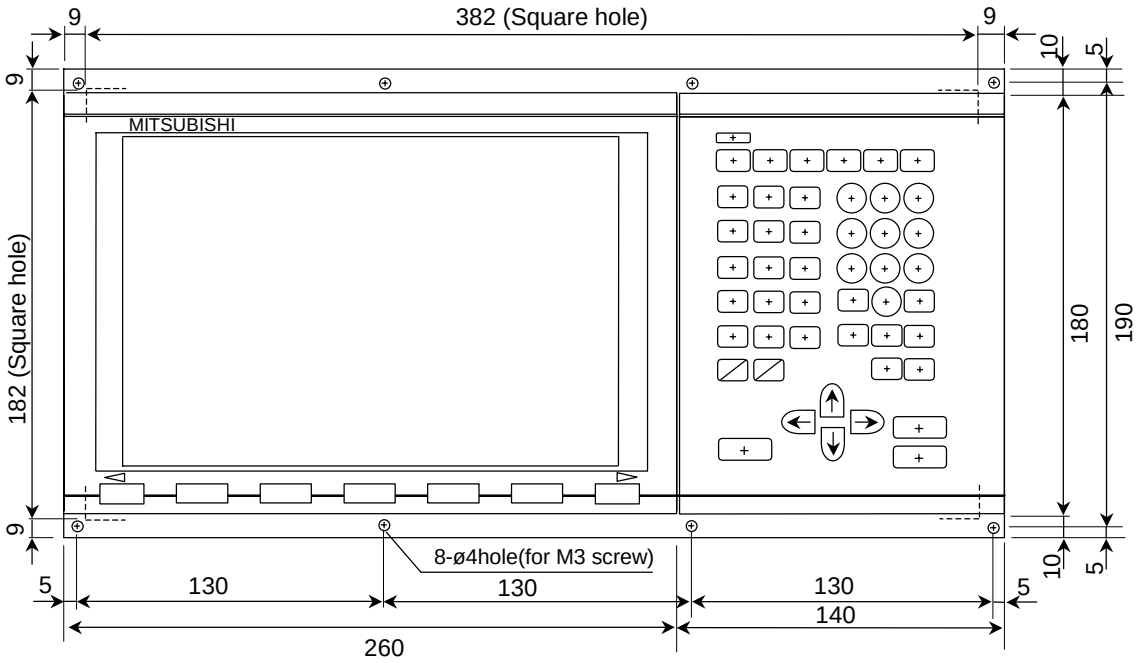
Appendix 2.2.2 FCUA-CR10

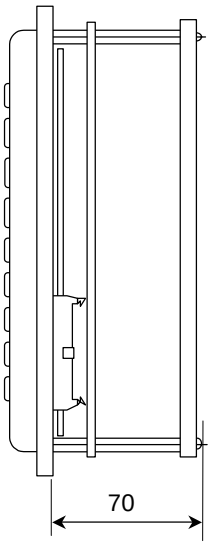


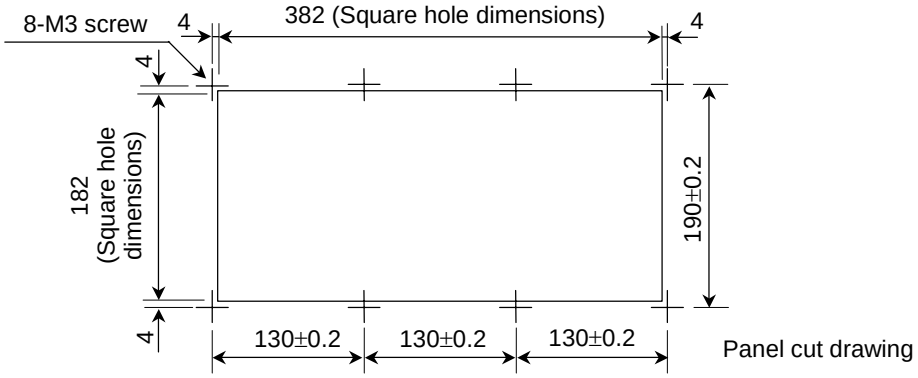




Appendix 2.2.3 FCUA-LD100

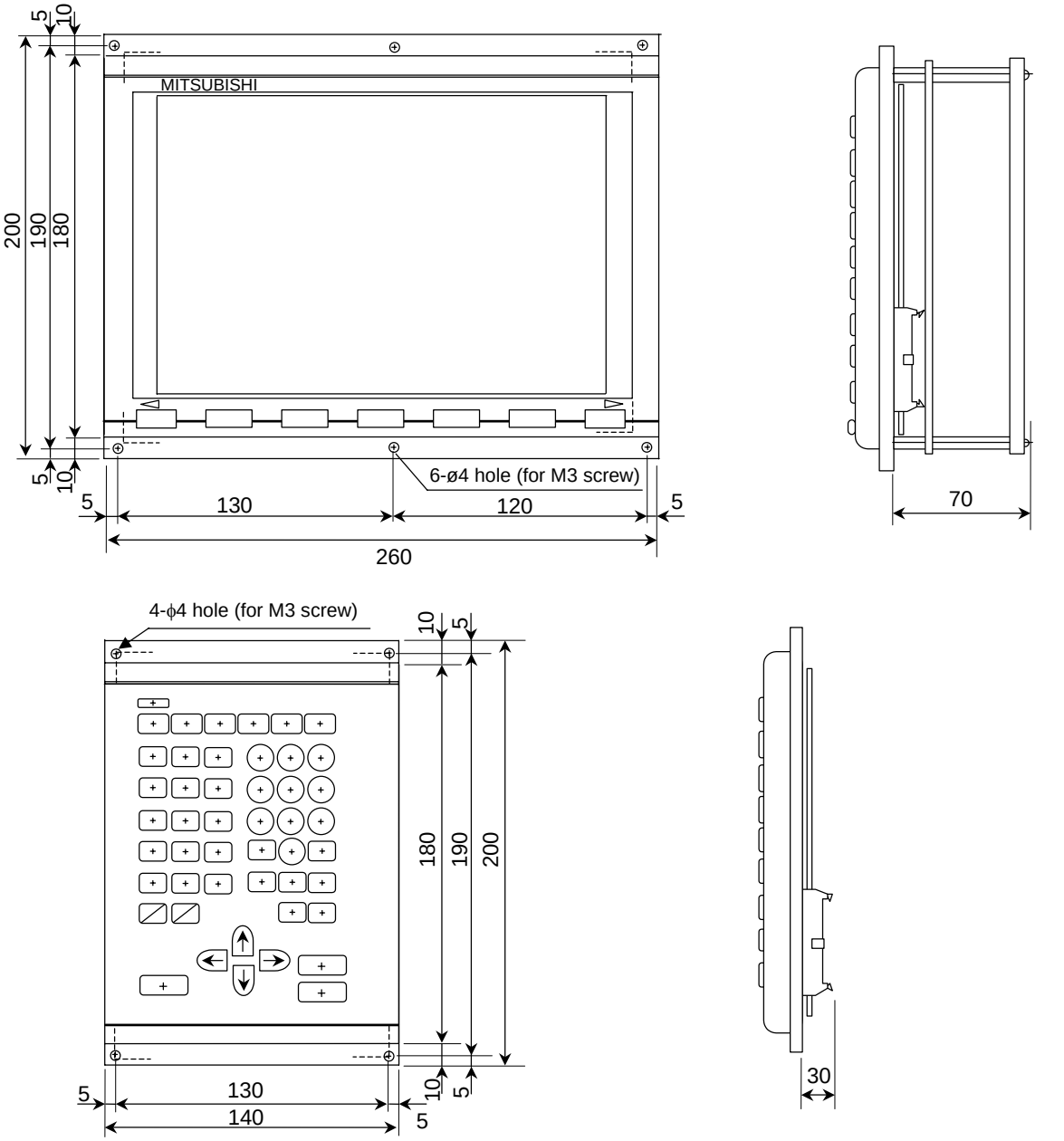






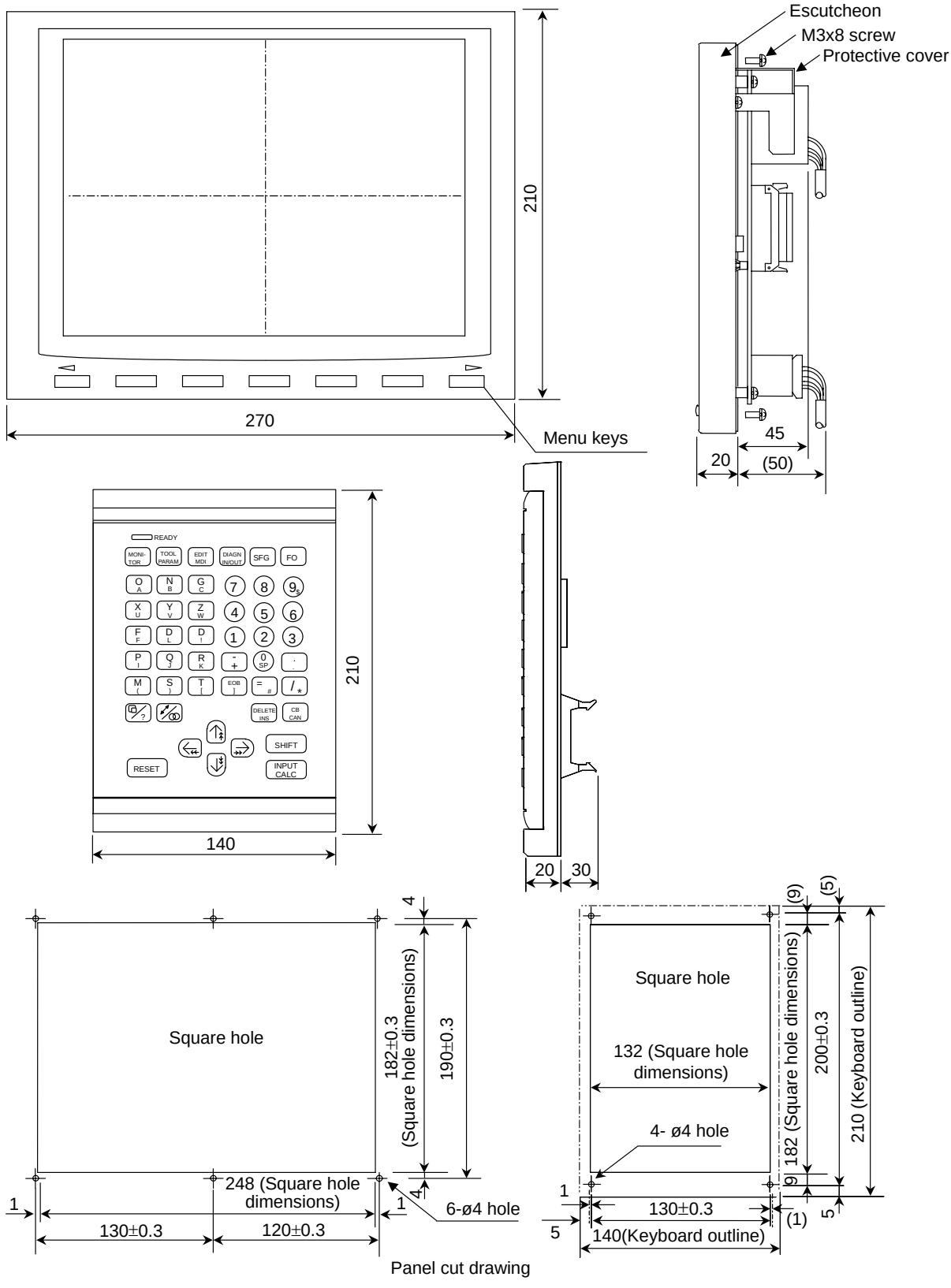
Panel cut drawing

Appendix 2.2.4 FCUA-LD10, KB20



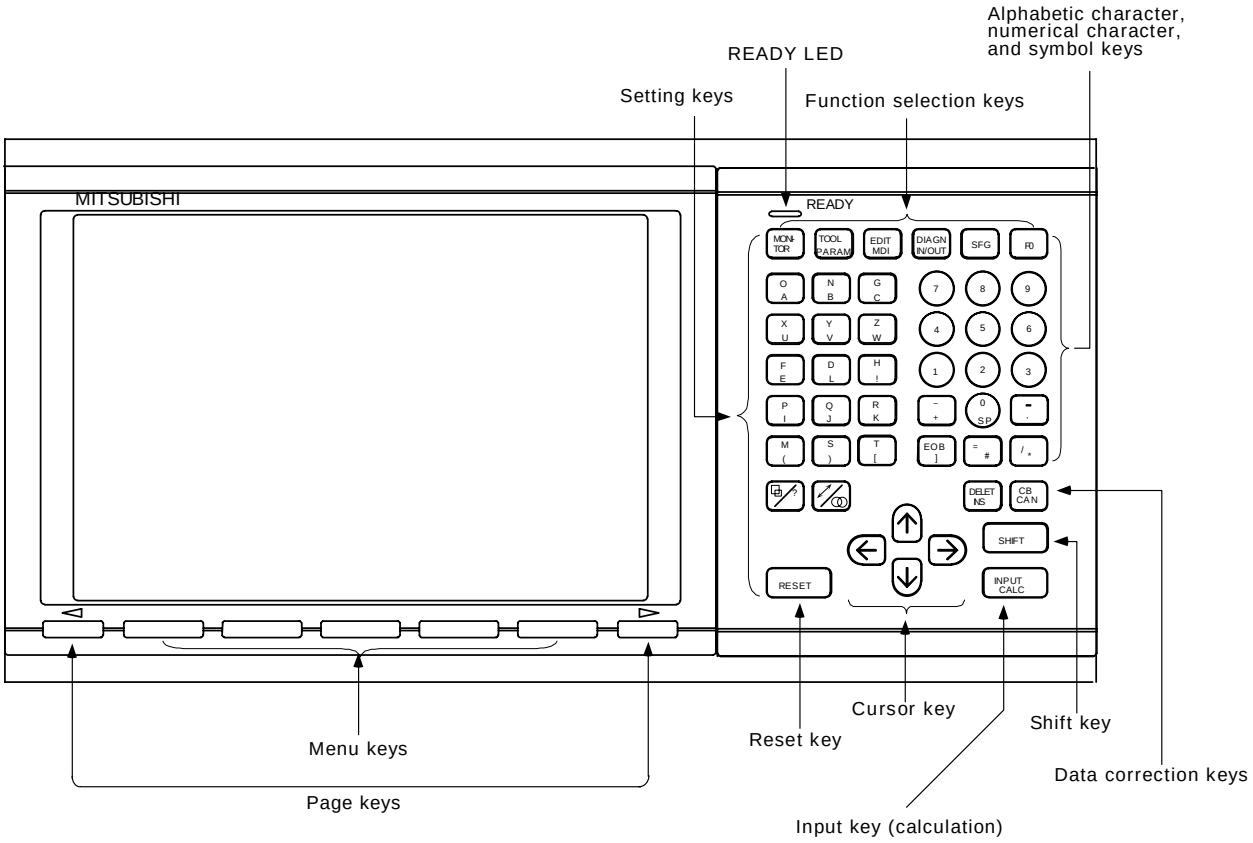
Panel cut drawing

Appendix 2.2.5 FCU6-DUT32, KB021



Appendix 2.2.6 Communication Terminal

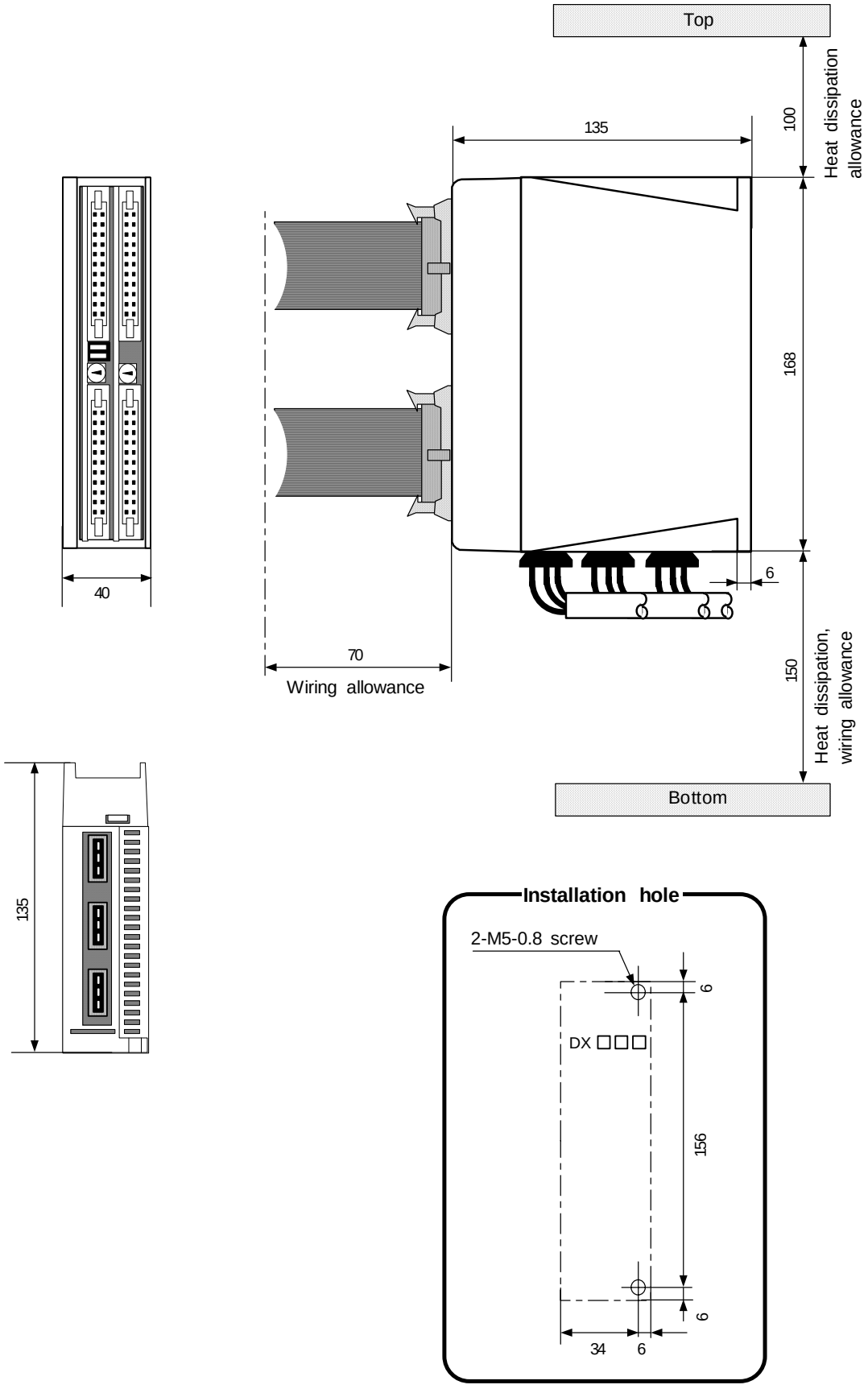
(1) Appearance of CT100/LD100/separate type FCUA-CR10 + KB10, FCUA-EL10 + KB10



(Note) To input the alphabetic characters or symbols on the lower of the alphabetic character and symbol keys, press [SHIFT] key, then press the corresponding key.

(Example) "A" is input by pressing [SHIFT], [O A] .

Appendix 2.3 Outline Drawing of Remote I/O Unit



Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class			C6		C64		
				for TRF	for FTL	for FTL		for TRF
				T system	L system	M system	L system	T system
1 Control axes								
	1 Control axes							
		1	Number of basic control axes (NC axes)	1	2	3	2	1
		2	Max. number of control axes (NC axes + Spindles + PLC axes + Auxiliary axes)	7	7	14	14	14
			Max. number of axes (NC axes + Spindles + PLC axes)	4	6	14	14	14
			Max. number of servo axes (NC axes + PLC axes)	2	4	14	14	14
			Max. number of NC axes (in total for all the part systems)	2	4	14	12	14
			Max. number of spindles	2 (1) (Note 1)	2 (1) (Note 1)	3	4	7 (1) (Note 1)
			Max. number of PLC axes	–	–	7	7	7
			Max. number of auxiliary axes (MR-J2-CT)	Δ5	Δ5	Δ7	Δ7	Δ7
		3	Number of simultaneous contouring control axes	2	2	4	4	2
		4	Max. number of NC axes in a part system	2	2	6	4	2
	2 Control part system							
		1	Standard number of part systems	1	1	1	1	1
		2	Max. number of part systems	Δ2	Δ2	Δ3	Δ3	Δ7
	3 Control axes and operation modes							
		2	Memory mode	○	○	○	○	○
		3	MDI mode	○	○	○	○	○
2 Input command								
	1 Data increment							
		1	Data increment and parameter					
		2	Least input increment					
		3	Least command increment					
			Least command increment 1μm	○	○	○	○	○
			Least command increment 0.1μm	Δ	Δ	Δ	Δ	Δ
		4	Least detection increment					
	2 Unit system							
		1	Inch/Metric changeover	Δ	Δ	Δ	Δ	Δ
	3 Program format							
		1	Character code	○	○	○	○	○
		2	Program format					
			1 Format 1 for Lathe (G code series 2, 3)	–	○	–	○	–
			4 Format 1 for Machining center (G code series 1)	○	–	○	–	○
	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	Δ	–	Δ	–	Δ
	2 Linear/Circular interpolation							
		1	Linear interpolation	○	○	○	○	○
		2	Circular interpolation (Center/Radius designation)	○	○	○	○	○
		3	Helical interpolation	–	–	Δ	–	–

(Note 1) Values in parentheses indicate the maximum number of spindles per part system.

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class				C6		C64		
					for TRF	for FTL	for FTL		for TRF
					T system	L system	M system	L system	T system
4 Feed									
	1 Feedrate								
		1	Rapid traverse rate (m/min)		1000	1000	1000	1000	1000
		2	Cutting feed rate (m/min)		1000	1000	1000	1000	1000
		3	Manual feed rate (m/min)		1000	1000	1000	1000	1000
	2 Feed rate input methods								
		1	Feed per minute		○	○	○	○	○
		2	Feed per revolution		Δ	Δ	Δ	Δ	Δ
		4	F 1-digit feed		○	○	○	○	○
	3 Overrite								
		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		○	○	○	○	○
				Linear acceleration/deceleration	○	○	○	○	○
				Soft acceleration/deceleration	○	○	○	○	○
				Exponential acceleration/deceleration	○	○	○	○	○
				Exponential acceleration/Linear deceleration	○	○	○	○	○
		2	Rapid traverse constant inclination acceleration/ deceleration		○	○	○	○	○
	5 Thread cutting								
		1	Thread cutting (Lead/Thread number designation)		Δ	○	Δ	○	Δ
		2	Variable lead thread cutting		–	○	–	○	–
		3	Synchronous tapping						
			1	Synchronous tapping cycle	Δ	Δ	Δ	Δ	Δ
		4	Chamfering		–	○	–	○	–
	6 Manual feed								
		1	Manual rapid traverse		○	○	○	○	○
		2	Jog feed		○	○	○	○	○
		3	Incremental feed		○	○	○	○	○
		4	Handle feed		Δ	Δ	Δ	Δ	Δ
	7 Dwell								
		1	Dwell (Time-based designation)		○	○	○	○	○
5 Program memory/editing									
		1	Memory capacity						
			1	Memory capacity (number of programs stored)					
				40m (64 programs)	○	○	○	○	○
				80m (128 programs)	Δ	Δ	Δ	Δ	Δ
				60m (200 programs)	Δ	Δ	Δ	Δ	Δ
				320m (200 programs)	Δ	Δ	Δ	Δ	Δ
				600m (400 programs)	Δ	Δ	Δ	Δ	Δ
		2	Editing method						
			1	Program editing	○	○	○	○	○
			2	Background editing	○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class				C6		C64		
					for TRF	for FTL	for FTL		for TRF
					T system	L system	M system	L system	T system
6 Operation and display									
	1 Structure of operation/display panel								
				7.2-type LCD monochrome display	□	□	□	□	□
				10.4-type LCD monochrome display	□	□	□	□	□
				9-type CRT monochrome display	□	□	□	□	□
				External PC display (connecting by Ethernet)	□	□	□	□	□
				Graphic operation terminal (GOT)	□	□	□	□	□
	2 Operation methods and functions								
		1		Memory switch (PLC switch)	Δ	Δ	Δ	Δ	Δ
	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 (Japanese/English)	○ 2 languages	○ 2 languages	○ 2 languages	○ 2 languages	○ 2 languages
		11		Additional languages (Japanese, English, Polish)					
			1	Japanese	○	○	○	○	○
			2	English	○	○	○	○	○
			13	Polish	○	○	○	○	○
		13		Screen deletion	○	○	○	○	○
	4 Display unit switch								
		1		Single-NC and multi-display unit switch	Δ	Δ	Δ	Δ	Δ
		2		Multi-NC and common-display unit	Δ	Δ	Δ	Δ	Δ
		4		Multi-NC and common-external PC display	Δ	Δ	Δ	Δ	Δ
		5		Display unit detachable	Δ	Δ	Δ	Δ	Δ
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	○	○	○	○	○
	2 Input/Output I/F								
		1		RS-232C I/F	○	○	○	○	○
		2		IC card I/F					
			1	I/F for IC card in control unit	○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class				C6		C64		
					for TRF	for FTL	for FTL		for TRF
					T system	L system	M system	L system	T system
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	Δ	Δ	Δ	Δ	Δ
			4	Coil change	○	○	○	○	○
			5	Automatic coil change	○	○	○	○	○
		2 Speed control							
			1	Constant surface speed control	–	Δ	–	Δ	–
			2	Spindle override	○	○	○	○	○
			3	Multiple-spindle control					
				1 Multiple-spindle control I	–	–	Δ	Δ	–
		3 Position control							
			1	Spindle orientation	○	○	○	○	○
			3	Spindle synchronization					
				1 Spindle synchronization I	–	–	Δ	Δ	–
				2 Spindle synchronization II	–	–	Δ	Δ	–
	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	○	○	○	○	○
			5	M code output during axis positioning	–	–	Δ	Δ	Δ
	4 2nd miscellaneous function (B)								
			1	2nd miscellaneous function	○	○	○	○	○
9 Tool compensation									
	1 Tool length/position offset								
			1	Tool length 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					
				2 40	○	–	○	–	○
				3 80	Δ	○	Δ	○	Δ
				4 100	Δ	–	Δ	–	Δ
				5 200	Δ	–	Δ	–	Δ
			2	Offset memory					
				1 Tool shape/wear offset amount	○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class			C6		C64		
				for TRF	for FTL	for FTL		for TRF
				T system	L system	M system	L system	T system
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	Δ	–	Δ	–	Δ
		7	Local coordinate system	○	○	○	○	○
		8	Coordinate system for rotary axis	○	○	○	○	○
		9	Plane selection	○	○	○	○	○
		10	Origin set	○	○	○	○	○
		11	Counter set	○	○	○	○	○
	2 Return							
		1	Manual reference point return	○	○	○	○	○
		2	Automatic 1st reference point return	○	○	○	○	○
		3	2nd, 3rd, 4th reference point return	○	○	○	○	○
		4	Reference point verification	○	○	○	○	○
		5	Absolute position detection	Δ	Δ	Δ	Δ	Δ
		6	Tool exchange 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	○	○	○	○	○
	3 Program search/start/stop							
		1	Program search	○	○	○	○	○
		2	Sequence number search	○	○	○	○	○
		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	–	Δ	–	Δ	–
		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 point retract	○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class			C6		C64		
				for TRF	for FTL	for FTL		for TRF
				T system	L system	M system	L system	T system
12 Program support functions								
	1 Machining method support functions							
		1 Program						
			1 Subprogram control	○8 layers	○8 layers	○8 layers	○8 layers	○8 layers
		2 Macro program						
			1 User macro	Δ4 layers	Δ4 layers	Δ4 layers	Δ4 layers	Δ4 layers
			3 Macro interruption	Δ	Δ	Δ	Δ	Δ
			4 Variable command					
			6 (50+50 × number of part systems) sets	Δ	Δ	Δ	Δ	Δ
			7 (100+100 × number of part systems) sets	Δ	Δ	Δ	Δ	Δ
			8 (200+100 × number of part systems) sets	Δ	Δ	Δ	Δ	Δ
		3 Fixed cycle						
			1 Fixed cycle for drilling	Δ	Δ	Δ	Δ	Δ
			2 Special fixed cycle	Δ	–	Δ	–	Δ
			3 Fixed cycle for turning machining	–	○	–	○	–
			4 Multiple repetitive fixed cycle for turning machining	–	○	–	○	–
		4 Mirror image						
			3 G code mirror image	Δ	–	Δ	–	Δ
			4 Mirror image for facing tool posts	–	Δ	–	Δ	–
		5 Coordinate system operation						
			1 Coordinate rotation by program	Δ	–	Δ	–	Δ
		6 Dimension input						
			1 Corner chamfering/Corner R	Δ	Δ	Δ	Δ	Δ
			3 Geometric command	–	○	–	○	–
		7 Axis control						
			5 Circular cutting	Δ	–	Δ	–	Δ
		8 Multi-part system control						
			1 Synchronization between part systems	Δ	Δ	Δ	Δ	Δ
			2 Start point designation synchronization	–	–	Δ	Δ	Δ
			6 Balance cut	–	–	–	○	–
			8 2-part system synchronous thread cutting	–	○	–	○	–
		9 Data input by program						
			1 Parameter input by program	Δ	Δ	Δ	Δ	Δ
			2 Compensation data input by program	Δ	Δ	Δ	Δ	Δ
		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 (G61.1)	Δ	–	Δ	–	Δ
	3 Programming support functions							
			2 Address check	○	○	○	○	○
13 Machine accuracy compensation								
	1 Static accuracy compensation							
			1 Backlash compensation	○	○	○	○	○
			2 Memory-type pitch error compensation	Δ	Δ	Δ	Δ	Δ
			3 Memory-type relative position error compensation	Δ	Δ	Δ	Δ	Δ
			4 External machine coordinate system compensation	Δ	Δ	Δ	Δ	Δ
			6 Ball screw thermal expansion compensation	Δ	Δ	Δ	Δ	Δ
	2 Dynamic accuracy compensation							
			1 Smooth high-gain control (SHG control)	○	○	○	○	○
			2 Dual feedback	○	○	○	○	○
			3 Lost motion compensation	○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class				C6		C64		
					for TRF	for FTL	for FTL		for TRF
					T system	L system	M system	L system	T system
14 Automation support functions									
	1 External data input								
		1	External search		Δ	Δ	Δ	Δ	Δ
		2	External workpiece coordinate offset		○	○	○	○	○
	2 Measurement								
		1	Skip						
			1	Skip	Δ	Δ	Δ	Δ	Δ
			2	Multiple-step skip	Δ	Δ	Δ	Δ	Δ
		5	Automatic tool length measurement		Δ	Δ	Δ	Δ	Δ
		6	Manual tool length measurement 1		Δ	Δ	Δ	Δ	Δ
	3 Monitoring								
		1	Tool life management						
			Tool life management II		Δ	Δ	Δ	Δ	Δ
		2	Number of tool life management sets						
			20/40/80 sets		–	Δ80	–	Δ80	–
			100/200 sets		Δ100	–	Δ100	–	Δ100
		3	Display of integrated time/number of parts		○	○	○	○	○
		4	Load meter		○	○	○	○	○
		5	Position switch		○16	○16	○16	○16	○16
	5 Others								
		1	Programmable current limitation		○	○	○	○	○
		4	Automatic restart		○	○	○	○	○
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	Δ	Δ	Δ	Δ	Δ
			3	Stored stroke limit IIB	Δ	Δ	Δ	Δ	Δ
			4	Stored stroke limit IC	Δ	Δ	Δ	Δ	Δ
		3	Stroke check before movement		○	○	○	○	○
		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		○Monitor	○Monitor	○Monitor	○Monitor	○Monitor
		3	Data sampling		○	○	○	○	○
		5	Machine operation history monitor		○	○	○	○	○
		6	NC data backup						
				RS-232C	○	○	○	○	○
				IC card	○	○	○	○	○
		7	PLC I/F diagnosis		○	○	○	○	○

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class			C6		C64		
				for TRF	for FTL	for FTL		for TRF
				T system	L system	M system	L system	T system
16 Cabinet and installation								
	1 Cabinet construction							
		1	Additional H/W I/F	2 slots	2 slots	2 slots	2 slots	2 slots
	2 Power supply							
		1	Power supply specification	24V	24V	24V	24V	24V
	3 Control power supply ON/OFF							
		1	Control power supply ON/OFF	○	○	○	○	○
	4 Environment							
		2	Temperature	○	○	○	○	○
		3	Humidity	○	○	○	○	○
		4	Vibration	○	○	○	○	○
		5	Ambient atmosphere	○	○	○	○	○
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)	□	□	□	□	□
		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)	□	□	□	□	□
		6	MDS-R-V1/R-V2 (200V Compact and small capacity)					
			Servo motor: HF□□-A51/E51 (1000kp/rev)	□	–	□	–	□
			Servo motor: HF□□-A42/E42 (100kp/rev)	□	□	□	□	□
			Servo motor: HF□□-A47 (100kp/rev)	□	□	□	□	□
	2 Spindle							
		1	MDS-C1-SP/C1-SPM/B-SP (200V)					
			Spindle motor: SJ/SJ-V	□	□	□	□	□
		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	Δ	Δ	Δ	Δ	Δ
		4	Power supply: MDS-A-CR (Resistance regeneration)	□	□	□	□	□

Appendix 3. List of Specifications

○ : Standard □ : Selection – : No specification
Δ : Optional ☆ : Special additional specifications

Primary class	Secondary class				C6		C64		
					for TRF	for FTL	for FTL		for TRF
					T system	L system	M system	L system	T system
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	Δ	Δ	Δ	Δ	Δ
		3	Built-in PLC capacity (Number of steps)		○32000	○32000	○32000	○32000	○32000
		4	Machine contact input/output I/F		○	○	○	○	○
		6	PLC development						
			2	MELSEC development tool	○	○	○	○	○
		7	C language function		Δ	Δ	Δ	Δ	Δ
		12	GOT connection						
			1	CPU direct connection (RS-422/RS-232C)	○	○	○	○	○
			2	CC-Link connection (Remote device)	Δ	Δ	Δ	Δ	Δ
			3	CC-Link connection (Intelligent terminal)	Δ	Δ	Δ	Δ	Δ
			5	Ethernet connection	Δ	Δ	Δ	Δ	Δ
		13	PLC message						
			1	Japanese	○	○	○	○	○
			2	English	○	○	○	○	○
			13	Polish	○	○	○	○	○
	2 Machine construction								
		1	Servo OFF		○	○	○	○	○
		2	Axis detach		Δ	Δ	Δ	Δ	Δ
		3	Synchronous control						
			1	Position tandem	Δ	–	Δ	Δ	Δ
			2	Speed tandem	Δ	–	Δ	Δ	Δ
			3	Torque tandem	Δ	–	Δ	Δ	Δ
		7	Auxiliary axis control (J2-CT)		Δ	Δ	Δ	Δ	Δ
	3 PLC operation								
		1	Arbitrary feed in manual mode		Δ	Δ	Δ	Δ	Δ
		3	PLC axis control		Δ	Δ	Δ	Δ	Δ
	4 PLC interface								
		1	CNC control signal		○	○	○	○	○
		2	CNC status signal		○	○	○	○	○
		5	DDB		Δ	Δ	Δ	Δ	Δ
	5 Machine contact I/O								
		Standard DI/DO (DI:16/DO:1)			○	○	○	○	○
		Operation board IO DI:32/DO:32			Δ	Δ	Δ	Δ	Δ
		Operation board IO DI:64/DO:48			Δ	Δ	Δ	Δ	Δ
		Remote IO 32/32			Δ	Δ	Δ	Δ	Δ
		Remote IO 64/48			Δ	Δ	Δ	Δ	Δ
		Additional built-in DI/DO (DI:32/DO:32)			Δ	Δ	Δ	Δ	Δ
	6 External PLC link								
		4	CC-Link		Δ	Δ	Δ	Δ	Δ
		6	DeviceNet (Master/Slave)		ΔMaster	ΔMaster	ΔMaster	ΔMaster	ΔMaster
		7	MELSEC Q series input/output/intelligent function unit connection		Δ	Δ	Δ	Δ	Δ
		9	MELSECNET/10		Δ	Δ	Δ	Δ	Δ
		10	Ethernet I/F (MELSEC communication protocol)		Δ	Δ	Δ	Δ	Δ
	7 Installing S/W for machine tools								
		1	APLC		Δ	Δ	Δ	Δ	Δ
		6	EZSocket I/F		Δ	Δ	Δ	Δ	Δ

Revision History

Date of revision	Manual No.	Revision details
Mar. 2002	BNP-B2266A	First edition created.
Jul. 2004	BNP-B2266C	• Due to changes in the List of Specifications (BNP-C3014-003), all items were generally reviewed, and order of listing was changed. • Details were revised to comply with software Version D. • Mistakes, etc., were corrected.

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 : MITSUBISHI DENKI BLDG., 2-2-3, MARUNOUCHI, CHIYODA-KU, TOKYO 100-8310, JAPAN

MODEL	MELDAS C6/C64
MODEL CODE	008-193
Manual No.	BNP-B2266C(ENG)

Specifications subject to change without notice.
(0407) MEE
Printed in Japan on recycled paper.