III Functional Specifications

1

Control Axes

1.1 Control Axes

The NC axis, spindle, PLC axis and auxiliary 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. X, Y, Z, U, V, W, A, B and C axis can be used.
The PLC axis is an axis that can be controlled from the PLC ladder.

1.1.1 Number of Basic Control Axes (NC Axes)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ 3	○ 3	○ 3	○ 3	○ 3	○ 3
L system	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2

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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	12	16	16	12	16	16
L system	12	16	16	12	16	16

A number of axes that are within the maximum number of axes, and that does not exceed the maximum number given for the NC axis, spindle and PLC axis can be used.
Connection specifications of NC axis, PLC axis and spindle
NC axes, PLC axes and spindles are connected to the optical servo channel (OPT).
Refer to the Connection manual for details.

1.1.2.1 Max. Number of NC Axes (In Total for All the Part Systems)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	8	16	16	8	16	16
L system	12	16	16	12	16	16

1.1.2.2 Max. Number of Spindles

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	4	4	4	4	4	4
L system	4	6	6	4	6	6

Includes analog spindles.

1.1.2.3 Max. Number of PLC Axes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	6	6	6	6	6	6
L system	6	6	6	6	6	6

1.1.3 Max. Number of Auxiliary Axes (MR-J2-CT)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	4	6	6	0	0	0
L system	4	6	6	0	0	0

Auxiliary axis: This can be connected to the channel (SV2) for J2-CT.

1.1.4 Max. Number of PLC Indexing Axes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	4	6	6	4	6	6
L system	4	6	6	4	6	6

PLC axis indexing function: This function is used to move the PLC axis to the positioning destination or an arbitrary coordinate position.

1.1.5 Number of Simultaneous Contouring Control Axes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	4	4	8	4	4	8
L system	4	4	8	4	4	8

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.

1.1.6 Max. Number of NC Axes in a Part System

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	6	8	8	6	8	8
L system	6	8	8	6	8	8

Listed are the maximum number of axes which can be controlled in a part system. Follow the specifications by each machine tool builder for actual use.

1.2 Control Part System

1.2.1 Standard Number of Part Systems

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	1	1	1	1	1	1
L system	1	1	1	1	1	1

1.2.2 Max. Number of Part Systems

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ 2	○ 2	○ 2	○ 2	○ 2	○ 2
L system	○ 2	○ 4	○ 4	○ 2	○ 4	○ 4

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

1.3 Control Axes and Operation Modes

1.3.1 Tape (RS-232C Input) Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

1.3.2 Memory Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

1.3.3 MDI Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The MDI data stored in the memory of the CNC 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 screen operations.

1.3.4 High-Speed Program Server Mode

1.3.4.1 CF Card in Control Unit

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	—	—	—
L system	△	△	△	—	—	—

The machining program stored in CF card can be operated by installing a CF card in the control unit CF (compact flash) card interface.
Machining programs can be copied to CF card with the front IC card or Ethernet on the input/output screen.

When a machining program stored in CF card is searched while "DS" is selected for device during operation search, the machining program in CF card can be operated as a main program. (The operation mode is "memory mode".)
Also, when "M198 Pp;" is commanded in the main program, the machining program in CF card can be called and operated as a sub program.

Macros such as WHILE, IF and GOTO can be used during high-speed program server mode, as well. Also, calling the sub program and macro program stored in memory or CF card is possible during high-speed program server mode operation.

1.3.5 Front IC Card Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The machining program stored in PCMCIA card can be operated by installing a PCMCIA card on the front of control unit.

When a machining program stored in PCMCIA card is searched while "Memory Card" is selected for device during operation search, the machining program in PCMCIA card can be operated as a main program. (The operation mode is "memory mode".)
Also, when "M98 Pp, Dd;" ("d" for designating a unit) is commanded in the main program, the machining program in PCMCIA card can be called and operated as a sub program.

Macros such as WHILE, IF and GOTO can be used during IC card operation, as well. Also, calling the sub program and macro program stored in memory or PCMCIA card is possible during IC card operation.

1.3.6 Hard Disk Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	—	—	—
L system	△	△	△	—	—	—

The machining program stored in the hard disk can be operated when using a high-resolution type display (a display with a hard disk mounted).

When a machining program stored in hard disk is searched while "HD" is selected for device during operation search, the machining program in the hard disk can be operated as a main program. (The operation mode is "memory mode".)
Also, when "M98 Pp ,Dd;" ("d" for designating a unit) is commanded in the main program, the machining program in the hard disk can be called and operated as a sub program.

Macros such as WHILE, IF and GOTO can be used during hard disk operation, as well. Also, calling the sub program and macro program stored in memory or the hard disk is possible during hard disk operation.

2

Input Command

2.1 Data Increment

2.1.1 Least Command Increment

The data increment handled in the controller includes the input setting increment and command increment. Each type is set with parameters.

(1) The input setting increment indicates the increment handled in the internal processing of the controller. The counter and tool compensation data, etc., input from the screen is handled with this increment. This increment is applied per part system (1st to 4th part system, PLC axis).

Input setting increment (parameter)	Metric unit system		Inch unit system	
	Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
1 μm (B)	0.001	0.001	0.0001	0.001
0.1 μm (C)	0.0001	0.0001	0.00001	0.0001
10nm (D)	0.00001	0.00001	0.000001	0.00001
1nm (E)	0.000001	0.000001	0.0000001	0.000001

(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.

Command increment (parameter)	Metric unit system		Inch unit system	
	Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
0	Accordance with #1003 iunit.			
1	0.0001	0.0001	0.00001	0.0001
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.

2.1.1.1 Least Command Increment 1 μm (Input Setting Increment 1 μm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

It is possible to command 0.001mm for the linear axis and 0.001° for the rotation axis.

2.1.1.2 Least Command Increment 0.1 μm (Input Setting Increment 0.1 μm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

It is possible to command 0.0001mm for the linear axis and 0.0001° for the rotation axis.

2.1.1.3 Least Command Increment 0.01 μm (10nm) (Input Setting Increment 10nm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	△	△	—	△	△

It is possible to command 0.00001mm for the linear axis and 0.00001°for the rotation axis.

2.1.1.4 Least Command Increment 0.001 μm (1nm) (Input Setting Increment 1nm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	△	△	—	△	△

It is possible to command 0.000001mm for the linear axis and 0.000001°for the rotation axis.

2.1.2 Least Control Increment

The least control increment includes 0.01 μm and 0.001 μm.
The least control increment determines the CNC's internal operation accuracy.

2.1.2.1 Least Control Increment 0.01 μm (10nm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

2.1.2.2 Least Control Increment 0.001 μm (1nm)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

2.1.3 Indexing Increment

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function limits the command value for the rotary axis.
This can be used for indexing the rotary table, etc. It is possible to cause a program error with a program command other than an indexing increment (parameter setting value).

(Example)When the indexing increment setting value is 2 degrees, only command with the 2-degree increment are possible.

G90 G01 C102.000 ; …Moves to the 102 degree angle.
G90 G01 C101.000 ; … Program error
G90 G01 C102 ; … Moves to the 102 degree angle. (Decimal point type II)

2.2 Unit System

2.2.1 Inch/Metric Changeover

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The unit systems of the data handled in the controller include the metric unit system and inch unit system.
The unit (inch/mm) for the setting and display, as well as for the handle/incremental feed can be switched with either the parameters or machining program (G20/G21 command).
An option is required when the unit is switched with the machining program command.

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

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

<table>
<tr><th colspan="2" rowspan="2">Parameter</th><th colspan="4">Data</th></tr>
<tr><th colspan="2">Machining program</th><th>Screen data
(Compensation amount, user parameter, counter, etc.)
/ Feedrate of handle, etc.</th><th>Machine parameter
/ PLC interface machine position, etc.</th></tr>
<tr><td rowspan="4">A</td><td rowspan="2">0</td><td>G20</td><td>Inch unit system</td><td rowspan="2">Metric unit system</td><td rowspan="4">Not affected</td></tr>
<tr><td>G21</td><td>Metric unit system</td></tr>
<tr><td rowspan="2">1</td><td>G20</td><td>Inch unit system</td><td rowspan="2">Inch unit system</td></tr>
<tr><td>G21</td><td>Metric unit system</td></tr>
<tr><td rowspan="2">B</td><td>0</td><td colspan="2" rowspan="2">Not affected</td><td rowspan="2">Not affected</td><td>Metric unit system</td></tr>
<tr><td>1</td><td>Inch unit system</td></tr>
</table>

(Note 1) The parameter changeover is valid after the power is turned ON again.
(Note 2) The unit system for the PLC axis can be switched with a parameter different from the one used with the NC axis.
The PLC axis unit system cannot be switched with the machining program (G20/G21 command).
(Note 3) When the power is turned ON or resetting is performed, the command increment depends on the parameter setting.

2.2.2 Input Command Increment Tenfold

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

The program's command increment can be multiplied by an arbitrary scale with the parameter designation.
This function is valid when a decimal point is not used for the command increment.

For example, this function allows a CNC unit, for which the command increment is set to 1 μ m, to run a machining program, which has been created with a 10 μ m input command increment, as same as before.
The scale is set with the parameters.

(Note 1) This function cannot be used for the dwell function G04_X_(P_);.
(Note 2) This function cannot be used for the compensation amount of the tool offset input.
(Note 3) This function can be used when decimal point type I is valid, but cannot be used when decimal point type II is valid.

2.3 Program Format

2.3.1 Program Format

This is G code (program) format.
The G-code of lathe system is selected by parameter.
This manual explains the G function with G-code list 3 as standard.

2.3.1.1 Format 1 for Lathe (G Code List 2, 3)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

2.3.1.2 Format 2 for Lathe (G Code List 4, 5)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

2.3.1.3 Special Format for Lathe (G Code List 6, 7)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

2.3.1.4 Format 1 for Machining Center

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

2.3.1.5 Format 2 for Machining Center (M2 Format)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

2.3.1.6 MITSUBISHI CNC Special Format

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

The formats of the turning fixed cycles (G77 to G79), multiple repetitive turning fixed cycles (G71 to G76) and drilling fixed cycles (G80 to G89) can be switched to the MITSUBISHI CNC special formats.

2.4 Command Value

2.4.1 Decimal Point Input I, II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(1) Decimal point input type I

When axis coordinates and other data are issued in machining program commands, the assignment of the program data can be simplified by using a decimal point. The minimum digit of a command not using a decimal point is the same as the least command increment.

The decimal point can be applied not only to axis coordinate position 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

As opposed to type I, the minimum digit of a command without a decimal point 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.	X100 μm, F20mm/min	X100mm, F20mm/min
G1 Y200 F100 (*1)	Y200 μm, F100mm/min	Y200mm, F100mm/min
G4 X1.5	Dwell 1.5s	<-
G4 X2	Dwell 2ms	Dwell 2s

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

2.4.2 Absolute/Incremental Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	O	O	O	O	O	O
L system	O	O	O	O	O	O

(1) M system

When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be selected.

The absolute and incremental commands can be both used in one block, and are switched with G90 or G91. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

G90 ... Absolute command (absolute command)

G91 ... Incremental command (incremental command)

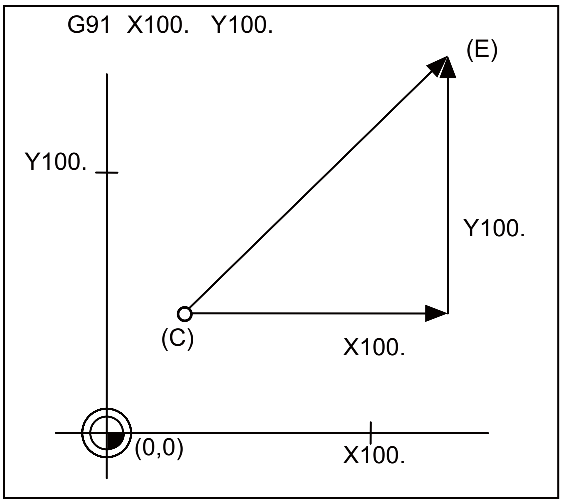
These G codes can be commanded multiple times in one block.

Example

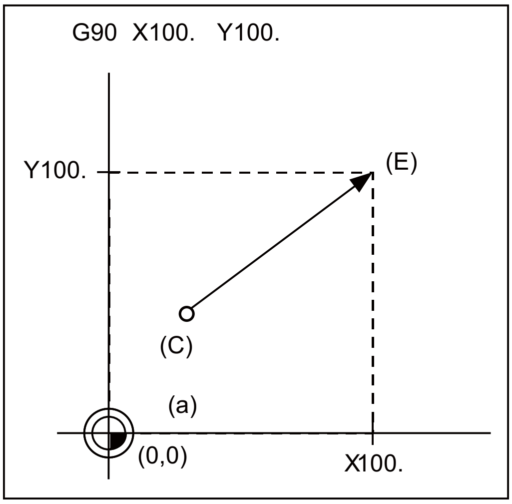
G90 X100.	G91 Y200.	G90 Z300.	;
Absolute position	Incremental position	Absolute position	

(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 command)



(Absolute command)



(a) Program coordinate
(C) Current position
(E) End point

(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 command, register the axis address to be commanded as the incremental axis name in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

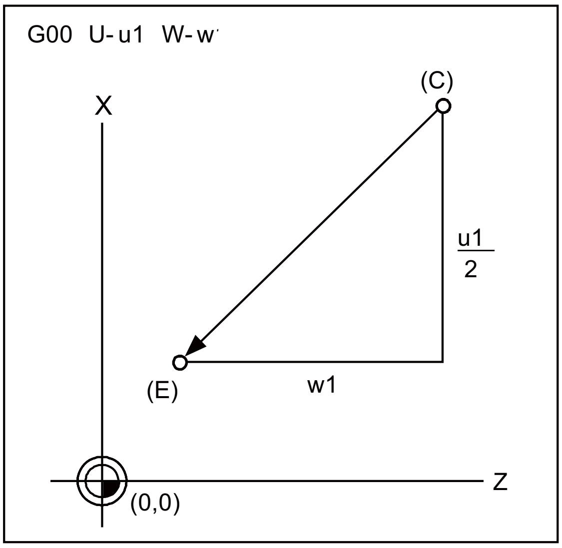
Absolute command (absolute command) ... X, Z

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

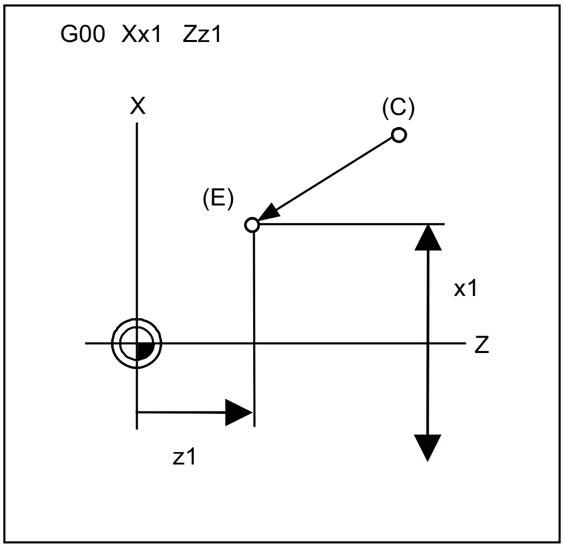
Example

G00 X100.	W200.	;
Absolute position	Incremental position	

(Incremental command)



(Absolute command)



(a) Program coordinate
(C) Current position
(E) End point

The above drawing shows the case for the diameter command.

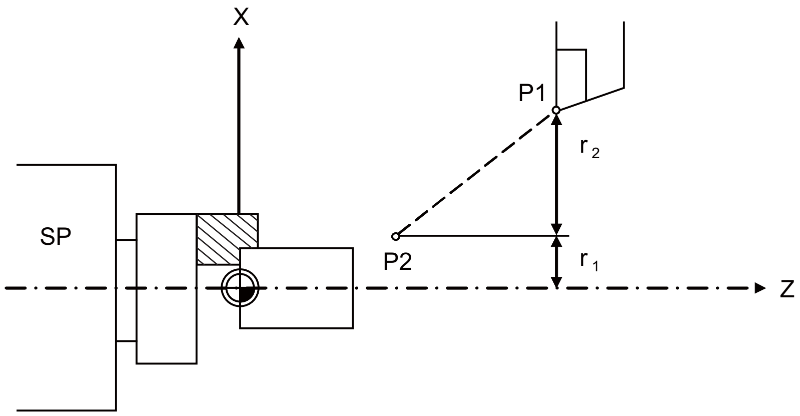
(Note) Absolute command and incremental command can be switched by the parameter. In addition to the command method using the axis addresses as indicated above, a command method using G code (G90/G91) may be selected.

2.4.3 Diameter/Radius Designation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L 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.
(Only the half (1/2) of the commanded amount moves.)
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.

When the tool is to be moved from point P1 to point P2



SP Spindle

Radius and diameter commands

X command		U command		Remarks
Radius	Diameter	Radius	Diameter	Even when a diameter command has been selected, only the U command can be made a radius command by parameter.
$X = r_1$	$X = 2r_1$	$U = r_2$	$U = 2r_2$	

3

Positioning/Interpolation

3.1 Positioning

3.1.1 Positioning

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

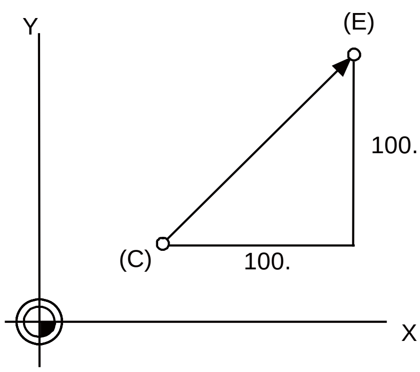
This function carries out high-speed positioning following the movement command given in a program.

G00 Xx1 Yy1 Zz1 ;(Also possible for additional axes A, B, C, U, V, W simultaneously)	
Xx1, Yy1, Zz1	:Position data

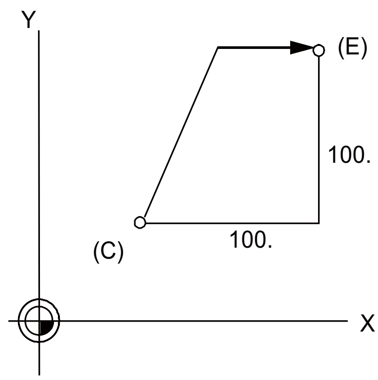
The above command positions the tool with rapid traverse rate. The tool path takes the shortest distance to the end point in the form of a straight line.
For details on the rapid traverse feed rate of the NC, refer to the section entitled "Rapid Traverse Rate".
Since the actual rapid traverse feed rate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

(1) The rapid traverse feed rate can be set for each axis with parameters.
(2) The number of axes which can be commanded 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 linearly to the end point.)
G00 G91 X100. Y100. ;



(Example) Non-linear type
(Each axis moves at each parameter speed.)
G00 G91 X100. Y100. ;



(C) Current position
(E) End point

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

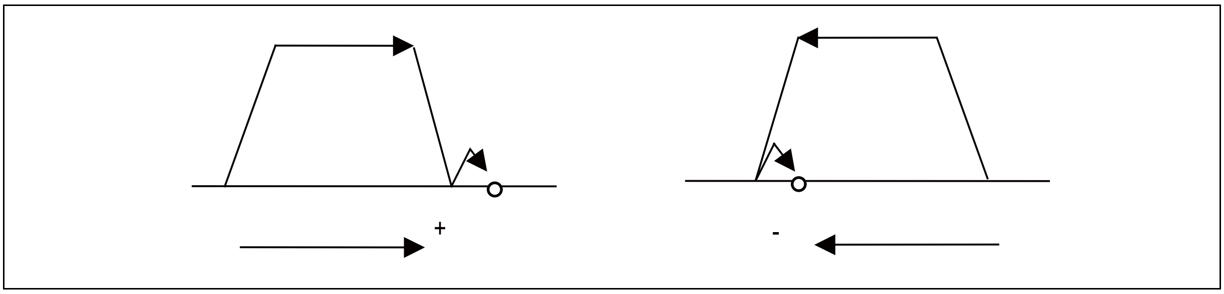
3.1.2 Unidirectional Positioning

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

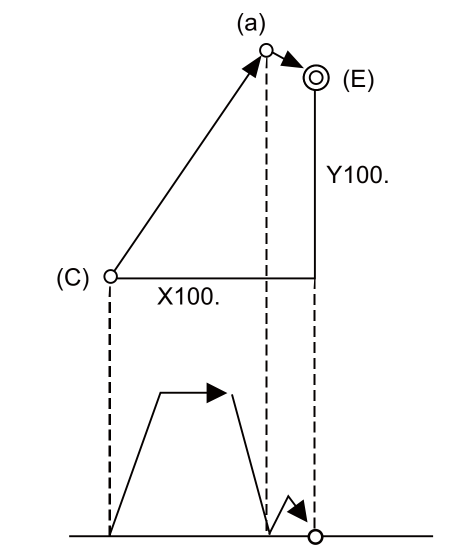
The G60 command always moves the tool to the final position in a direction determined with parameters. The tool can be positioned without backlash.

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

With the above command, the tool is first moved to a position distanced from the end point by an amount equivalent to the creep distance (parameter setting) with rapid traverse 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. ;



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

(a) Interim point
(C) Current position
(E) End point

(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

3.2.1 Linear Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

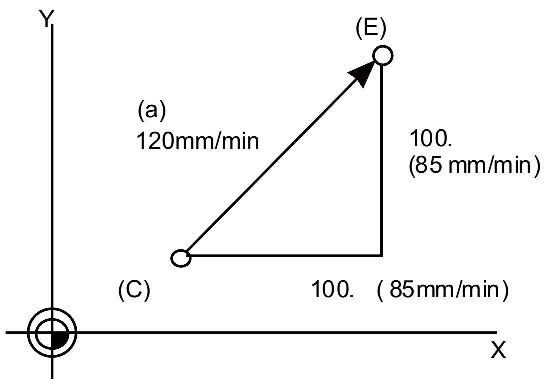
Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code.

G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)	
Xx1,Yy1,z1	:Position data
Ff1	: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 ;



The cutting feed rate command moves the tool in the vector direction.
The component speeds of each axis are determined by the proportion of respective command values.

(a) Feed rate
(C) Current position
(E) End point

(1) The number of axes which can be commanded 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)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

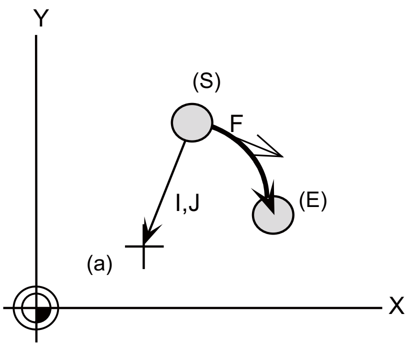
This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command supplied in the program.

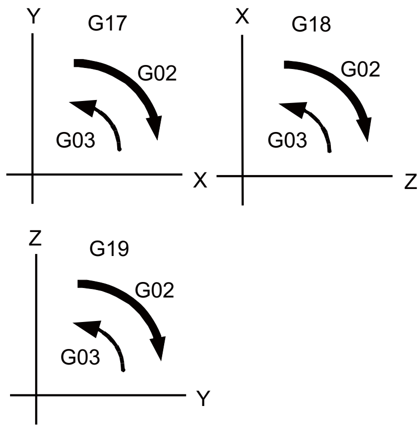
G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)

G02,G03	: Arc rotation direction
Xx1,Yy1	: End point coordinate
Ii1,Jj1	: Arc center
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) Center
(E) End point
(S) Start point

(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

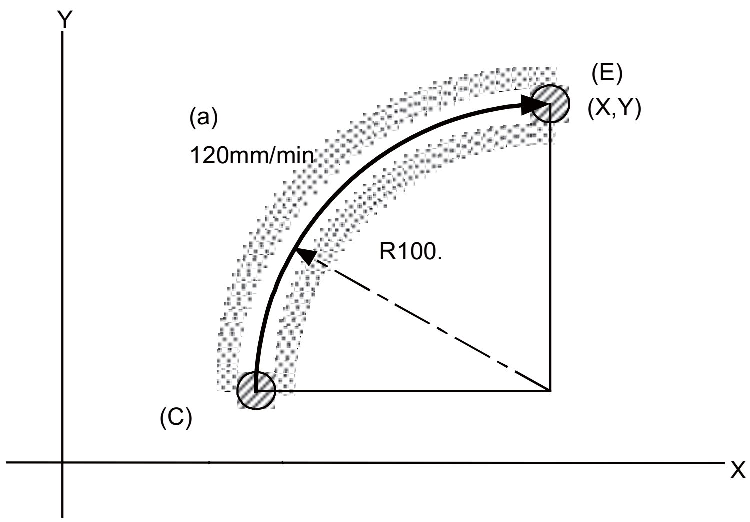
Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

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
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)
G02 G91 X100. Y100. R100. F120 ;



(a) Feed rate
(C) Current position (arc start point)
(E) Arc end point

(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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

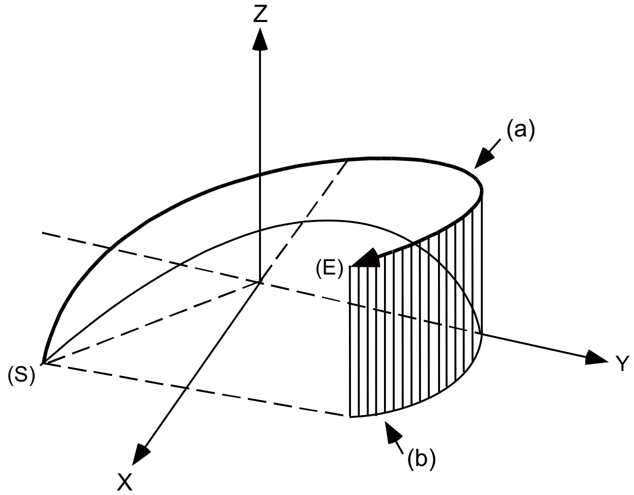
With this function, any two of three axes intersecting orthogonally are made to perform circular interpolation while the third axis performs linear interpolation in synchronization with the arc rotation. This simultaneous 3-axis control can be exercised to machine large-diameter screws or 3-dimensional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

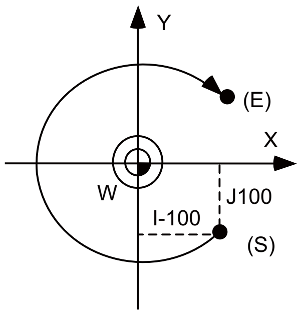
G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Pp1 Ff1 ; (Specify arc center)
G17 G02(G03) Xx1 Yy1 Zz1 Rr1 Ff1 ; (Specify arc radius "R")

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	: Number of pitches
Ff1	: Feed rate
Rr1	: Arc radius

(1) The arc plane is designated by G17, G18 or G19.
(2) G02 or G03 is used to designate the direction of the arc rotation.
(3) Absolute or incremental values can be assigned for the arc end point coordinates and the end point coordinates of the linear axis, but incremental values must be assigned for the arc center coordinates.
(4) The linear interpolation axis is the other axis which is not included in the plane selection.
(5) Command the speed in the component direction that represents all the axes combined for the feed rate.
Pitch l1 is obtained by the formula below.
$l1 = z1/((2\pi * p1+\theta)/2\pi)$
$\theta = \theta e-\theta s = \arctan(ye/xe)-\arctan(ys/xs)$
Where xs, ys are the start point coordinates $(0 \leqq \theta < 2\pi)$
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)
G91 G17 G02 X0. Y200. Z100. I-100. J100. F120 ;

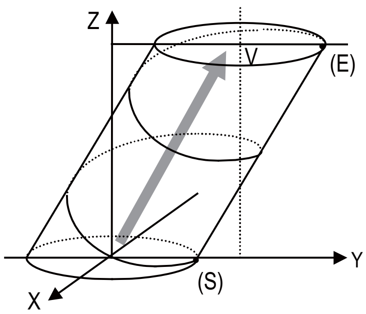




(a) Command program path
(b) XY plane projection path in command program
(S) Start point
(E) End point

(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 be carried out 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.



(E) Start point
(S) End point

3.2.4 Spiral/Conical Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function interpolates arcs where the start point and endpoint are not on the circumference of the same circle into spiral shapes.
There are two types of command formats which can be changed with the parameters.

(1) For command format type 1

(a) Spiral interpolation

G17 G02.1(G03.1) Xx1 Yy1 Ii1 Jj1 Pp1 Ff1 ;	
G17	: Arc plane
G02.1,G03.1	: Arc rotation direction
Xx1,Yy1	: End point coordinate
Ii1,Jj1	: Arc center
Pp1	: Number of pitches
Ff1	: Feed rate

The circular interpolation operation is performed at the feed rate f1 by the commands listed above. The tool draws a spiral arc path whose center is at the position from the start point which is designated by distance i1 for the X-axis direction and distance j1 for the Y-axis direction as the tool moves toward the end point.

The arc plane is designated by G17, G18 or G19.

G17……………XY plane
G18……………ZX plane
G19……………YZ plane

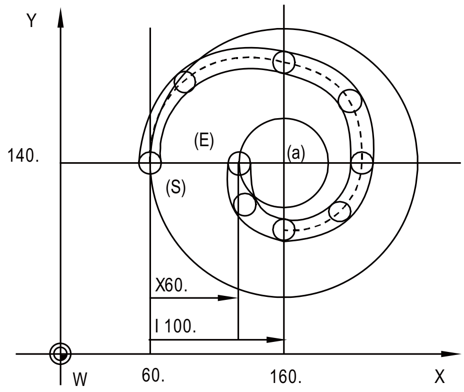
The direction of the arc rotation is designated by G02.1 or G03.1.

G02.1……………Clockwise (CW)
G03.1……………Counterclockwise (CCW)

The number of pitches (number of rotations) is designated by p1.
By assigning zero to p1, the pitch designation can be omitted in this case, the interpolation is obtained as a spiral rotation of less than one full turn. Assigning 1 to p1 yields a spiral rotation of more than one full turn but less than two full turns.

(Example)

```
G91 G17 G01 X60. F500 ;
Y140. ;
G2.1 X60. Y0  I100. P1  F300 ;
G01 X-120 ;
G90
G17  G01  X60. F500;
Y140.;
G2.1   X120. Y140. I100. P1 F300 ;
```



(a) Center
(E) End point
(S) Start point

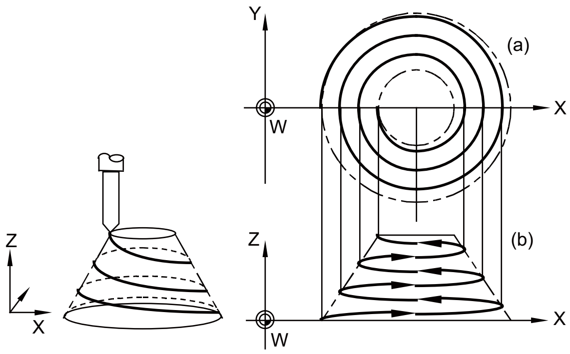
- The combination of the axes which can be commanded simultaneously depends on the specifications. Any combination can be used within the specified range.
- The feed rate is a constant tangential rate.

(Note 1) This function cannot be used in combination with a tool radius compensation command (G41, G42).

(Note 2) The arc plane is always based on the G17, G18 or G19 command. Arc control is performed on a plane by the G17, G18 or G19 command even when two addresses which are not on the selected plane are designated.

(b) Conical interpolation

When an axis other than the ones for the spiral interpolation plane has been designated at the same time, the other axis will also be interpolated in synchronization with the spiral interpolation.
G17 G91 G02.1 X100. Z150. I150. P3 F500;
In the example given above, truncated cone interpolation is performed.



(a) XY plane
(b) XZ plane

(2) For command format type 2

(a) Spiral interpolation

G17 G02(G03) Xx1 Yy1 Ii1 Jj1 Qq1/Ll1 Ff1 ;	
G17	: Arc plane
G02,G03	: Arc rotation direction
Xx1,Yy1	: End point coordinate
Ii1,Jj1	: Arc center
Qq1	: Incremental/decremental amount of radius per spiral rotation
Ll1	: Number of pitches
Ff1	: Feed rate

- Relation between Q and L

  L = ｜ (arc end point radius - arc start point radius) ｜ / ｜ Q ｜
- Q takes precedence if both Q and L have been designated at the same time.

(b) Conical interpolation

G17 G02(G03) Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 /Qq /Ll1 Ff1 ;	
G17	: Arc plane
G02,G03	: Arc rotation direction
Zz1	: End point coordinate in height direction
Ii1,Jj1	: Arc center
Kk1	: Amount by which height is incremented or decremented per spiral rotation
Qq1	: Amount by which radius is incremented or decremented per spiral rotation
Ll1	: Number of pitches
Ff1	: Feed rate

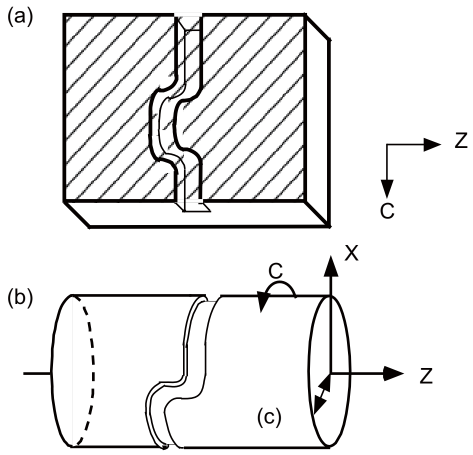
- Relation between L and (I, J) K

  L = ｜ Height ｜ / ｜ Amount by which height is incremented or decremented (I,J,K) ｜
- Q takes precedence over K which in turn takes precedence over L if Q, K and L have been designated at the same time.
- The tolerable error range (absolute position) for when the commanded end point position is deviated from the end point position obtained from the number of pitches and increment/ decrement amount is set with the parameters.

3.2.5 Cylindrical Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function transfers the shape that is on the cylinder's side surface (shape yielded by the cylindrical coordinate system) onto a plane, and when the transferred shape is designated in the program in the form of plane coordinates, the shape is converted into a movement along the linear and rotary axes of the original cylinder coordinates, and the contours are controlled by means of the CNC unit during machining.
Since the programming can be performed for the shapes produced by transferring the side surfaces of the cylinders, this function is useful when it comes to machining cylindrical cams and other such parts.
This function can be used only with the G code list 6 or 7.



(a) Program coordinate plane
(b) Cylindrical interpolation machining
(c) Cylinder radius value

(1) Cylindrical interpolation mode start

(G07.1 name of rotary axis cylinder radius value;)
Cylindrical interpolation is performed between the rotary axis designated in the G07.1 block and any other linear axis.

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

(2) Cylindrical interpolation mode cancel

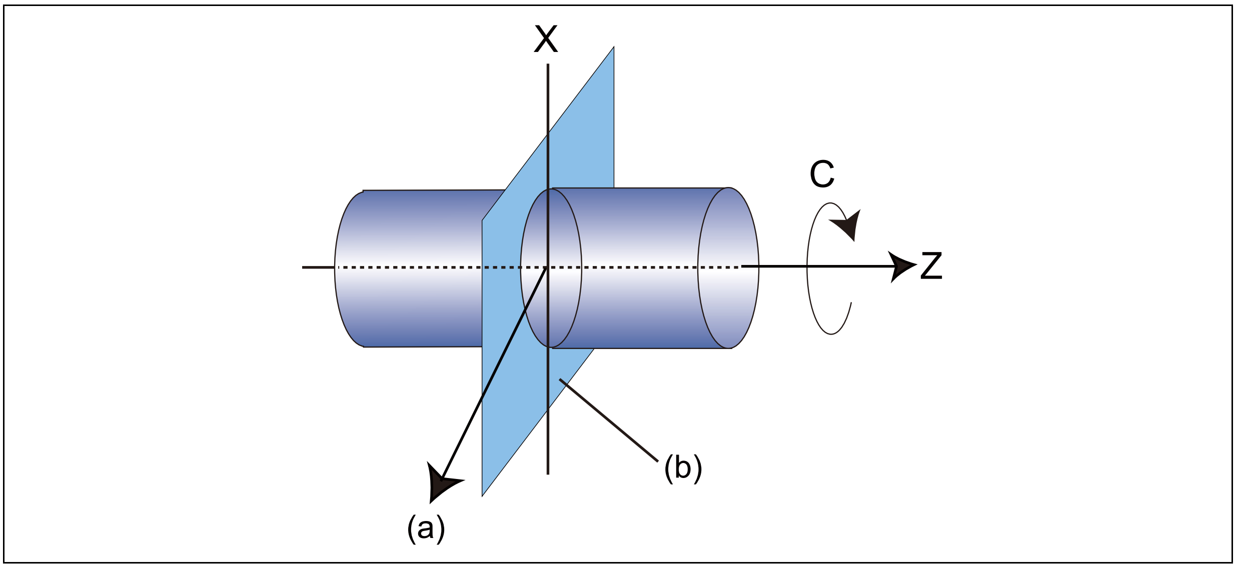
(G07.1 name of rotary axis 0;)
If "C" is the name of the rotary axis, the cylindrical interpolation cancel mode is established with the command below.

G07.1 C0;

3.2.6 Polar Coordinate Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function converts the commands programmed by the orthogonal coordinate axes into linear axis movements (tool movements) and rotary axis movements (workpiece rotation) to control the contours. It is useful for cutting linear cutouts on the outside diameter of the workpiece, grinding cam shafts, etc.
This function can be used only with the G code list 6 or 7.



(a) Hypothetical axis
(b) Polar coordinate interpolation plane (G17 plane)

(1) Polar coordinate interpolation mode

(G12.1)
The polar coordinate interpolation mode is established by designating the G12.1 command.
Polar coordinate interpolation plane consists of a linear axis and a hypothetical axis, which are at right angles to each other.
Polar coordinate interpolation is performed on this plane.

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

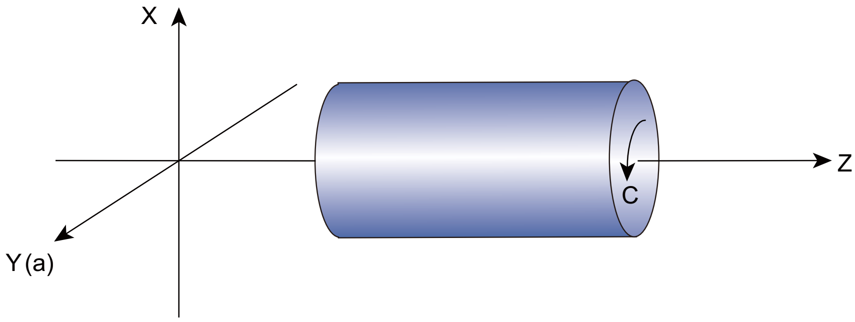
(2) Polar coordinate interpolation cancel mode

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

3.2.7 Milling Interpolation

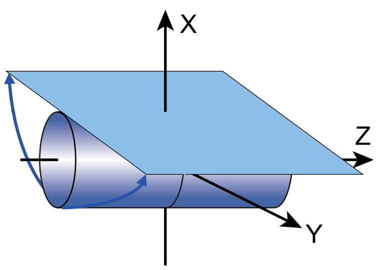
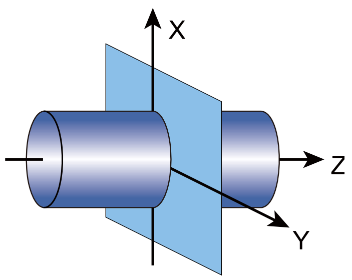
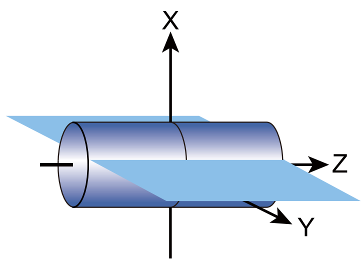
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

When a lathe with linear axes (X, Z axes) and rotary axis (C axis) serving as the control axes is to perform milling at a workpiece end face or in the longitudinal direction of the workpiece, this function uses the hypothetical axis Y which is at right angles to both the X and Z axes to enables the milling shape to be programmed as the X, Y and Z orthogonal coordinate system commands.
With this function, the workpiece can be treated as a cylinder with radius X, and commands can be designated on the plane formed by transferring the cylinder side surface instead.
With milling interpolation, the commands programmed by the orthogonal coordinate system are converted into linear axis and rotary axis movements (workpiece rotation) to control the contours.



(a) (Hypothetical axis)

G12.1 ;	**Milling mode ON**
G13.1 ;	**Milling mode OFF (Turning mode)**

G16(Y-Z cylindrical plane)	G17(X-Y plane)	G19(Y-Z plane)
		
Plane on which radius X cylinder is developed. Select this to machine the cylindrical plane of a workpiece.	X-Y plane in XYZ orthogonal coordinate system. Select this to machine the workpiece end face.	Y-Z plane in XYZ orthogonal coordinate system. Select this to machine a plane of a cylinder cut in the longitudinal direction.

3.2.8 Hypothetical Axis Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

Take one of the axes of the helical interpolation or spiral interpolation, including a linear axis, as a hypothetical axis (axis with no actual movement) and perform pulse distribution. With this procedure, an interpolation equivalent to the helical interpolation or spiral interpolation looked from the side (hypothetical axis), or SIN or COS interpolation, will be possible.
The setting of this hypothetical axis is commanded with G07.

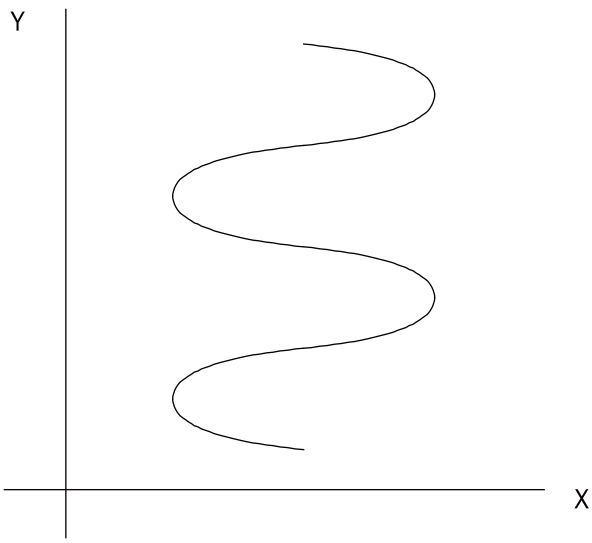
G07 Y0 ; **(X axis command cancel ON)**
G07 Y1 ; **(X axis command cancel OFF)**

	:Hypothetical axis interpolation command
G07	:Designate the axis for which hypothetical axis interpolation is performed
Y	Designation of the axis for which axis command cancellation is performed applies for all the NC axes.
	(0: Cancel (normal), 1: Handle as hypothetical axis)

(1) Interpolation functions that are used for hypothetical interpolation are helical interpolation and spiral interpolation.

(2) During G07 α 0; to G07 α 1;, α axis will be the hypothetical axis. Thus, when α axis is commanded independently during this time, dwell mode will be held until finishing the pulse distribution to the hypothetical axis.

(Example) G07 Z0 ;
G18 G02 X50.Z0.Y100. K30. P3 ;



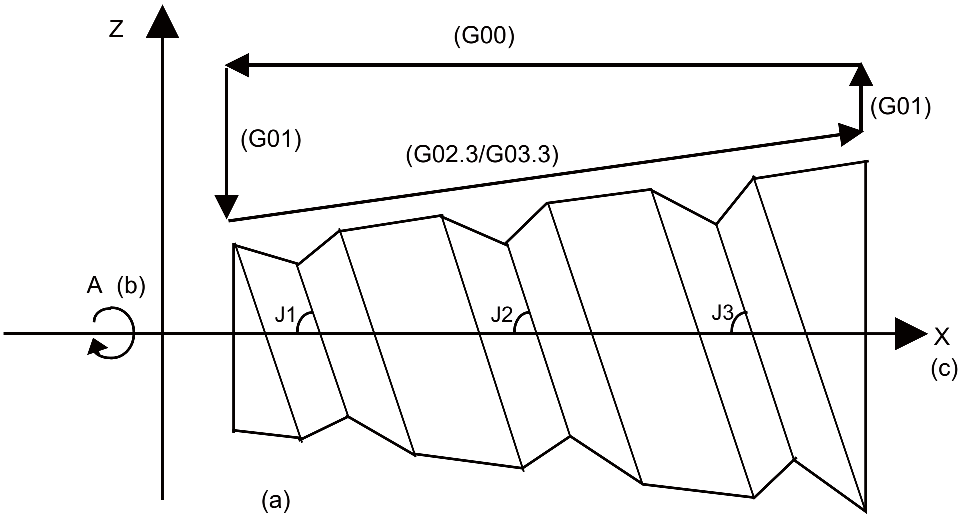
(Note) In order to perform hypothetical axis interpolation, helical interpolation must be added.

3.3 Curve Interpolation

3.3.2 Exponential Interpolation

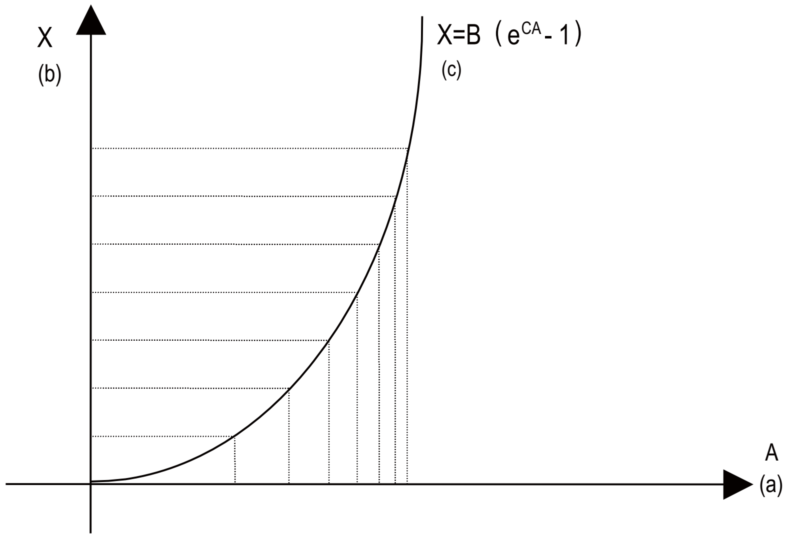
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

With this function, the rotary axis movement is changed into exponential functions vis-a-vis the linear axis movements.
When exponential function interpolation is performed, linear interpolation is performed between the other axes and the linear axis. This makes it possible to machine tapered grooves (regular helix machining of tapered shapes) whose helix angle is always constant.
The function can be used for slotting and grinding end mills and other tools.
[Regular helix machining of tapered shapes]



(a) Helix angle : J1=J2=J3
(b) (rotary axis)
(c) (linear axis)

[Relationship between linear and rotary axes]

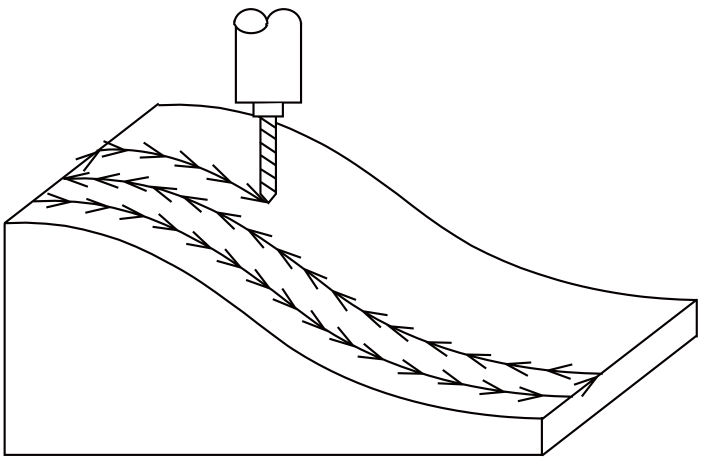


(a) (rotary axis)
(b) (linear axis)
(c) {B, C = constants}

3.3.3 Spline Interpolation (1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function automatically generates spline curves that smoothly pass through rows of dots designated by a fine-segment machining program, and performs interpolation for the paths along the curves. This enables high-speed and high-accuracy machining to be achieved.
To use this function, the high-accuracy control function 1 (G08P1) is required.

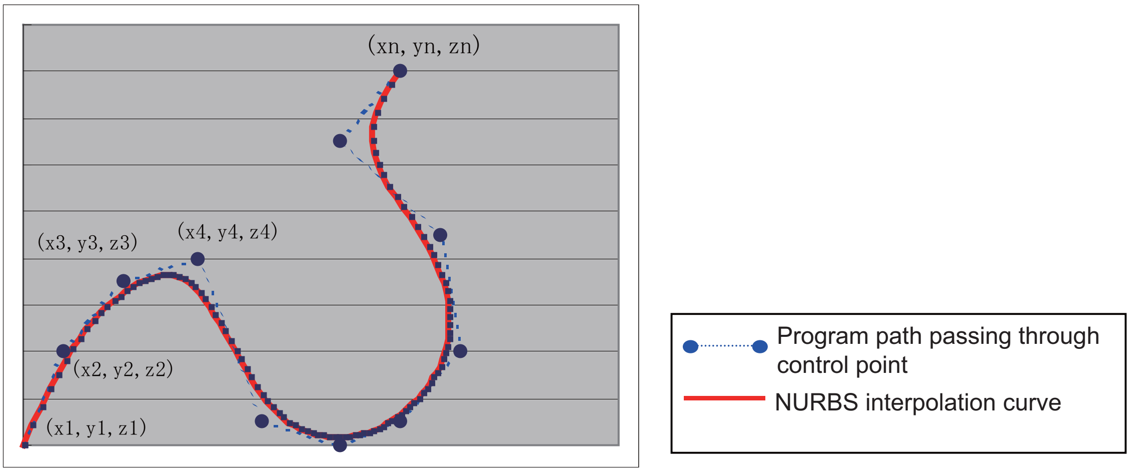


3.3.4 NURBS Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

This function realizes NURBS curve machining by commanding NURBS curve parameters (number of stages, weight, knot, control point). The path does not need to be replaced with fine segments.
This function operates only in the high-speed high-accuracy control 2 mode, so the high-speed high-accuracy control 2 option is required.
During NURBS interpolation, interpolation takes place at the commanded speed. However, if the curvature is large, the speed is clamped so that the machine's tolerable acceleration rate is not exceeded.

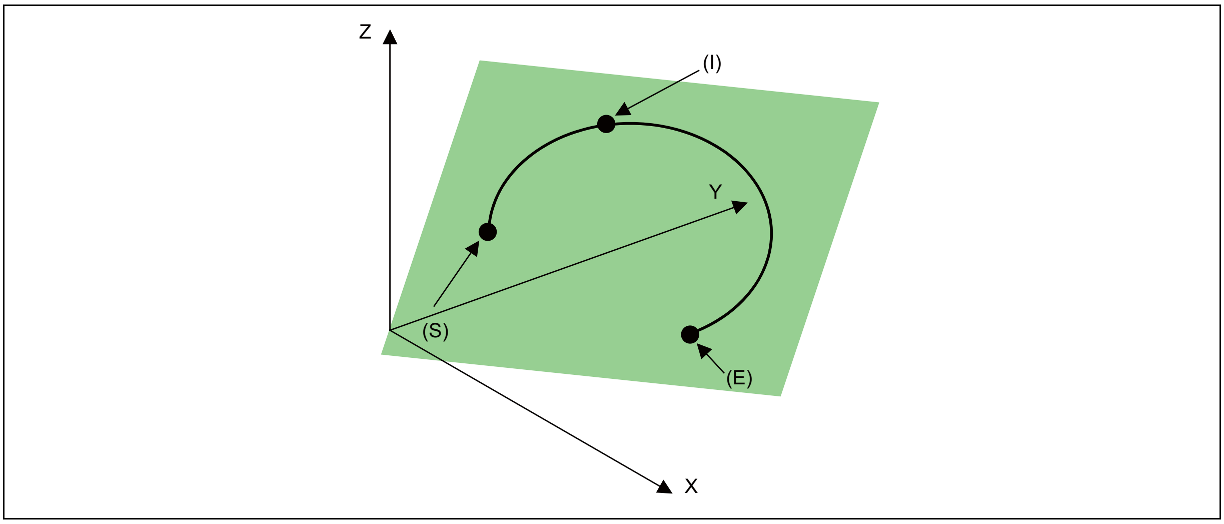
NURBS interpolation cannot be used during graphic check (continuous/step check).
Linear interpolation that connects the control points is used during graphic check.



3.3.5 3-Dimensional Circular Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

To issue a circular command over a three-dimensional space, an arbitrary point (intermediate point) must be designated on the arc in addition to the start point (current position) and end point. Using the 3-dimensional circular interpolation command, an arc shape determined by the three points (start point, intermediate point, end point) designated on the three-dimensional space can be machined.



(E) End point
(I) Intermediate point
(S) Start point (current position)

The command format is shown below.

G02.4(G03.4) Xx1 Yy1 Zz1 αα1 ... ;	**Intermediate point designation (1st block)**
Xx2 Yy2 Zz2 αα2 ... ;	End point designation (2nd block)
G02.4(G03.4)	: 3-dimensional circular interpolation command
Xx1, Yy1, Zz1	: Intermediate point coordinates
Xx2, Yy2, Zz2	: End point coordinates
αα1	: Arbitrary axis other than axis used as the reference in 3-dimensional circular interpolation (May be omitted)

- The operation is the same for G02.4 and G03.4. (The rotation direction cannot be designated.)
- The axes used as the reference in 3-dimensional circular interpolation are the three basic axes set with the parameters.
- The X, Y, Z address in the block may be omitted. The intermediate point coordinates omitted in the 1st block become the start point coordinates, and the end point coordinates omitted in the 2nd block become the intermediate point coordinates.
- When using the 3-dimensional circular interpolation command, an arbitrary axis can be commanded in addition to the orthogonal coordinate system (X, Y, Z) used as the reference. The arbitrary axis designated in the intermediate point designating block (1st block) will interpolate to the command point when moving from the start point to intermediate point movement. The arbitrary axis designated in the end point command block (2nd block) will interpolate to the command point when moving from the intermediate point to the end point. The number of arbitrary axes that can be commanded differs according to the number of simultaneous contour control axes. The total of the basic three axes used as the reference of the 3-dimensional circular interpolation and the arbitrary axes commanded simultaneously must be less than the number of simultaneous contour control axes.

4

Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	1000	1000	1000	1000	1000	1000
L system	1000	1000	1000	1000	1000	1000

[M system]

The rapid traverse rate can be set independently for each axis by the parameter.
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.
If the high-accuracy control mode's rapid traverse rate is set, the axis will move at that feed rate during high-accuracy control, high-speed high-accuracy control, high-accuracy spline control or SSS control.

- If the value set for the high-accuracy control mode rapid traverse rate is 0, the axis will move at the rapid traverse rate.
- The high-accuracy control mode rapid traverse rate can be set independently for each axis.
- The high-accuracy control mode rapid traverse rate is effective for the G00, G27, G28, G29, G30 and G60 commands.
- Override can be applied on the high-accuracy control mode rapid traverse rate using the external signal supplied.

Rapid traverse rate and high-accuracy control mode rapid traverse rate setting

Least command increment	B	C	D	E
Metric input (mm/min, °/min)	1 to 1000000	1 to 1000000	1 to 1000000	1 to 1000000
Inch input (inch/min)	1 to 100000	1 to 100000	1 to 100000	1 to 100000

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

[L system]

The rapid traverse rate can be set independently for each axis by the parameter.
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 command increment	B	C	D	E
Metric input (mm/min, °/min)	1 to 1000000	1 to 1000000	1 to 1000000	1 to 1000000
Inch input (inch/min)	1 to 100000	1 to 100000	1 to 100000	1 to 100000

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

4.1.2 Cutting Feed Rate (m/min)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	1000	1000	1000	1000	1000	1000
L system	1000	1000	1000	1000	1000	1000

[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).

If the high-accuracy control mode's cutting clamp speed is set, the cutting feed rate will be clamped at that speed during high-accuracy control, high-speed high-accuracy control, high-accuracy spline control or SSS control.

- If the value set for high-accuracy control mode cutting clamp speed is 0, the axis will be clamped at the cutting feed clamp speed.
- High-accuracy control mode cutting clamp speed is set with the parameters.

Cutting feed rate setting range

Least command increment	B	C	D	E
Metric input (mm/min, ° / min)	0.001 to 1000000	0.0001 to 1000000	0.00001 to 1000000	0.000001 to 1000000
Inch input (inch/min)	0.0001 to 100000	0.00001 to 100000	0.000001 to 100000	0.0000001 to 100000

Least command increment B : 0.001mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

- G code command for which the cutting feed rate is effective

  For others such as G01,G02,G03,G02.1,G03.1,G33, etc., 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 command increment	B	C	D	E
Metric input (mm/min, ° / min)	0.001 to 1000000	0.0001 to 1000000	0.00001 to 1000000	0.000001 to 1000000
Inch input (inch/min)	0.0001 to 100000	0.00001 to 100000	0.000001 to 100000	0.0000001 to 100000

Least command increment B : 0.001mm (0.0001inch)

Least command increment C : 0.0001mm (0.00001inch)

- G code command for which the cutting feed rate is effective

  For others such as G01,G02,G03,G02.1,G03.1,G33, etc., refer to the interpolation specifications.

4.1.3 Manual Feed Rate (m/min)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	1000	1000	1000	1000	1000	1000
L system	1000	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 include two methods, the code method and value setting 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.

4.1.4 Rotary Axis Command Speed Tenfold

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function multiplies the rotary axis' command speed by 10 during initial inching.
The commanded speeds are as follow.

Automatic operation	**Cutting feed rate**	For the inch system, the rotary axis command speed is multiplied by 10. For example, if the B axis is the rotary axis in the inch system and the following type of machining program is executed, the rotary axis command speed will be multiplied by 10, and the rotary axis will move at 1000 deg./min. N1 G1 B100. F100.;
	Rapid traverse rate	The rapid traverse rate is not multiplied by 10, and is the speed set in the parameters.
Manual operation		The command speeds related to manual operation, such as JOG feed, are not multiplied by 10. The display speed unit also remains as "deg./min".

4.2 Feed Rate Input Methods

4.2.1 Feed per Minute (Asynchronous Feed)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

[M system]

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

Metric input

Input Setting unit	B(0.001mm)	C(0.0001mm)	D(0.00001mm)	E(0.000001mm)
Command Mode	Feed per minute	Feed per minute	Feed per minute	Feed per minute
Command Address	F(mm/min)	F(mm/min)	F(mm/min)	F(mm/min)
Minimum command unit	1(=1.00) (1.=1.00)	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)
Command range	0.01 - 1000000.00	0.001 - 1000000.000	0.0001 - 1000000.0000	0.00001 - 1000000.00000

Inch input

Input setting unit	B(0.0001inch)	C(0.00001inch)	D(0.000001inch)	E(0.0000001inch)
Command Mode	Feed per minute	Feed per minute	Feed per minute	Feed per minute
Command Address	F(inch/min)	F(inch/min)	F(inch/min)	F(inch/min)
Minimum command unit	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)
Command range	0.001 - 100000.000	0.0001 - 100000.0000	0.00001 - 100000.00000	0.000001 - 100000.000000

[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

Input Setting unit	B(0.001mm)	C(0.0001mm)	D(0.00001mm)	E(0.000001mm)
Command Mode	Feed per minute	Feed per minute	Feed per minute	Feed per minute
Command Address	F(mm/min)	F(mm/min)	F(mm/min)	F(mm/min)
Minimum command unit	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)
Command range	0.001 - 1000000.000	0.0001 - 1000000.0000	0.00001 - 1000000.00000	0.000001 - 1000000.000000

Inch input

Input setting unit	B(0.0001inch)	C(0.00001inch)	D(0.000001inch)	E(0.0000001inch)
Command Mode	Feed per minute	Feed per minute	Feed per minute	Feed per minute
Command Address	F(inch/min)	F(inch/min)	F(inch/min)	F(inch/min)
Minimum command unit	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)
Command range	0.0001 - 100000.0000	0.00001 - 100000.00000	0.000001 - 100000.000000	0.0000001 - 100000.0000000

4.2.2 Feed per Revolution (Synchronous Feed)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	○	○	○	○	○	○

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

Metric input

Input Setting unit	B(0.001mm)	C(0.0001mm)	D(0.00001mm)	E(0.000001mm)
Command Mode	Feed per revolution	Feed per revolution	Feed per revolution	Feed per revolution
Command Address	F(mm/rev)	F(mm/rev)	F(mm/rev)	F(mm/rev)
Minimum command unit	1(=0.001) (1.=1.0)	1(=0.0001) (1.=1.0)	1(=0.00001) (1.=1.0)	1(=0.000001) (1.=1.0)
Command range	0.001 - 999.999	0.0001 - 999.9999	0.00001 - 999.99999	0.000001 - 999.999999

Inch input

Input setting unit	B(0.0001inch)	C(0.00001inch)	D(0.000001inch)	E(0.0000001inch)
Command Mode	Feed per revolution	Feed per revolution	Feed per revolution	Feed per revolution
Command Address	F(inch/rev)	F(inch/rev)	F(inch/rev)	F(inch/rev)
Minimum command unit	1(=0.00001) (1.=1.00)	1(=0.000001) (1.=1.00)	1(=0.0000001) (1.=1.00)	1(=0.00000001) (1.=1.00)
Command range	0.00001 - 99.99999	0.000001 - 99.999999	0.0000001 - 99.9999999	0.00000001 - 99.99999999

[L system]

Metric input

Input Setting unit	B(0.001mm)	C(0.0001mm)	D(0.00001mm)	E(0.000001mm)
Command Mode	Feed per revolution	Feed per revolution	Feed per revolution	Feed per revolution
Command Address	F(mm/rev)	F(mm/rev)	F(mm/rev)	F(mm/rev)
Minimum command unit	1(=0.0001) (1.=1.00)	1(=0.00001) (1.=1.00)	1(=0.000001) (1.=1.00)	1(=0.0000001) (1.=1.00)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.000001 - 999.999999	0.0000001 - 999.9999999

Inch input

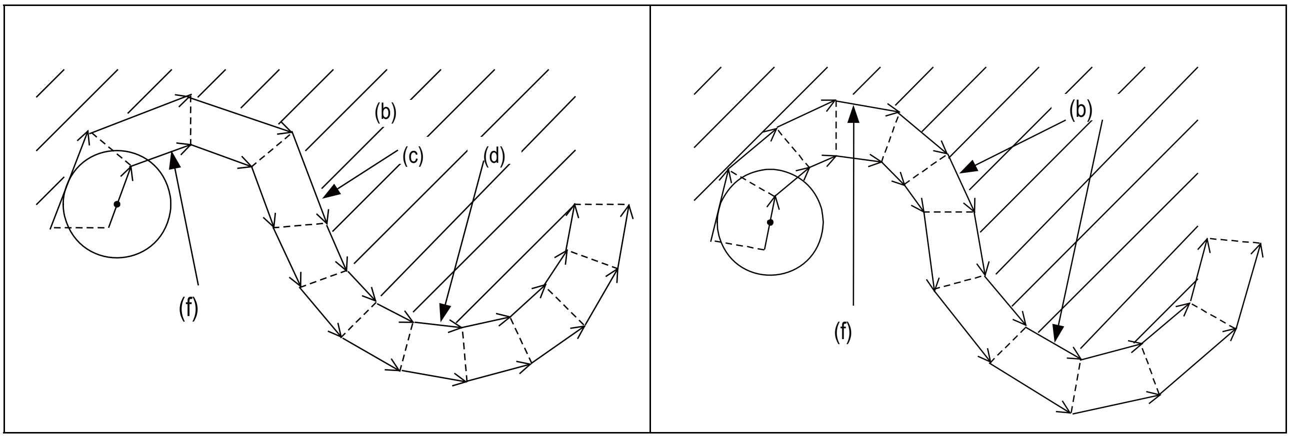
Input setting unit	B(0.0001inch)	C(0.00001inch)	D(0.000001inch)	E(0.0000001inch)
Command Mode	Feed per revolution	Feed per revolution	Feed per revolution	Feed per revolution
Command Address	F(inch/rev)	F(inch/rev)	F(inch/rev)	F(inch/rev)
Minimum command unit	1(=0.000001) (1.=1.000)	1(=0.0000001) (1.=1.000)	1(=0.00000001) (1.=1.000)	1(=0.000000001) (1.=1.000)
Command range	0.000001 - 99.999999	0.0000001 - 99.9999999	0.00000001 - 99.99999999	0.000000001 - 99.999999999

4.2.3 Inverse Time Feed

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

During inside cutting when machining curved shapes with tool radius compensation applied, the machining speed on the cutting surface becomes faster than the tool center feedrate. Therefore, problems such as reduced accuracy on the cutting surface may occur.
This reduced accuracy can be prevented with inverse time feed. This function can, in place of normal feed commands, issue one block of machining time (inverse) in F commands. The machining speed on the cutting surface is constantly controlled, even if radius compensation is applied to the machining program that expresses the free curve surface with fine segment lines.
Note that when the calculated machining time exceeds the cutting feed clamp speed, the F command value in the inverse time feed follows the cutting feed clamp speed.

Regular F command **Inverse time feed**



(b) Actual machining speed
(c) Large
(d) Small
(f) F command

(b) Same
(f) F command

Command format is as shown below.

G93 ;	**Inverse time feed**

Inverse time feed (G93) is a modal command and is valid until feed per minute (G94) or feed per revolution (G95) is commanded.

G00 Xx1 Yy1 ;	
G93 ;	→ Inverse time feed mode ON
G01 Xx2 Yy2 Ff2 ;	→ In inverse time feed mode
G02 Xx3 Yy3 Ii3 Jj3 Ff3 ;	:
G94(G95) ;	→ Inverse time feed mode OFF

In movement blocks, since processing time is commanded to a line segment, command the feedrate "F" each time.

4.2.4 F 1-digit Feed

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The feed rate registered by parameter 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 feedrate set to support the code serves as the valid rate command.
If F6 or larger value is command, the value is regarded as the cutting feedrate which has been directly commanded with numerical values.
When an F 1-digit command has been issued, the external output signal is output.

When the programmed feed rate has been issued as an F 1-digit command, the feed rate can be increased or reduced by turning the manual handle.
The feed rate cannot be changed by the 2nd and 3rd handles.

(1) Amount by which speed is varied by manual handle

Speed variation amount ΔF is expressed by the equation below:

$$\Delta F = \Delta P \times \frac{FM}{K}$$

ΔP : Handle pulses (±)
FM : F1 to F5 upper limit (parameter setting)
K :Speed variation constant (parameter setting)

(Example) When the feed rate is to be increased or reduced by 10 mm/min per manual handle scale increment
If FM is 3600 mm/min, then:

$$\Delta F = 10 = 1 \times \frac{3600}{K}$$

Therefore, K = 360.

(2) Conditions under which F1-digit feed is valid

(a) The automatic operation must be selected.
(b) Automatic start must be underway.
(c) Cutting feed must be underway, and the F 1-digit feed rate must be designated.
(d) The F 1-digit valid parameter must be ON.
(e) The F 1-digit feed rate change valid signal must be ON.
(f) A dry run must not be in progress.
(g) Machine lock must not be activated.

4.2.5 Manual Speed Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

In the memory or MDI mode, validate the manual speed command and select either handle feed or jog (manual) feed so that the automatic operation is carried out at the feedrate.
With a command in the (-) direction, the program path can be reversed. Note that, however, program path can be reversed only within the currently executing block and not beyond the block.
Whether or not to execute reverse run with a command in the (-) direction is set with the PLC interface.
Furthermore, by setting the parameter, handle, jog and manual feed speed can be executed at the feed speed according to the ratio of program command speed of running block when issuing the manual speed command in multiple systems.

4.3 Override

4.3.1 Rapid Traverse Override

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(2) Type 2 (value setting method)
Override can be applied in 1% steps from 0% to 100% to manual or automatic rapid traverse using the external input signal supplied.

(Note 1) Type 1 and type 2 can be selected by PLC processing.
(Note 2) A PLC must be built into the unit for type 2.

4.3.2 Cutting Feed Override

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(1) Type 1 (code method)
Override can be applied in 10% steps from 0% to 300% to the feed rate command designated in the machining program using the external input signal supplied.

(2) Type 2 (value setting method)
Override can be applied in 1% steps from 0% to 327% to the feed rate command designated in the machining program using the external input signal supplied.

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

4.3.3 2nd Cutting Feed Override

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

4.3.4 Override Cancel

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

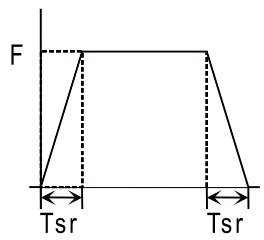
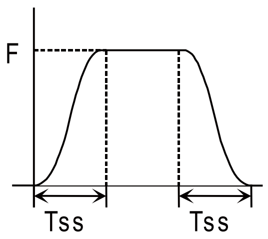
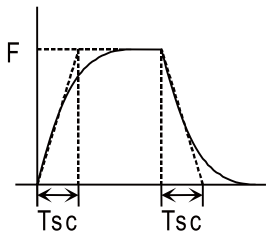
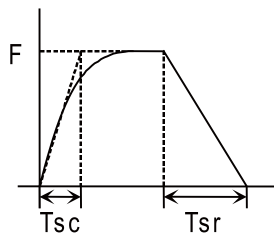
(Note 1) The override cancel signal is not valid for manual operation.
(Note 2) When the cutting feed override or second cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration after Interpolation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

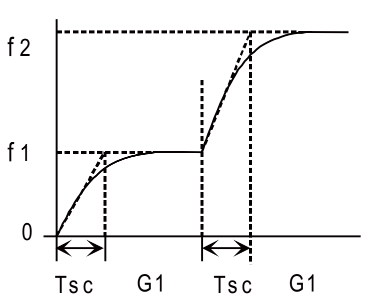
Linear acceleration/deceleration	Soft acceleration/deceleration	Exponential acceleration/ deceleration	Exponential acceleration / linear deceleration
			

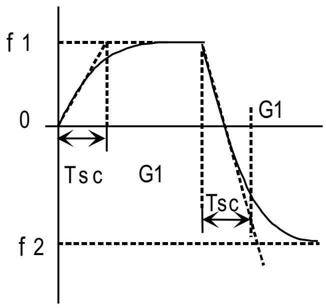
(Note 1) The rapid traverse feed acceleration/deceleration patterns are effective for the following:
G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG feed, incremental feed, return to reference position.

(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

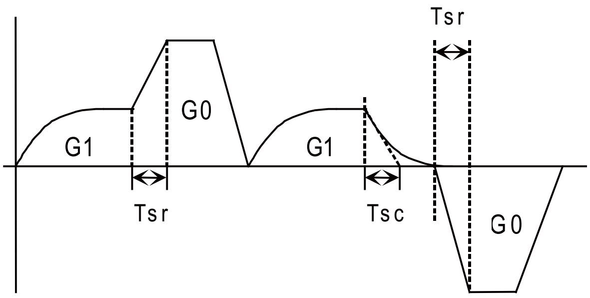
(1) Continuous G1 blocks

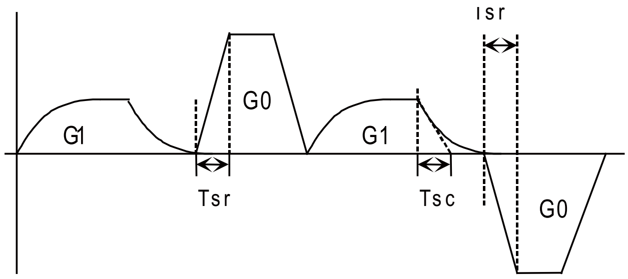




The tool does not decelerate between blocks.

(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.)

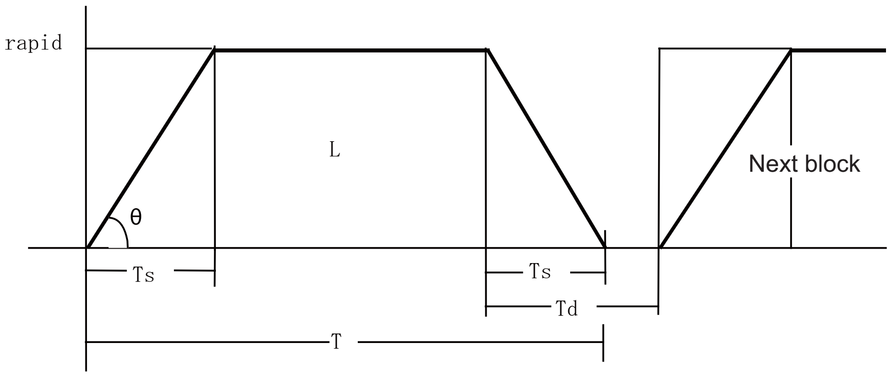
4.4.2 Rapid Traverse Constant Inclination Acceleration/Deceleration

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

Rapid traverse constant inclination acceleration/deceleration are valid only for a rapid traverse command. Also, this function is effective only when the rapid traverse command acceleration/deceleration mode is linear acceleration and linear deceleration.
The acceleration/deceleration patterns in the case where rapid traverse constant inclination acceleration/ deceleration are performed are as follows.

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



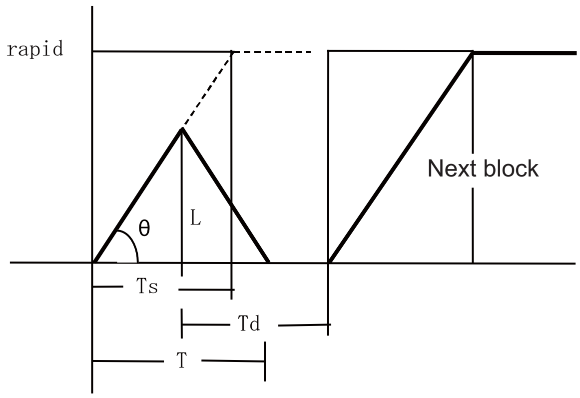
$$T = \frac{L}{rapid} + Ts$$

Td = Ts + (0 ～1.7ms)

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

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

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



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

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

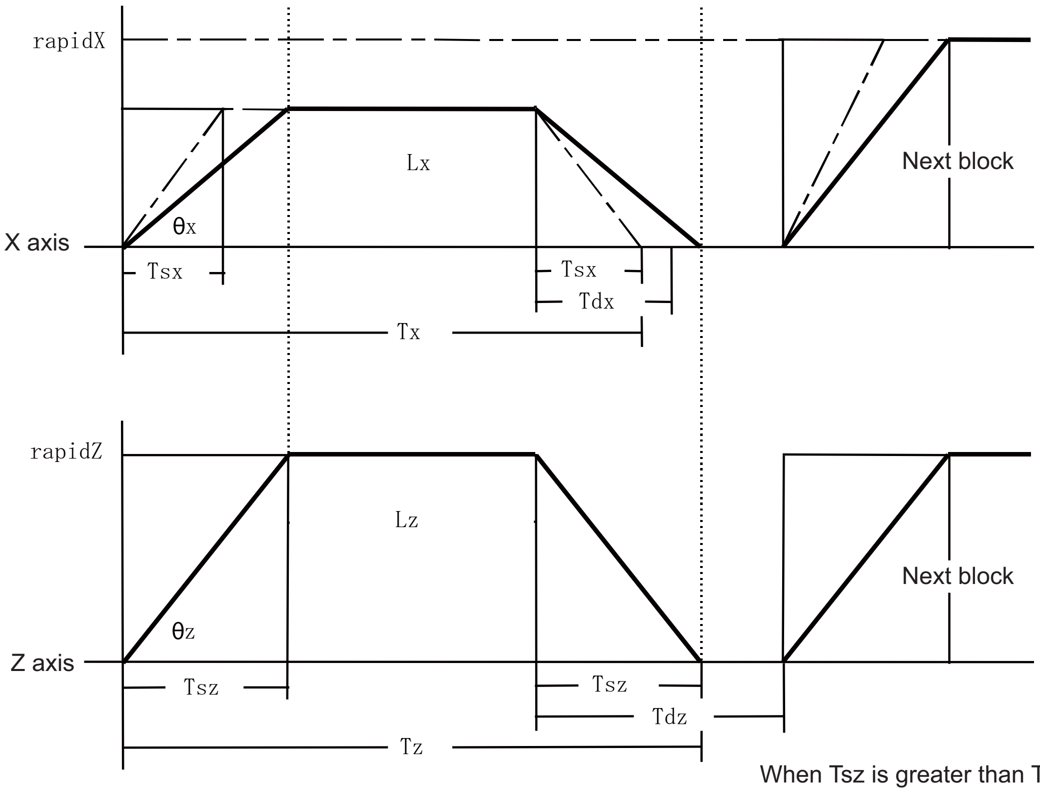
$$q = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

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

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.

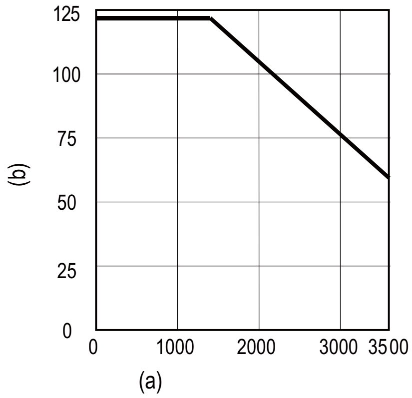
The program format of G0 (rapid traverse command) when rapid traverse constant inclination acceleration/deceleration are executed is the same as when this function is invalid (time constant acceleration/deceleration).

This function is valid only for G0 (rapid traverse).

4.4.3 Rapid Traverse Constant Inclination Multi-step Acceleration/Deceleration (1st Part System Only)

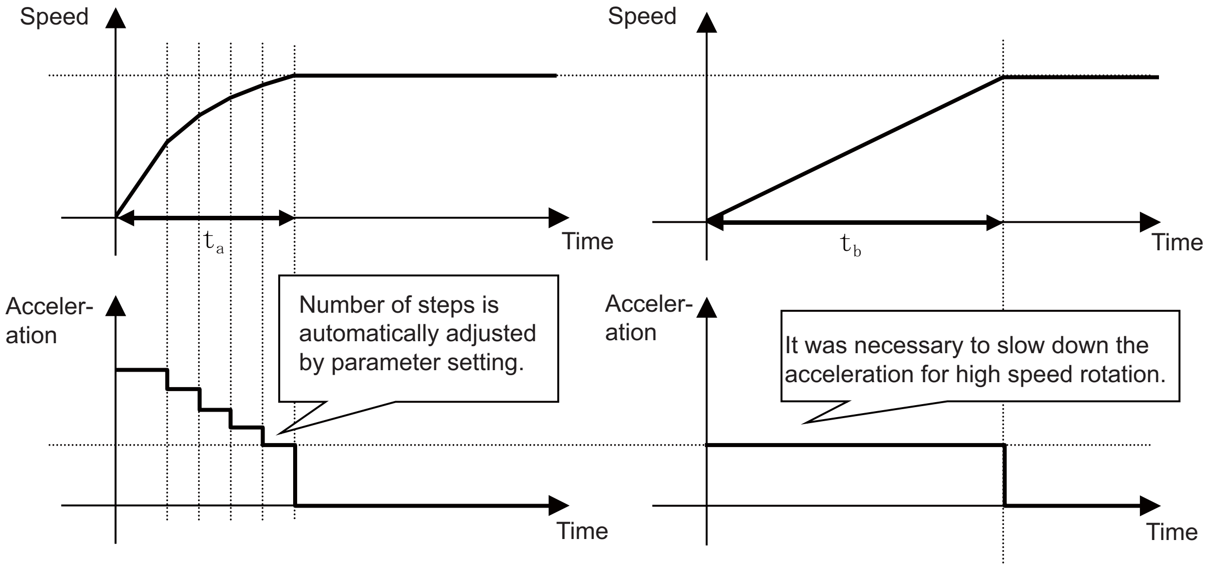
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function carries out the acceleration/deceleration according to the torque characteristic of the motor in the rapid traverse mode during automatic operation. (This function is not available in manual operation.) The rapid traverse constant inclination multi-step acceleration/deceleration method makes for improved cycle time because the positioning time is shortened by using the motor ability to its maximum.
In general, the servomotor has the characteristic that the torque falls in the high-speed rotation range.



(a) Rotation speed [r/min]
(b) Torque [N*m]
(Note) This characteristic is data at input voltage 380VAC.

In the rapid traverse constant inclination acceleration/deceleration method, the acceleration has been treated constantly because this torque characteristic is not considered. So, it is necessary to use a minimum acceleration within the used speed range. Therefore, the margin of acceleration must be had in a low-speed range. Or if the acceleration is used to its maximum, the upper limit of the rotation speed must be slowed.
Then, to use the servomotor ability to its maximum, acceleration/deceleration to which the torque characteristic is considered is carried out by the rapid traverse constant inclination multi-step acceleration/deceleration method.
The acceleration/deceleration patterns in the case where rapid traverse constant inclination multi-step acceleration/deceleration are performed are as follows.



(a) Rapid traverse constant inclination multi-step acceleration/deceleration

(b) Rapid traverse constant inclination acceleration/deceleration

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	○	○	○	○	○	○

Thread cutting with designated lead can be performed. Designate the number of threads per inch with the E address to cut inch threads.

(1) Lead designation

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

G33 Zz1/Ww1 Xx1/Uu1 Qq1 Ff1/Ee1 ;

G33	: Thread cutting command
Zz1/Ww1,Xx1/Uu1	: Thread end point coordinates
Qq1	: Shift angle at start of thread cutting (0.000 to 360.000°)
Ff1	: Thread lead (normal lead threads)
Ee1	: Thread lead (precise lead threads)

The thread cutting with designated lead can be performed. Inch threads are cut by designating the number of threads per inch with the E address.

(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/Ww1 Xx1/Uu1 Qq1 Ee1 ;

G33	: Thread cutting command
Zz1/Ww1,Xx1/Uu1	: Thread end point coordinates
Qq1	: Shift angle at start of thread cutting (0.000 to 360.000°)
Ee1	: Thread number per inch

[M system]

Thread cutting metric input

Input setting unit	B (0.001mm)			C (0.0001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=1.000) (1.=1.000)	1(=1.0000) (1.=1.0000)	1(=1.00) (1.=1.00)	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000) (1.=1.000)
Command range	0.001 - 999.999	0.0001 - 999.9999	0.03 - 999.99	0.0001 - 999.9999	0.00001 - 999.99999	0.026 - 222807.017

Input setting unit	D (0.00001mm)			E (0.000001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.0000) (1.=1.0000)	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)	1(=1.00000) (1.=1.00000)
Command range	0.00001 - 999.99999	0.000001 - 999.999999	0.0255 - 224580.0000	0.000001 - 999.999999	0.0000001 - 999.9999999	0.02541 - 224719.00000

III Functional Specifications

Thread cutting inch input

Input setting unit	B (0.0001inch)			C (0.00001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.000) (1.=1.000)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.0000) (1.=1.0000)
Command range	0.0001 - 39.3700	0.00001 - 39.37007	0.025 - 9999.999	0.00001 - 39.37007	0.000001 - 39.370078	0.0255 - 9999.9999

Input setting unit	D (0.000001inch)			E (0.0000001inch)		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)	1(=1.00000) (1.=1.00000)	1(=1.0000000) (1.=1.0000000)	1(=1.00000000) (1.=1.00000000)	1(=1.000000) (1.=1.000000)
Command range	0.000001 - 39.370078	0.0000001 - 39.3700787	0.02541 - 9999.99999	0.0000001 - 39.3700787	0.00000001 - 39.37007873	0.025401 - 9999.999999

[L system]

Thread cutting metric input

Input setting unit	B (0.001mm)			C (0.0001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=1.0000) (1.=1.0000)	1(=1.00000) (1.=1.00000)	1(=1.00) (1.=1.00)	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.000) (1.=1.000)
Command range	0.0001 - 999.9999	0.00001 - 999.99999	0.03 - 999.99	0.00001 - 999.99999	0.000001 - 999.999999	0.026 - 222807.017

Input setting unit	D (0.00001mm)			E (0.000001mm)		
Command address	F (mm/rev)	E (mm/rev)	E (ridges/inch)	F (mm/rev)	E (mm/rev)	E (ridges/inch)
Least Command Increments	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)	1(=1.0000) (1.=1.0000)	1(=1.0000000) (1.=1.0000000)	1(=1.00000000) (1.=1.00000000)	1(=1.00000) (1.=1.00000)
Command range	0.000001 - 999.999999	0.0000001 - 999.9999999	0.0255 - 224580.0000	0.0000001 - 999.9999999	0.00000001 - 999.99999999	0.02540 - 224719.00000

Thread cutting inch input

Input setting unit	**B (0.0001inch)**			**C (0.00001inch)**		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.00000) (1.=1.00000)	1(=1.000000) (1.=1.000000)	1(=1.000) (1.=1.000)	1(=1.000000) (1.=1.000000)	1(=1.0000000) (1.=1.0000000)	1(=1.0000) (1.=1.0000)
Command range	0.00001 - 39.37007	0.000001 - 39.370078	0.025 - 9999.999	0.000001 - 39.370078	0.0000001 - 39.3700787	0.0254 - 9999.9999

Input setting unit	**D (0.000001inch)**			**E (0.0000001inch)**		
Command address	F (inch/rev)	E (inch/rev)	E (ridges/inch)	F (inch/rev)	E (inch/rev)	E (ridges/inch)
Least Command Increments	1(=1.0000000) (1.=1.0000000)	1(=1.00000000) (1.=1.00000000)	1(=1.00000) (1.=1.00000)	1(=1.00000000) (1.=1.00000000)	1(=1.000000000) (1.=1.000000000)	1(=1.000000) (1.=1.000000)
Command range	0.0000001 - 39.3700787	0.00000001 - 39.37007873	0.02540 - 9999.99999	0.00000001 - 39.37007873	0.000000001 - 39.370078736	0.025400 - 9999.999999

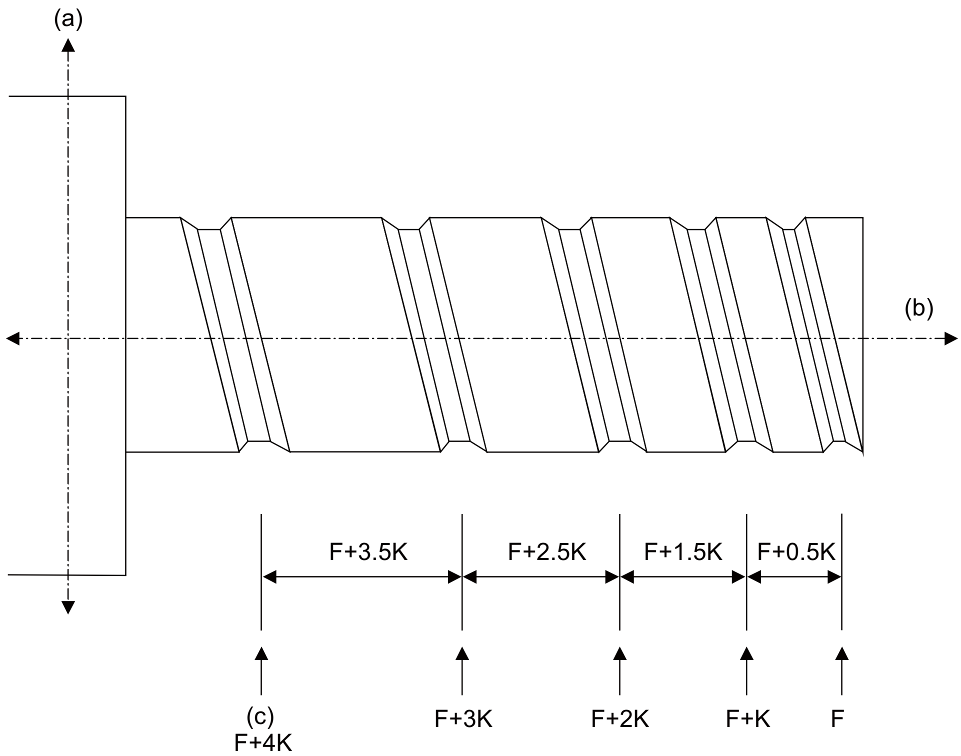
4.5.2 Variable Lead Thread Cutting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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

G34 Xx1/Uu1 Zz1/Ww1 Ff1/Ee1 Kk1 ;

G34	: Variable lead thread cutting command
Xx1/Uu1	: Thread end point X coordinate
Zz1/Ww1	: Thread end point Z coordinate
Ff1/Ee1	: Thread's basic lead
Kk1	: Lead increment/decrement amount per thread rotation



(a) Non-lead axis
(b) Lead axis
(c) Lead speed

4.5.3 Synchronous Tapping

4.5.3.1 Synchronous Tapping Cycle

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function performs tapping through the synchronized control of the digital 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 Dd1 Pp1 Ff1 Kk1 Ss1 ,Ss2 ,Rr2 ,Ii1 ,Jj1 Mm1 ;

G84	: Mode, forward tapping
G74	: Mode, reverse tapping
Xx1,Yy1	: Hole position data, hole drilling position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, point R position
Dd1	: Tool spindle number (d is 1 to number of spindles) Depending on the parameter setting, command as "-d1" to carry out reverse tapping.
Pp1	: Hole machining data, dwell time at hole bottom
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Kk1	: Number of repetitions
Ss1	: Spindle speed
,Ss2	: Rotation speed of spindle during retract
,Rr2	: Synchronization method selection (r2=1 Synchronous, r2=0 Asynchronous)
,Ii1/,Jj1	: In-position width of positioning axis/hole drilling axis
Mm1	: M function designation

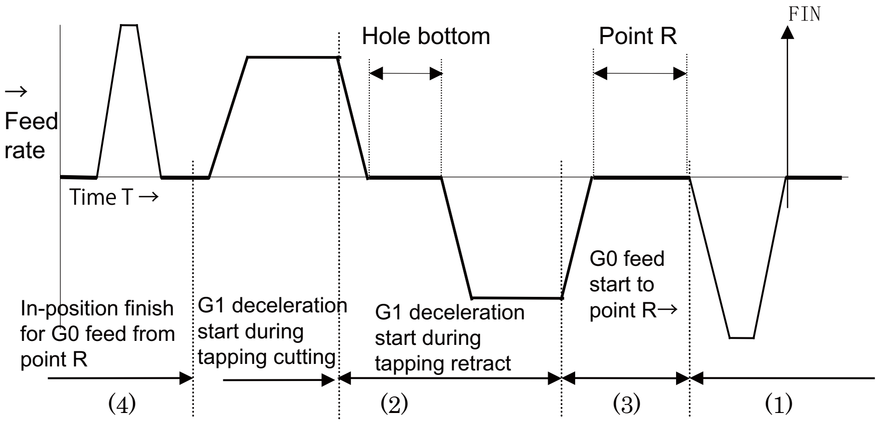
(2) Tapping thread number assignment

G84(G74) Xx1 Yy1 Zz1 Rr1 Dd1 Pp1 Ee1 Kk1 Ss1 ,Ss2 ,Rr2 ,Ii1 ,Jj1 Mm1 ;

G84	: Mode, forward tapping
G74	: Mode, reverse tapping
Xx1,Yy1	: Hole position data, hole drilling position
Zz1	: Hole machining data, hole bottom position
Rr1	: Hole machining data, point R position
Dd1	: Tool spindle number (d is 1 to number of spindles) Depending on the parameter setting, command as "-d1" to carry out reverse tapping.
Pp1	: Hole machining data, dwell time at hole bottom
Ee1	: Tap thread number per 1-inch feed of Z axis
Kk1	: Number of repetitions
Ss1	: Spindle speed
,Ss2	: Rotation speed of spindle during retract
,Rr2	: Synchronization method selection (r2=1 synchronous, r2=0 asynchronous)
,Ii1/,Jj1	: In-position width of positioning axis/hole drilling axis
Mm1	: M function designation

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

tapping axis of the synchronous tapping in-position check.



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

4.5.3.2 Pecking Tapping Cycle

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

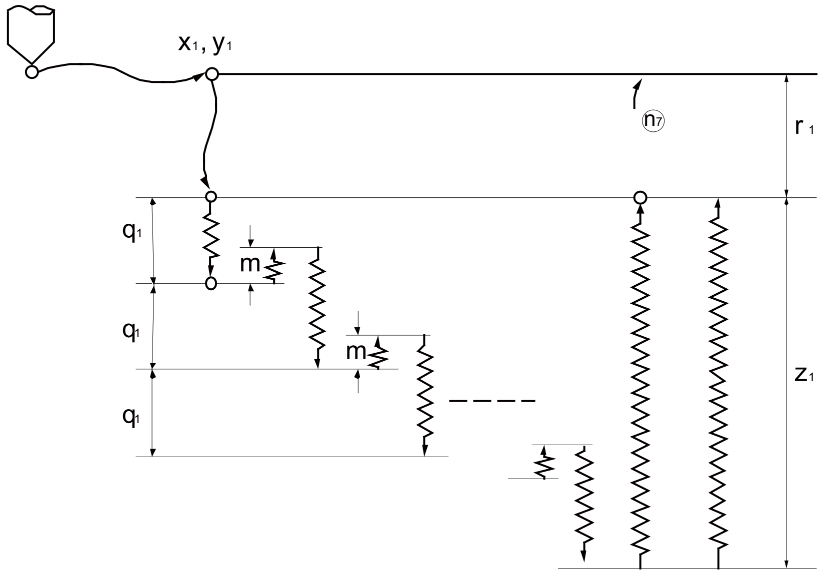
The load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.
The amount retracted from the hole bottom is set to the parameters.
Select either the pecking tapping cycle or the deep-hole tapping cycle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

When Q is not designated
When the command value of Q is zero

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii1 ,Jj1 ,Rr2 ;

G84	: G84 forward tapping cycle
G74	: G74 reverse tapping cycle
Xx1,Yy1	: Hole drilling position
Zz1	: Hole bottom position
Rr1	: Point R position
Qq1	: Depth of cut per pass (designated as an incremental position)
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Ee1	: Tap thread number per 1-inch feed of Z axis
Pp1	: Dwell time at hole bottom position
Ss1	: Rotation speed of spindle
,Ss2	: Rotation speed of spindle during retract
,Ii2	: In-position width of positioning axis
,Jj2	: In-position width of hole drilling axis
,Rr2	: Synchronization method selection (r2=1 synchronous, r2=0 asynchronous)

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



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

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

4.5.3.3 Deep-hole Tapping Cycle

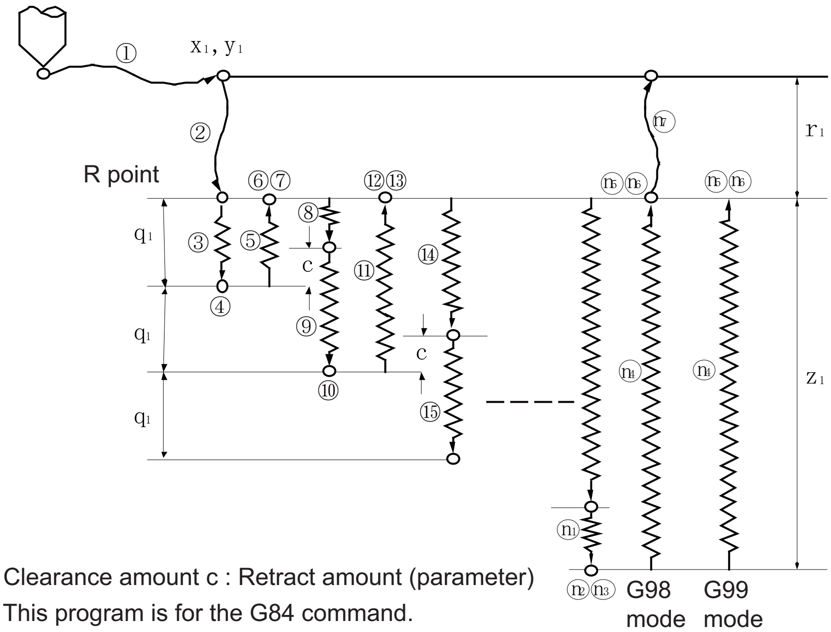
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

In the deep-hole tapping, the load applied to the tool can be reduced by designating the depth of cut per pass and cutting the workpiece to the hole bottom for a multiple number of passes.
Adding this option, the pecking tapping cycle option is also added.
Under the deep-hole tapping cycle, the tool is retracted to the R-point every time.
Select either the pecking tapping cycle or the deep-hole tapping cycle by parameter.
When the deep-hole tapping cycle is executed in the synchronous tapping mode, the synchronous tapping cycle option and deep-hole tapping cycle option are required.
When "depth of cut per pass Q" is designated in the block containing the G84 or G74 command in the state where the deep-hole tapping cycle is selected by parameter, the deep-hole tapping cycle is executed.
In the following cases, the normal tapping cycle is established.
When Q is not designated
When the command value of Q is zero

G84(G74) Xx1 Yy1 Zz1 Rr1 Qq1 Ff1(Ee1) Pp1 Ss1 ,Ss2 ,Ii ,Jj ,Rr2 ;

G84	: G84 forward tapping cycle
G74	: G74 reverse tapping cycle
Xx1,Yy1	: Hole drilling position
Zz1	: Hole bottom position
Rr1	: Point R position
Qq1	: Depth of cut per pass (designated as an incremental position)
Ff1	: Z-axis feed amount (tapping pitch) per spindle rotation
Ee1	: Tap thread number per 1-inch feed of Z axis
Pp1	: Dwell time at hole bottom and point R return
Ss1	: Rotation speed of spindle
,Ss2	: Rotation speed of spindle during retract
,Ii2	: In-position width of positioning axis
,Jj2	: In-position width of hole drilling axis
,Rr2	: Synchronization method selection (r2=1 synchronous, r2=0 asynchronous)

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



※

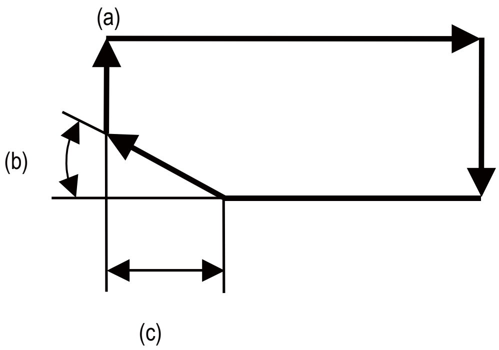
1. Clearance amount c : Retract amount (parameter)
2. This program is for the G84 command.
   The spindle forward rotation (M3) and reverse rotation (M4) are reversed with the G74 command.

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

4.5.4 Chamfering

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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



(a) Thread cutting cycle
(b) Chamfer angle
(c) Chamfer amount

4.5.6 Circular Thread Cutting

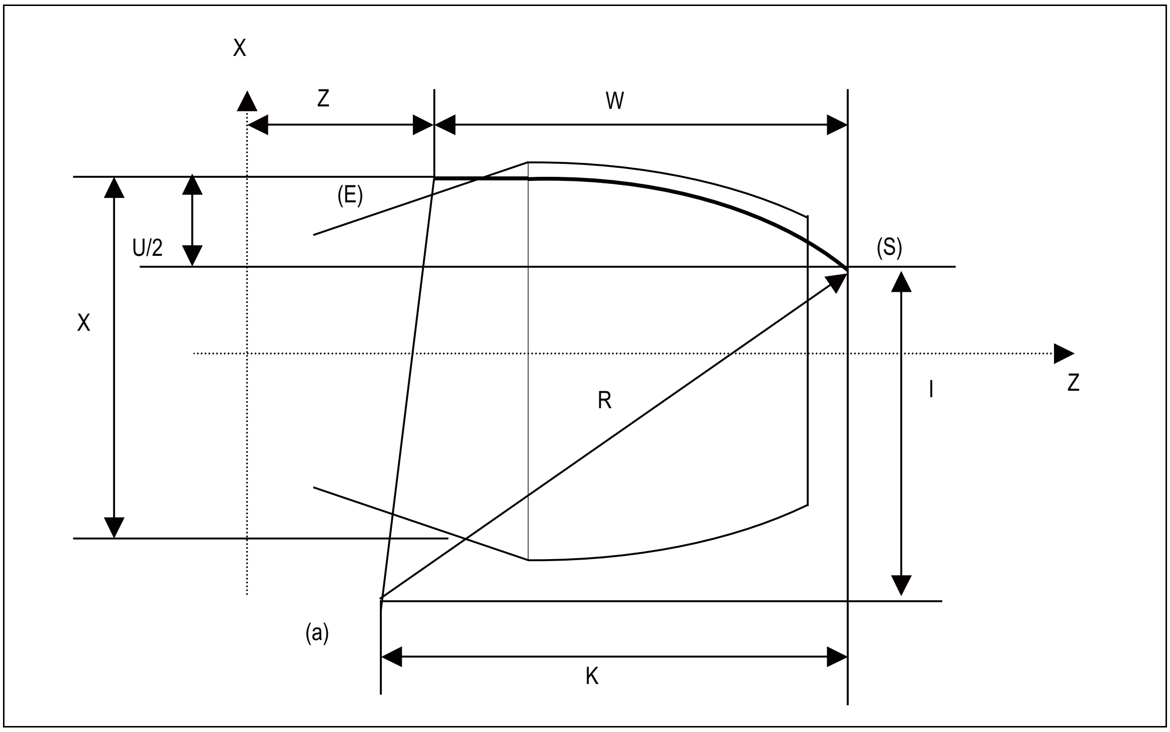
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

Circular thread in which the lead is in longitudinal direction can be cut.
This function can be used with the G code list 6 or 7.

Command format

G35(G36) Xx/Uu Zz/Ww { Ii Kk / Rr } Ff/Ee Qq ;

G35	: Clockwise (CW)
(G36)	: Counterclockwise (CCW)
Xx/Uu	: X-axis arc end point coordinate
Zz/Ww	: Z-axis arc end point coordinate
Ii	: X-axis arc center (incremental position of arc center as referenced from start point)
Kk	: Z-axis arc center (incremental position of arc center as referenced from start point)
Rr	: Arc radius
Ff/Ee	: Longitudinal axis (axis with most travel) direction lead (Ff: normal lead thread cutting, Ee: precise lead threads or inch threads)
Qq	: Thread cutting start shift angle (0.000 to 360.000°)

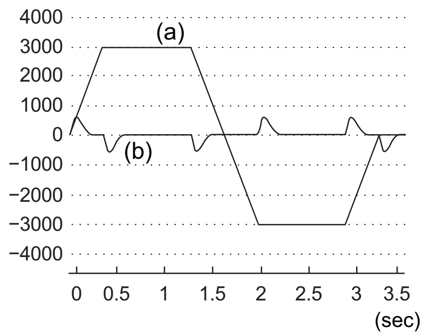


(a) Center
(E) End point
(S) Start point

4.5.8 High-speed Synchronous Tapping (OMR-DD)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The servo axis directly detects and compensates of the spindle's delay in tracking by using the communication between drive unit over the high-speed optical servo network. By minimizing the synchronization error, the accuracy of the synchronous tapping is increased.

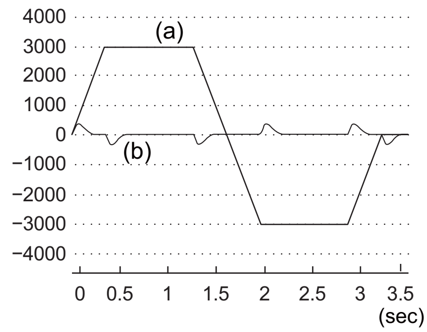


Conventional (without OMR-DD control)

(a) Spindle rotation speed

(b) Synchronization error

Implement to increase the accuracy of the synchronous tapping



MDS-D/DH Series (with DMR-DD control)

(a) Spindle rotation speed

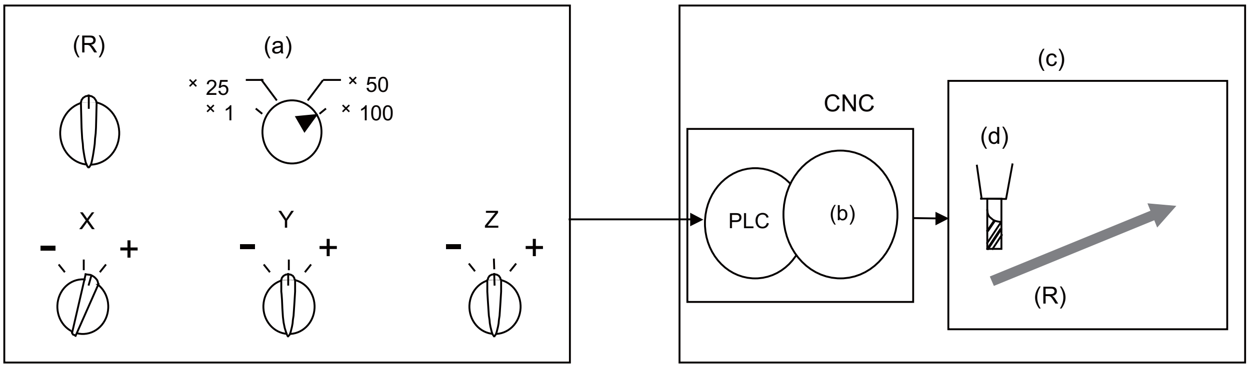
(b) Synchronization error

4.6 Manual Feed

4.6.1 Manual Rapid Traverse

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

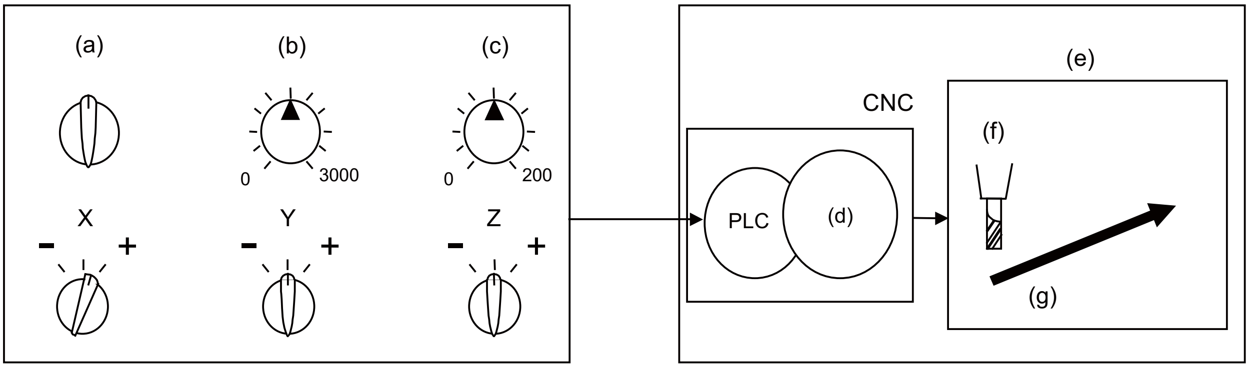


(a) Rapid traverse override
(b) Axis movement control
(c) Machine tool
(d) Tool
(R) Rapid traverse

4.6.2 Jog Feed

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When the jog feed mode is selected, the tool can be moved in the axis direction (+ or -) in which the machine is to be moved at the per-minute feed.
The jog feed rate can be set for each part system respectively.

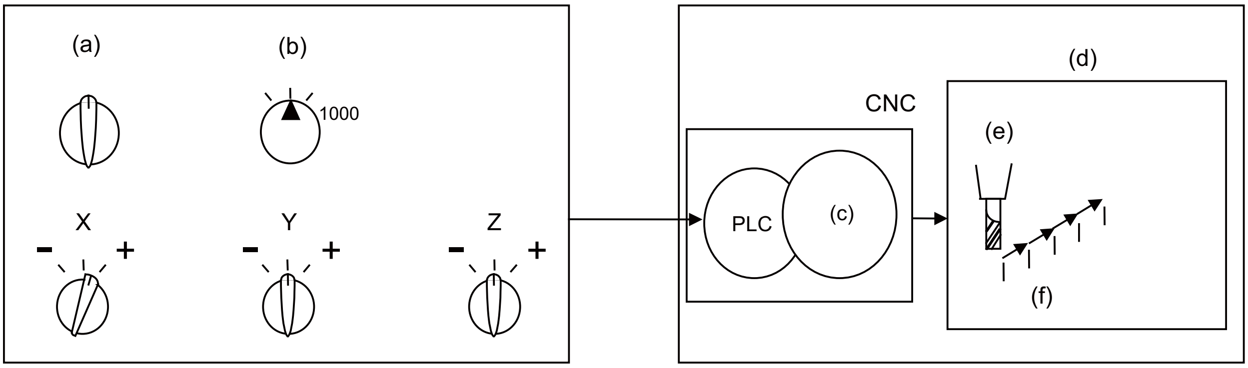


(a) Jog
(b) Feed rate
(c) Override
(d) Axis movement control
(e) Machine tool
(f) Tool
(g) Manual cutting feed

4.6.3 Incremental Feed

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



(a) Incremental
(b) Scale factor
(c) Axis movement control
(d) Machine tool
(e) Tool
(f) Step feed

4.6.4 Handle Feed

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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.
If the least command increment is 10nm or 1nm, the scale can be selected from X5000, X10000 or X100000, as well.
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.

Up to three handles can be used with the Mitsubishi CNC.

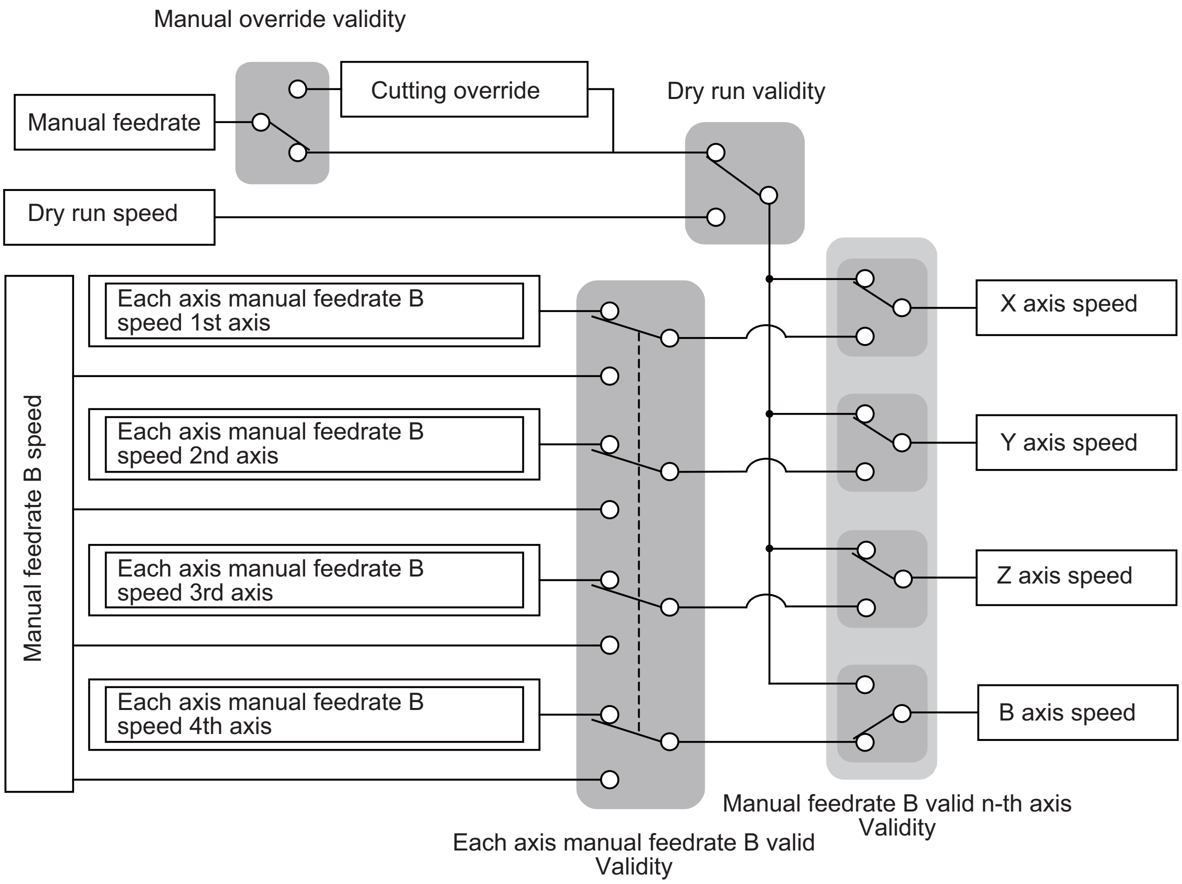
4.6.5 Manual Feed Rate B

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

"Manual feedrate B" is a function that sets an arbitrary axis feedrate from the user PLC separately from the "manual feedrate". The "manual feedrate B" feedrate setting can be selected from the feedrate common for all axes and the feedrate independent of reach axis.
By combining the "manual feedrate B" function with the manual/automatic simultaneous function, an arbitrary axis can be moved at the "manual feedrate B" independently of the machining program operation even during automatic operation. Similarly, if the jog mode and other manual operation mode are set simultaneously, an arbitrary axis can be moved at a speed independent from the "manual feedrate" even during the manual operation mode.

The "manual feedrate B" function can move an axis at a speed different from the "manual feedrate". This is not affected by dry run, or by manual or cutting override, so an arbitrary axis can be moved independently even in operations during automatic operation or override during manual axis movement.

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



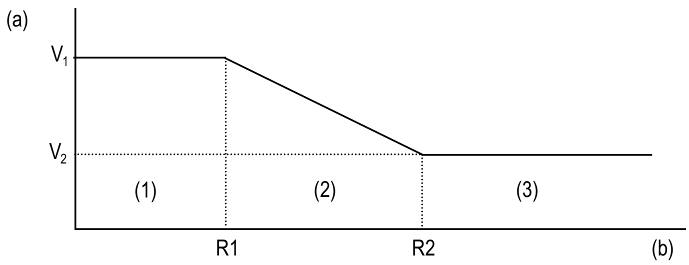
(Note) For the axis to which manual feedrate B is applied, the feedrate is not displayed on the screen.

4.6.6 Manual Feed Rate B Surface Speed Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

When using the manual feed rate B function and machining by moving the orthogonal axis while rotating the rotary table, the tool nose and workpiece's relative speed will drop as the tool nears the rotation center if the table rotation speed remains under the set conditions. This function controls the table rotation speed according to the distance from the rotation center.

As shown below, the distance (radius) from the rotation center at two points and the rotary axis speed at each point is set in the parameters. When the "manual feed rate B constant surface speed control valid" signal is turned ON, the rotary axis speed will be automatically calculated according to the current radius R.



(a) Rotary axis speed
(b) Radius

(1) If R ≦ R1, then V1 will be applied.
(2) If R1<R<R2, the speed V is calculated with the following expression.

$$V=\frac{(V_2 - V_1)}{(R_2 - R_1)} *(R - R_1)+V_1$$

(3) If R2 ≦ R, then V2 will be applied.

Override can be applied in the range of 0 to 200% in respect to the rotary axis's speed for which the manual feed rate B surface speed control is valid.

This function can be used with a rotary axis for which the manual feed rate B function is valid.
The manual feed rate B speed and each axis' manual feed rate B speed which are issued from the user PLC is ignored for an axis for which this function is valid.

4.7 Dwell

4.7.1 Dwell (Time-based Designation)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(1) M system

G04 Xx1 ; or G04 Pp1 ;	
G04	: Dwell
Xx1,Pp1	: Dwell time

The time-based dwell can be designated in the range from 0.001 to 99999.999 seconds.
(The input command increment for the dwell time depends on the parameter.)

(2) L system

(G94) G04 Xx1/Uu1 ; or (G94) G04 Pp1 ;	
G94	: Asynchronous
G04	: Dwell command
Xx1,Uu1,Pp1	: Dwell time

The time-based dwell can be designated in the range from 0.001 to 99999.999 seconds. (The input command increment for the dwell time depends on the parameter.)
U address can be used in the L system only.

5

Program Memory/Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory, DS, and external memory device (front IC card, HD disk, etc.).
When using devices such as HD, FD, DS and memory card, mass-editing, which is carried out on those devices, is possible.
The data size that can be handled in the mass-editing differs depending on the devices.

5.1.1 Memory Capacity (Number of Programs Stored)

(Note) The tape length for the multi-part system specifications is the total for all part systems.

5.1.1.1 15kB [40 m] (64 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

5.1.1.2 30kB [80 m] (128 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.3 60kB [160 m] (200 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.4 125kB [320 m] (200 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.5 230kB [600 m] (400 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.6 500kB [1280 m] (1000 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.7 1000kB [2560 m] (1000 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.1.1.8 2000kB [5120 m] (1000 Programs)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

5.2 Editing

5.2.1 Program Editing

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The following editing functions are possible.

(1) Program erasing

(a) Machining programs can be erased individually or totally.

(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.

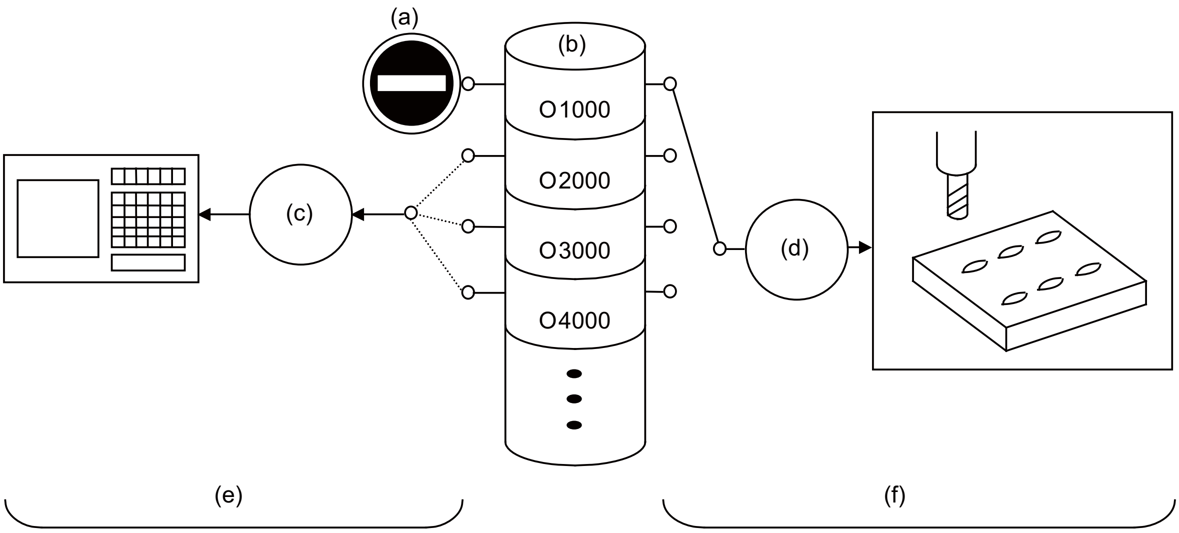
(5) Mass editing

With mass-editing, up to 20MB can be editing for FCU7-DA2-xx, or up to 1GB can be editing for FCU7-DA3-xx/DA4-xx.
The specification and restrictions are different from those with the regular editing.
Mass-editing is applied when all of the following conditions are satisfied.
- When the storage destination for the program to be opened is either HD, FD, or memory card.
- When a file size is 1.0MB or larger. (The size could be 2.0MB or larger, depending on the parameter settings.)

5.2.2 Background Editing

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



(a) Prohibited
(b) Program registered in memory
(c) Editing
(d) Memory operation
(e) Program editing
(f) Machining with memory operation

(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.

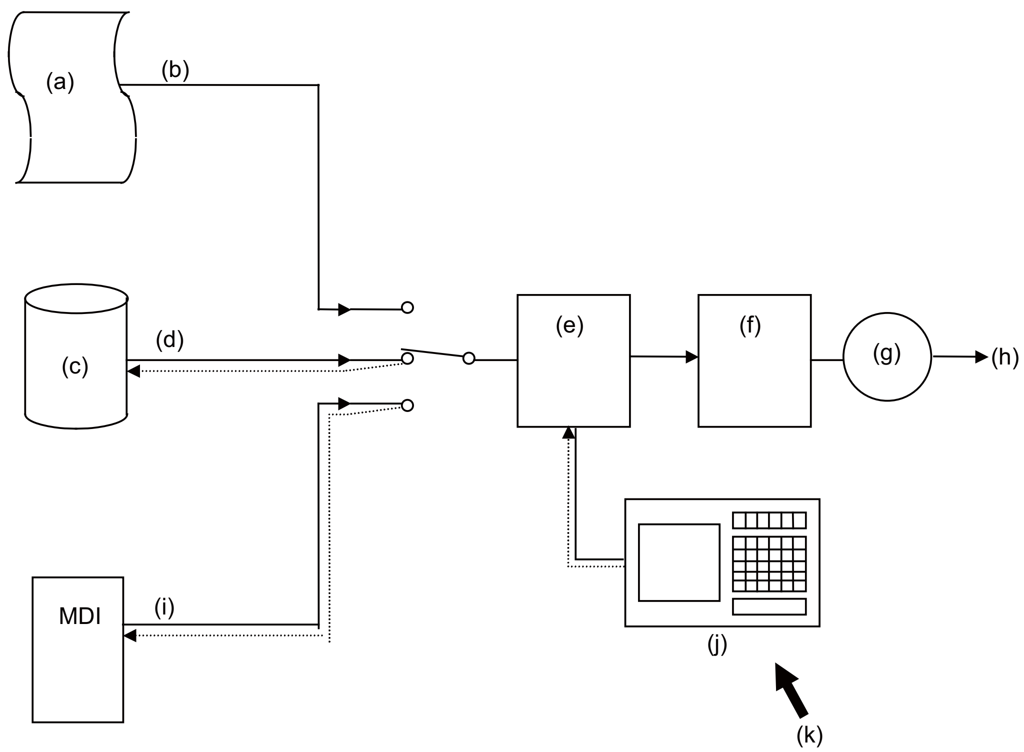
(2) The editing functions mentioned in the preceding section can be used at any time for machining programs which are not being used for memory operation.
This makes it possible to prepare and edit the next program for machining, and so the machining preparations can be made more efficiently.

(3) The machining program will not be searched as the operation target even when searched in the edit screen.

5.2.3 Buffer Correction

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



(a) Tape
(b) Tape mode
(c) HD, DS, Memory
(d) Memory mode
(e) Pre-read block
(f) Execution block
(g) NC operation
(h) Machine control
(i) MDI mode
(j) Setting and display unit
(k) Buffer correction

6

Operation and Display

6.1 Structure of Operation/Display Panel

The setting and display unit is configured of the setting section and keyboard section.
Refer to "General Specifications" for details.

6.1.1 Color Display (8.4-type LCD TFT)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	□	□	□
L system	—	—	—	□	□	□

6.1.2 Color Display (10.4-type LCD TFT)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	□	□	□
L system	—	—	—	□	□	□

6.1.3 Color Display (10.4-type LCD TFT/WindowsXPe)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

6.1.4 Color Display (15-type LCD TFT/WindowsXPe)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

6.1.5 Color Touch-panel Display (10.4-type LCD TFT/WIndowsXPe)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

6.1.6 Color Touch-panel Display (10.4-type LCD TFT)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	□	□	□
L system	—	—	—	□	□	□

6.1.7 Color Touch-panel Display(15-type LCD TFT/WindowsXPe)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

6.2 Operation Methods and Functions

6.2.1 Operation Input

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

In addition to the method of directly inputting numeric data, a method to input the operation results using four rules operators and function symbols can be used for specific data settings.

Numeric values, function symbols, operators and parentheses ( ) are combined and set in the data setting area.
The operation results appear when the INPUTkey is pressed. If the INPUT key is pressed again, the data is processed and displayed on the screen. The contents in the data setting area are erased.

Examples of operator settings and results			Function symbols, setting examples and results			
Operation	Setting example	Operation results	Function	Function symbol	Setting example	Operation results
Addition	= 100+50	150.000	Absolute value	ABS	= ABS(50-60)	10
Subtraction	= 100-50	50.000	Square root	SQRT	= SQRT(3)	1.732
Multiplication	= 12.3*4	49.200	Sine	SIN	= SIN(30)	0.5
Division	= 100/3	33.333	Cosine	COS	= COS(15)	0.966
Function	= 1.2*(2.5+SQRT(4))	5.4	Tangent	TAN	= TAN(45)	1
			Arc tangent	ATAN	= ATAN(1.3)	52.431

6.2.2 Absolute/Incremental Value Setting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When setting the data, the absolute/incremental setting can be selected from the menu.

The absolute/incremental settings can be selected on the following screens.
- Common variable screen
- Tool compensation amount screen
- Coordinate system offset screen

6.2.5 Displayed Part System Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The part system displayed on the screen can be changed with the [$<->$] keys.
The number of displayed part systems is counted by one each time the [$<->$] keys are pressed. The screen corresponding to that part system opens.
If the number of displayed part systems exceeds the valid number of part systems, the number of displayed part systems will return to 1.

6.2.6 Menu List

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The menu list function displays the menu configuration of each screen as a list making it possible to directly select the menu for other screens.
When the cursor is moved to the menu, the outline of that menu's functions will also appear. The menu can be selected while checking the details of the menu.

6.2.7 Display Switch by Operation Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The screen display changes when the screen mode selection switch is changed.
The details corresponding to the operation mode are displayed.

6.2.8 External Signal Display Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The screen display changes with the signal from PLC.

6.2.10 Screen Saver

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The screen saver function protects the screen display unit by turning the backlight OFF after the time set in the parameters has elapsed. The backlight is turned OFF after a certain period of time (automatic change function) or after the key operations (manual change function).
The screen is displayed again by pressing any key, or by touching anywhere on the screen if the display unit carries a touch-sensitive screen.

6.2.11 Parameter/Operation Guidance

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The parameter/operation guidance function displays the details of the parameters or the operation methods according to the state of the screen currently being displayed. The operation guidance can also be selected from the Contents and displayed.

If the ? key is pressed on any screen, the parameter/operation guidance window will open. If a pop-up window other than the parameter/operation guidance window is opened, the parameter/operation guidance window will open over the currently opened pop-up window.

6.2.12 Alarm Guidance

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Guidance is displayed for the alarm currently issued.
By utilizing the guidance information, identify the cause from possible factors and determine the countermeasures.
When a multiple number of alarms are issued at the same time, guidance will be displayed for all the alarm issued.

6.2.13 Machining Program Input Mistake Check Warning

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

If an illegal input is found in the decimal point after the current cursor position, the cursor will move to that position, and a warning message will appear.
If this function is ON when editing the program, the decimal point will be checked for the block each time an edit key (alphabet, number, symbol, Delete, etc.) is pressed. The block is not checked when the cursor keys or page feed keys are pressed.
The warning for illegal machining program can also be issued while editing an MDI program.
The comment block is also subject to the warning for illegal machining program.

A warning does not appear in the following cases.

(1) When the data in the address subject to the illegal decimal point input is "0", a warning will not be issued regardless of whether there is a decimal point or not. (Example: A warning is not issued for "X0".)
(2) When the data in the address subject to the illegal decimal point input is omitted, a warning will not be issued. (Example: A warning is not issued for "G28XYZ".)
(3) Blocks containing "[" or "]" are not subject to the warning for illegal machining program.

6.2.15 Screen Capture

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	–	–	–	○	○	○
L system	–	–	–	○	○	○

This function allows to output a bitmap file of a screen displayed on the setting and display unit.

6.2.16 User Selectable Menu Configuration

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function allows to change the display order of operations, procedure and edit screen, and to change display/non-display selection.

6.2.17 PC-NC Network Automatic Connection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	―	―	―
L system	○	○	○	―	―	―

This function supports to restore the connection when the network connection cannot be created between the display unit and the control unit.
When the connection is not established even after the time out has expired, the connectable control unit IP address list appears. From the IP address list, the network connection can be re-established and restored when the IP address, which is to be connected with the control unit, is selected.

6.2.18 Device Open Parameter

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function can set or change the user backed up area of the PLC device from the NC screen.
The following settings are available from the NC screen.

- Divide the device area accordance with the specifications of the machine maker and set and display for each divided area.
- Switch the display format or data type for each divided area.

6.2.19 SRAM Open Parameter

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function can set or change the SRAM open area for the machine maker from the NC screen.
The following settings are available from the NC screen.

- Divide the SRAM area accordance with the specifications of the machine maker and set and display for each divided area.
- Switch the display format or data type for each divided area.

6.3 Display Methods and Contents

6.3.1 Status Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The status of the program now being executed is indicated.

(1) Display of G, S, T, M commands and 2nd miscellaneous command modal values
(2) Feed rate display
(3) Tool compensation No. and compensation 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 Clock Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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.

6.3.3 Operation Screen Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Various information related to operation, such as the axis counter, speed display and MSTB command are displayed on the Monitor screen. The following operations regarding operation can be executed.

(1) Operation search
(2) Restart search
(3) Editing of searched machining program
(4) Trace (Display of machine movement path)
(5) Check (Display of NC program's tool movement path)
(6) Correction of running program's buffer
(7) Counter set
(8) Manual numeric command, etc.

6.3.4 Preparation Screen Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Tool/workpiece related settings, user parameter settings, MDI editing, counter setting, manual numeric command issuing and pallet program registration (option) can be carried out on the Preparation screen.

6.3.5 Edit Screen Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Machining program editing (addition, deletion, change) and checking, simple program creation, and machining program input/output can be carried out on the Edit screen.

6.3.6 Diagnosis Screen Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The following operations related to the CNC diagnosis can be carried out on the Diagnosis screen.
(1) Display of hardware and software configuration
(2) Display of CNC options
(3) Diagnosis of PLC interface
(4) Display of drive unit information
(5) Display of alarm message / alarm history list etc.

6.3.7 Maintenance Screen Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Parameter setting and display, and NC data input/output, etc., can be carried out on the Maintenance screen.

6.3.8 Additional Language

6.3.8.1 Japanese

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

6.3.8.2 English

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

6.3.8.3 German

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.4 Italian

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.5 French

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.6 Spanish

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.7 Chinese

6.3.8.7.1 Chinese (Traditional Chinese Characters)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.7.2 Chinese (Simplified Chinese Characters)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.8 Korean

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.9 Portuguese

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.10 Hungarian

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.11 Dutch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.12 Swedish

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.13 Turkish

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.14 Polish

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.15 Russian

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

6.3.8.16 Czech

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □
L system	△ / □	△ / □	△ / □	△ / □	△ / □	△ / □

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.

7.1.1 Machining Program Input / Output (Including User Macros and Fixed Cycle Macros)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.1.2 Tool Offset Data Input / Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.1.3 Common Variable Input / Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.1.4 Parameter Input / Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.1.5 History Data Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.1.7 System Configuration Data Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

7.2 Input/Output I/F

7.2.1 RS-232C I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

There are 2 ports (port 1/2) available with the RS-232C interface for both display unit and control unit.

	Display unit	Control unit
Port	Port 1/2	Port 1/2
Baudrate	Up to 19.2kbps	
Handshake method	DC code method, RTS/CTS method possible	

Each port can be used for the following application.

<Display unit>

Port 1: Input/output

Port 2: Input/output

<Control unit>

Port 1: Input/output, Tape operation, Anshin-net, Machine tool builder network system

Port 2: Input/output, Tape operation, GX Developer communication, computer link, Anshin-net, Machine tool builder network system, handy terminal

7.2.2 IC Card I/F

7.2.2.1 I/F for CF Card in Control Unit [Up to 2GByte]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	—	—	—
L system	○	○	○	—	—	—

Interface card to use CF card can be attached inside the NC control unit and used.

7.2.2.2 Front IC Card I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Interface card to use PCMCIA card can be attached in front of the NC control unit and used.

7.2.3 Ethernet I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Ethernet interface card can be attached onto the NC unit and used.

7.2.4 Hard Disk I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	—	—	—
L system	△	△	△	—	—	—

A hard disk drive can be mounted and used.

7.2.6 USB Memory I/F [Up to 2GByte]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	○	○	○
L system	△	△	△	○	○	○

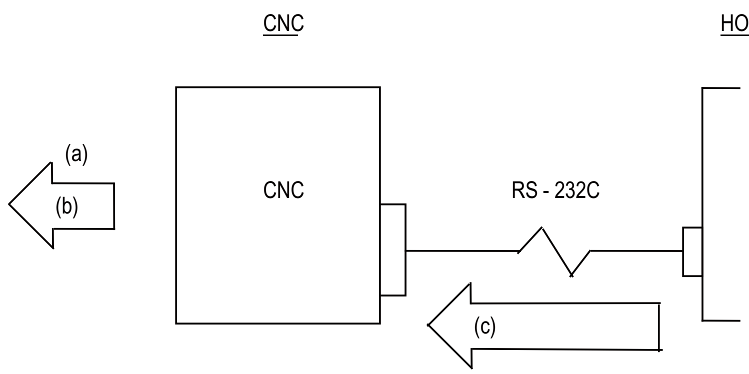
A USB memory can be mounted.

7.3 Computer Link

7.3.1 Computer Link B

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Computer link B is a function that passes the data between host computer (hereafter abbreviated to "HOST") and CNC.
This function sends [DC1] to the HOST at the CNC cycle start, and it enables operation to be performed while the machining programs are received from the HOST.
The computer link has a reception buffer so that operation will be less susceptible to the effects of the data transfer status at the HOST end.
The high-speed machining mode option is required for high-speed fine-segment machining.
This function cannot be operated in the 2nd and following part systems.



(a) Operation
(b) Command
(c) Machining program

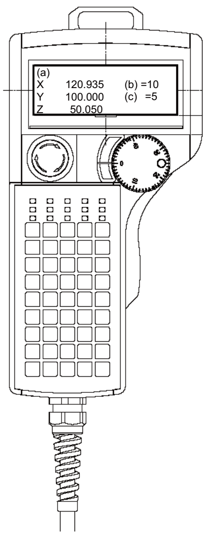
7.4 Others

7.4.1 Handy Terminal Connection

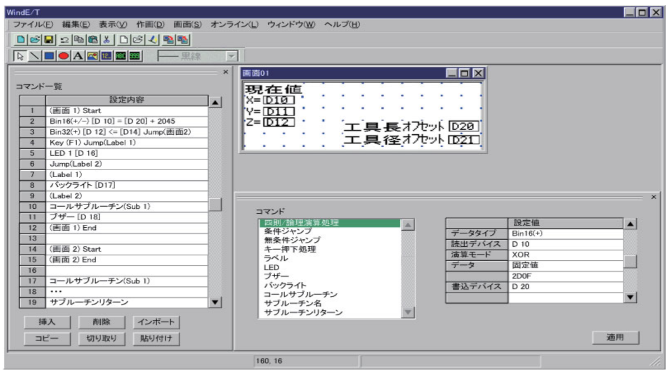
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	—	—	—
L system	△	△	△	—	—	—

Machine operations, such as setup operations, are possible at hand by using a handy terminal.

(1) Machine operation using manual pulse generator, jog and inching
(2) Displaying CNC status such as machine position
(3) Displaying PLC messages
(2 languages x 20 one-byte characters x 256 messages can be stored.)
(4) Emergency stop button
(5) Screen display covering 4 lines x 20 characters (64 dots lengthwise x 192 dots widthwise)
(6) Operation valid or invalid switch for 3 positions (OFF-ON-OFF)
(7) Setting for tool compensation, workpiece coordinates, etc.
(8) Creating display data using NC Designer (handy terminal)



Handy terminal



NC Designer Handy terminal

(Note 1) CNC outputs the data received from the handy terminal to the PLC interface.
CNC transmits the data of which PLC sets in the PLC interface to the handy terminal.
Thus, the machine tool builder must create a PLC program corresponding to the handy terminal.
(Note 2) Before using the handy terminal, it is necessary to customize the display area configuration, key input, communication conditions with CNC, etc.
(Note 3) This function cannot be used in combination with the serial GPP communication.

8

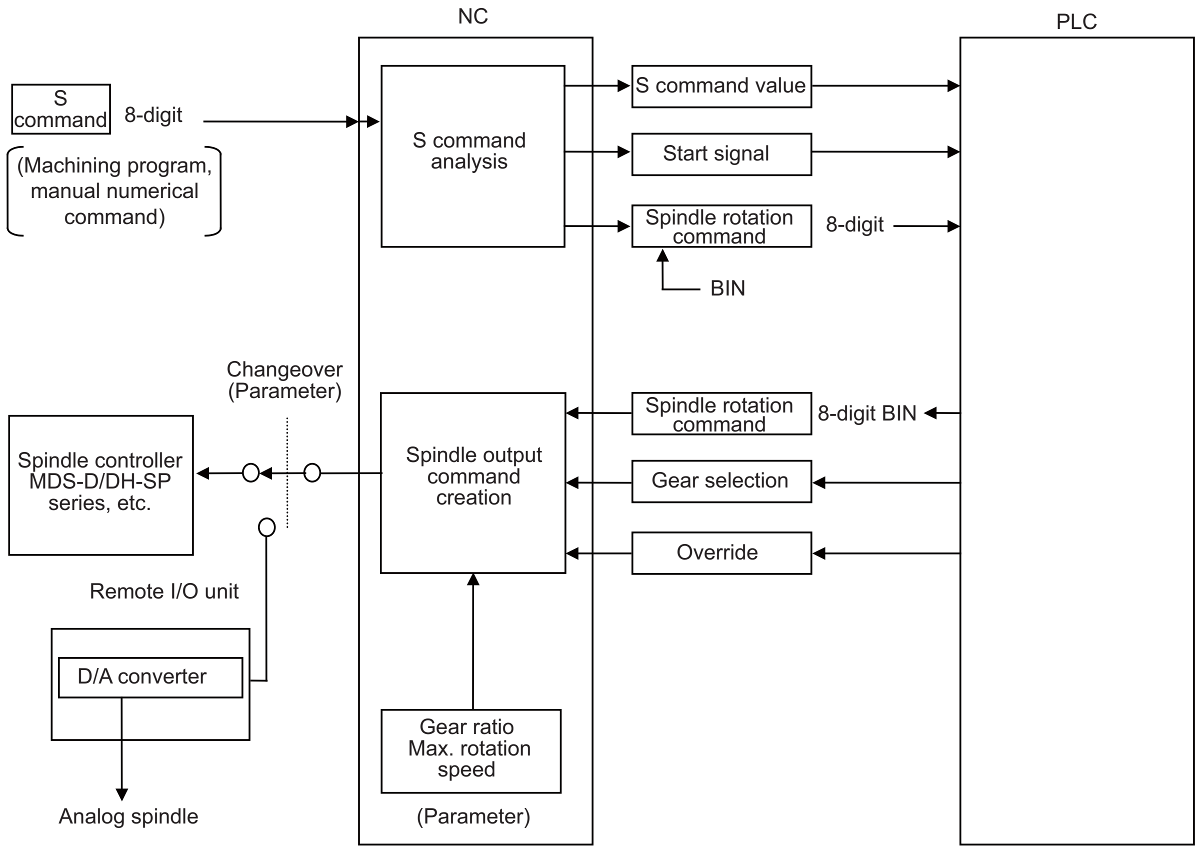
Spindle, Tool and Miscellaneous Functions

8.1 Spindle Functions (S)

8.1.1 Spindle Control Functions

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 (S0 to S±99999999) is commanded, a signed 32-bit binary data and start signal, or a non-signed 32-bit binary data and start signal will be output to the PLC.
Up to 1 set of S commands can be commanded in one block.
Processing and complete sequences must be incorporated on the PLC side for all S commands.



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

8.1.1.1 Spindle Digital I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This interface is used to connect the digital spindle (AC spindle motor and spindle driver).

8.1.1.2 Spindle Analog I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Spindle control can be executed using an analog spindle instead of the digital spindle.

8.1.1.3 Coil Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

8.1.1.4 Automatic Coil Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

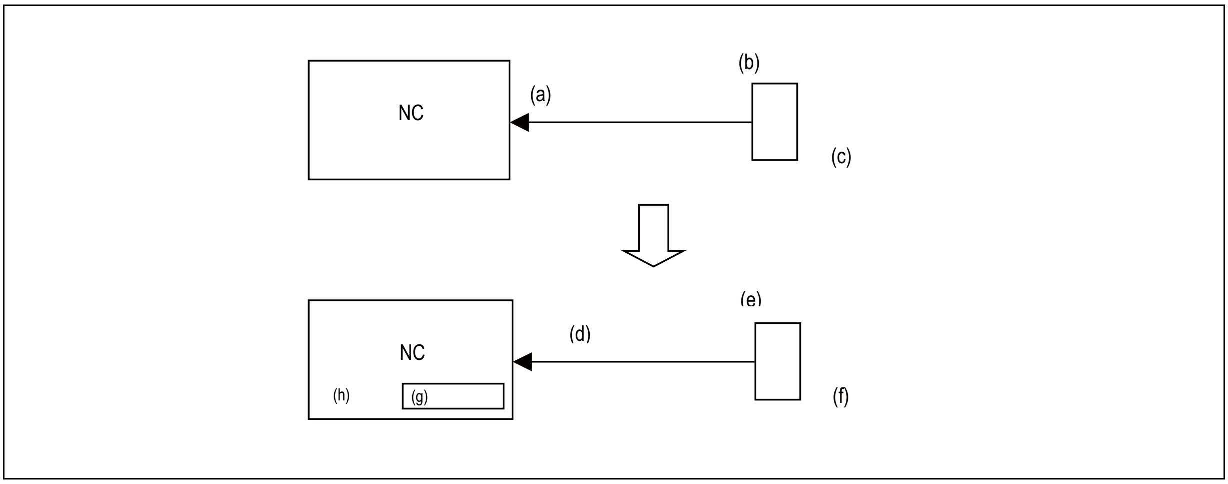
Constant output characteristics can be achieved across a broad spectrum down to the low-speed range by switching the spindle motor connections.
This is a system under which the NC unit switches the coils automatically in accordance with the motor speed.

8.1.1.5 Encoder Input I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Encoder pulse input used to be fixed to 1024 pulse input on the conventional analogue interface. With this function, arbitrary pulse can be input by parameters set in R register.
The maximum number of input pulse is 76800. Synchronous feed function can be activated with the arbitrary pulse if the number of pulse to be used is set in R register and the external signal is turned ON.

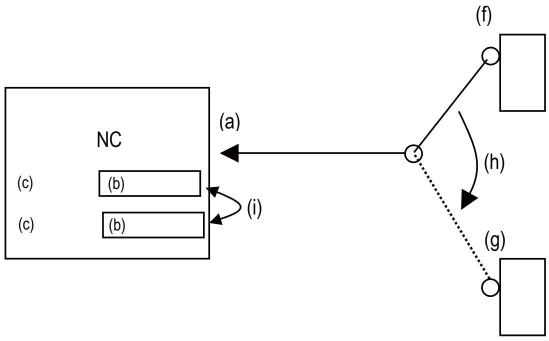
Encoder arbitrary pulse input



(a) ENC interface
(b) Encoder
(c) 1024 pulse fixed
(d) ENC interface
(e) High resolution Encoder
(f) Max76800 pulse
(g) Input pulse
(h) R register

Two kinds of encoders can be switched over and connected. Encoder switch-over is possible, using PLC device.

Encoder changeover pulse input



(a) ENC interface
(b) Input pulse
(c) R register
(f) Encoder1 (High resolution)
(g) Encoder2 (Low resolution)
(h) Switch with ladder
(i) Switch internally

8.1.2 S Code Output

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When an 8-digit number following address S (S0 to S±99999999) is commanded, a signed 32-bit binary data and start signal, or a non-signed 32-bit binary data and start signal will be output to the PLC.
One set of S commands can be issued in one block.
Processing and complete sequences must be incorporated on the PLC side for all S commands.

S function can be designated with any other kind of commands. In the case where a movement command is in the same block, two different command sequences are available. Depending on user PLC specifications, either one of the following two will be applied.

(1) S function is executed after the movement is completed.
(2) S function is executed at the same time as when the movement command is issued.

8.1.3 Constant Surface Speed Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

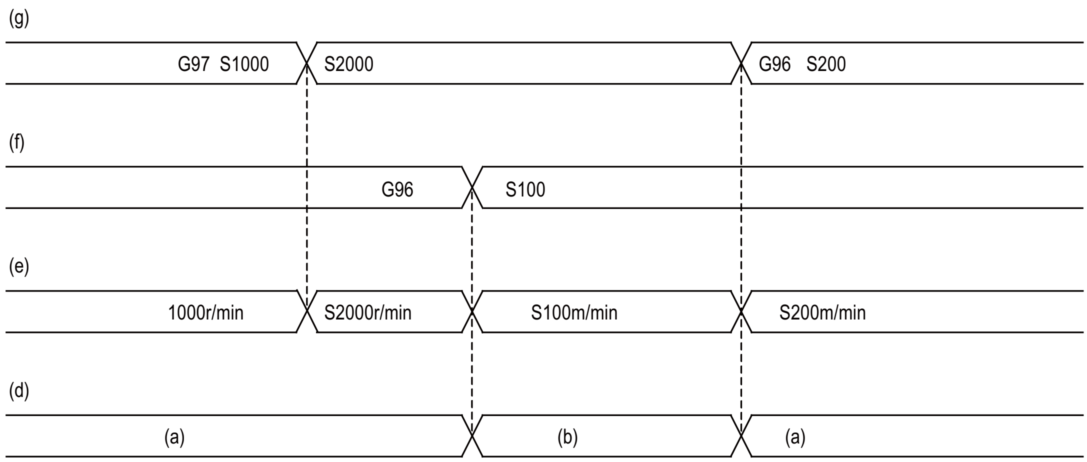
With radial direction cutting, this function enables the spindle speed to be changed in accordance with changes in the radial direction coordinates and the workpiece to be cut with the cutting point always kept at a constant speed (constant surface speed).

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 1) Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine or may cause damage to the operators. Thus make sure to use this control while the "spindle speed clamp" is enabled. When the constant surface speed control is commanded, keep enough distance from the program zero point.

(Note 2) 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 timing synchronization function.
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.



(a) Part system 1
(b) Part system 2
(d) Spindle control rights
(e) Spindle speed
(f) Part system 2 program
(g) Part system 1 program

8.1.4 Spindle Override

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

(2) Type 2 (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%.

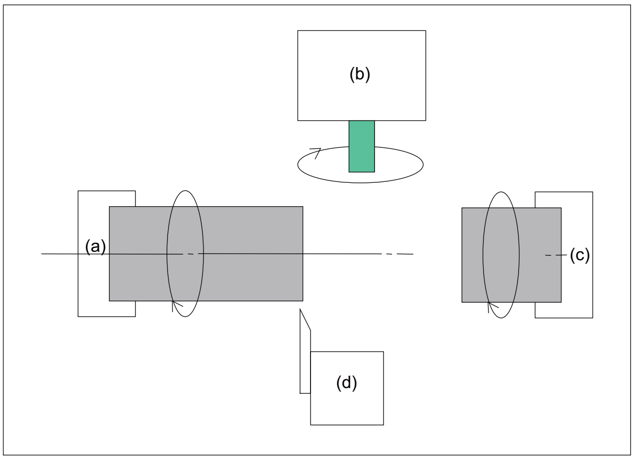
(Note 1) Selection between type 1 and type 2 can be designated by user PLC processing.

8.1.5 Multiple-spindle Control

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

Multiple-spindle control I (L system only)	Control based on a spindle selection command (such as G43.1) and spindle control command ([S*****;] or [SO=*****;]), etc.
Multiple-spindle control II	Control based on an external signal (spindle command selection signal, spindle selection signal) and spindle control command ([S*****;] only), etc. Spindle selection command and control command of [SO=*****;] cannot be used.

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



(a) First spindle
(b) Tool spindle (third spindle)
(c) Second spindle
(d) Tool post 1

8.1.5.1 Multiple-spindle Control I

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

(1) Spindle selection command

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 fourth spindles to which the subsequent S command (S*****) is to apply.

Command format

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

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

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

Command format

S ○ =*** ;**	
○	:Number assigned as the spindle number (1: first spindle; 2: second spindle; 3: third spindle; 4: fourth spindle); variables can be designated.
*****	: Rotational speed or surface speed value assigned by 5-digit analog command; variables can be designated.

8.1.5.2 Multiple-spindle Control II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

Spindle command selection, spindle selection

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

8.1.6 Spindle Orientation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function stops the spindle rotation at a certain position.
When Z-phase has not yet been passed, pass Z-phase twice (excluding reciprocations within one rotation), and then position to the orientation position.
When Z-phase has been passed already, immediately position to the orientation position.
The spindle does not rotate if Z-phase has been passed and the orientation position is already established when orientation command is issued.
When the spindle has been rotating in the direction opposite from that of orientation rotation when the orientation command is issued, orientation operation will be executed after decelerating to stop.

(a) Orientation

This function stops the spindle rotation at a certain position when using the digital spindle.
When the orientation command is used, the spindle will rotate several times and then stop at the orientation point.
The orientation position differs depending on the detector.

- When the encoder orientation (PLG and external encoder/ring sensor) is used:
  …At the Z-phase position
- When the magnetic sensor (proximity switch) is used:
  …At the magnetic sensor installation position

(b) Multi-point orientation

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 35999. (Unit: 360°/36000=0.01°)
(Note 1) Multi-point orientation cannot be executed when using the magnetic sensor.
(Note 2) Orientation is possible only when the gear ratio is 1:1 for the PLG orientation.
(The orientation is completed at the PLG encoder's Z-phase, so when using reduction gears, the orientation points will be generated at several points during one spindle rotation.)

(c) Orientation imposition advance output

This function turns the spindle in-position signal ON as soon as the spindle reaches within the second in-position width. Then, the spindle 2nd in-position signal is turned ON as soon as the spindle reaches within the in-position width.
Since orientation completion can be predicted using this function, it is possible to eliminate the sequence delay time, etc. for tool changes and other such operations, thereby achieving a faster tact time.

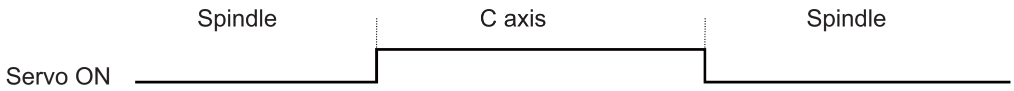
(d) Proximity switch orientation

Proximity switch orientation enables the spindle positioning by providing the proximity switch position installed on the spindle side as the spindle position zero point.
When the spindle motor and the spindle are connected with V-belt, the spindle position zero point calculated from the spindle motor position varies due to belt slipping, etc. Thus, the spindle position zero point must be detected from the proximity switch signal every time orientation or zero point return is carried out.
In the conventional spindle position zero point detection method, the spindle position zero point is calculated from the spindle drive unit and transmitted to NC when the position detector detects the one-time rotation signal (Z-phase) for the first time after the system has been turned ON, and the zero point will not be changed thereafter.
In the case of proximity switch signal orientation, NC updates the spindle position zero point every time orientation or zero point return is carried out.

8.1.7 Spindle Position Control (Spindle/C Axis Control)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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



At servo OFF: ---------- Spindle (C axis cannot be controlled).

At servo ON: ----------- C axis (spindle cannot be controlled).

- Reference position return state
  Reference position return is incomplete when the Z phase has not been passed.
  Reference position return is complete when the Z phase has been passed.
- C axis potion data
  The NC's internal C axis position data is updated even for the spindle rotation during spindle control.
  The C axis coordinate position counter is held during spindle control, and is updated according to the amount moved during spindle control when the C axis servo READY is turned ON. (The C axis position at servo ON may differ from the position just before the previous servo OFF.)

8.1.8 Spindle Synchronization

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). This function can be assigned by G code or by PLC.
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.

8.1.8.1 Spindle Synchronization I

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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

Command format

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

```
G113 ;
```

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

```
G114.1  Hh1  Dd1  Rr1  Aa1 ;
Hh1      :Selects the basic spindle.
Dd1      :Selects the spindle to be synchronized with the basic spindle.
Rr1      :Designates the synchronized spindle phase shift amount.
Aa1      :Designates the spindle synchronization acceleration/deceleration time constant.
```

8.1.8.2 Spindle Synchronization II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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

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

8.1.9 Tool Spindle Synchronization I (Polygon)

8.1.9.1 Tool Spindle Synchronization IA (Spindle-Spindle Polygon)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

With a machine equipped with two or more spindles under serial connection control, this function enables spindle-spindle polygon machining (IA) by controlling the workpiece spindle rotation in synchronization with the rotary tool spindle rotation. The rotary tool spindle and workpiece spindle are designated among the spindles subject to serial connection control.

Serial connection control for the workpiece spindle and rotary tool spindle can be performed by MDS-*-SP.
This function can be used with the G code list 2 to 5.

Command format

Spindle-spindle polygon machining mode command (G114.2)
This command establishes the spindle-spindle polygon machining mode (IA) in which two spindles are rotated in synchronization at two different speeds. It is required to designate the rotary tool spindle, workpiece spindle and the rotational ratios (number of rotary tool teeth and number of work angles) of the two designated spindles.

G114.2 Hh1 Dd1 Ee1 Ll1 Rr1 ;	
Hh1	: Selects the rotary tool spindle.
Dd1	: Selects the workpiece spindle.
Ee1	: Designates the rotary tool spindle rotational ratio.
Ll1	: Designates the workpiece spindle rotational ratio.
Rr1	: Synchronized spindle phase shift amount

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

G113 ;

8.1.9.2 Tool Spindle Synchronization IB (Spindle-Spindle Polygon)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

With a machine equipped with two or more spindles under serial connection control, this function enables spindle-spindle polygon machining (IB) by controlling the rotary tool spindle rotation in synchronization with the workpiece spindle rotation. The rotary tool spindle and workpiece spindle are designated among the spindles subject to serial connection control.
A parameter is used to switch from spindle-spindle polygon machining (IB) to spindle-NC axis polygon machining (IC) or vice versa.

Serial connection control for the workpiece spindle and rotary tool spindle can be performed by MDS-*-SP.
This function can be used with the G code list 6 and 7.

Command format

Spindle-spindle polygon machining mode ON (G51.2 or G251)
This command establishes the spindle-spindle polygon machining mode in which two spindles are rotated in synchronization at two different speeds. It is required to designate the rotary tool spindle, workpiece spindle and the rotational ratios (number of work angles and number of rotary tool teeth) of the two designated spindles.

G51.2 Hh1 Dd1 Pp1 Qq1 Rr1 ;	
Hh1	: Selects the workpiece spindle (spindle).
Dd1	: Selects the rotary tool spindle (spindle).
Pp1	: Designates the workpiece spindle rotational ratio.
Qq1	: Designates the rotary tool spindle rotational ratio.
Rr1	: Synchronized spindle phase shift amount

Spindle-spindle polygon machining cancel command (G50.2)

G50.2 ;

8.1.9.3 Tool Spindle Synchronization IC (Spindle-NC Axis Polygon)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

This function controls so that the workpiece (spindle) and tool (NC axis) synchronously rotate at the commanded ratio and allows polygon machining. Spindle-spindle polygon machining (IB) and spindle-NC axis polygonal machining (IC) can be switched by the parameter.
This function can be used with the G code list 6 and 7.

Command format

Polygon machining mode ON (G51.2 or G251)

G51.2 Pp1 Qq1 ;	**Spindle synchronization start (Polygon machining mode start)**
Pp1 Qq1	:Workpiece axis selection (spindle) P: Spindle rotational ratio Q: Rotary tool axis rotational ratio Rotation direction is specified with a sign. +: Forward run / -: Reverse run

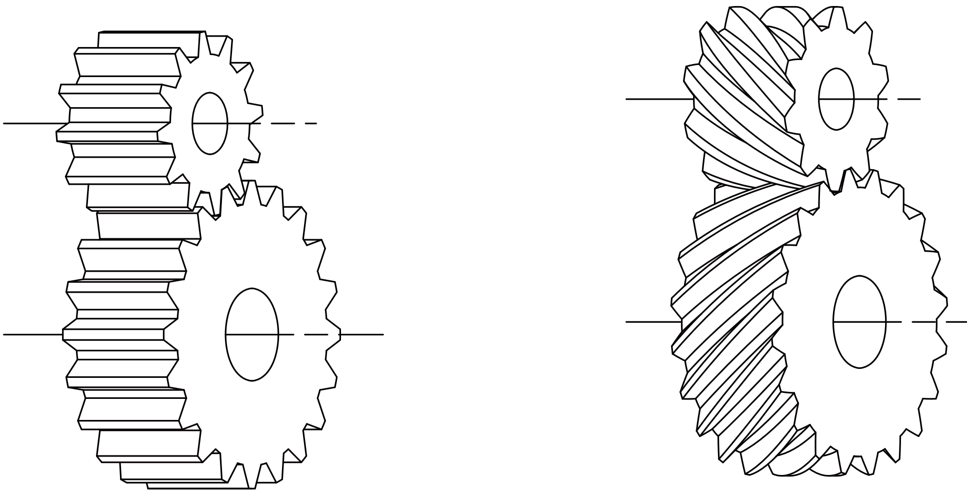
Polygon machining mode OFF (G50.2)

G50.2 ;	**Spindle synchronization cancel (Polygon machining mode cancel)**

8.1.10 Tool Spindle Synchronization II (Hobbing)

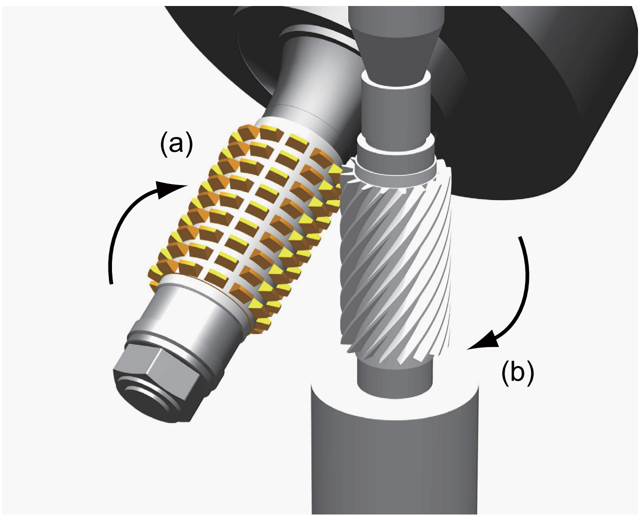
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

This function is to cut the gear with hob (hob cutter).
A spur gear can be machined by synchronizing and rotating the hob axis and the workpiece axis in a constant ratio.
A helical gear can be machined by compensating the workpiece axis according to the gear torsion angle for the Z axis movement.



Spur gears Helical gears

By synchronizing and rotating the hob axis and the workpiece axis in a constant rotation ratio, a gear is machined so that the cutter is engaged with gear.



(a) Hob
(b) Gear
Hob axis : Rotary tool axis on which a hob is mounted.
Workpiece axis : Rotary axis on which a workpiece is mounted.

8.1.11 Spindle Speed Clamp

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function is to limit the spindle rotation speed to the ranges between the maximum rotation speed and the minimum rotation speed.
This function can be set by a parameter or a program.

G92 S__ Q__ ;	**Spindle clamp speed setting**
S	: Maximum clamp rotation speed
Q	: Minimum clamp rotation speed

An alarm will occur if the constant surface speed control is conducted without issuing the spindle clamp speed command (G92).

8.2 Tool Functions (T)

8.2.1 Tool Functions (T Command)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The command is issued with an 8-digit number following address T (T0 - T99999999). The tool function is used to command the tool No. In the lathe specification controller, the tool compensation (tool length compensation, tool nose wear compensation) Nos. are also indicated.

(1) M system

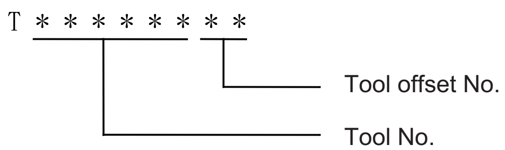
Tool function, or T function, is used to designate the tool No. and tool compensation No.
This can be designated with an 8-digit number following address T (0 to 99999999). Up to four sets of T commands can be issued in one block. Note that the number of T commands to be issued within the same block is determined by parameter.
BCD output or binary output can be selected by parameter.
Output signal is 8-digit BCD code and start signal, signed 32-bit binary data and start signal, or non-signed 32-bit binary data and start signal.
T function can be designated with any other kind of commands. In the case where a movement command is in the same block, two different command sequences are available. Depending on user PLC specifications, either one of the following two will be applied.

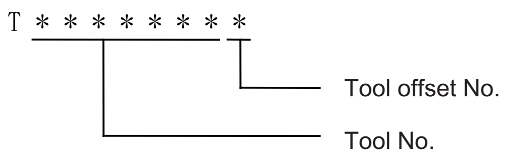
(a) T function is executed after the movement is completed.
(b) T function is executed at the same time as when the movement command is issued.

Processing and completion sequences are required for all the T commands.
(Note 1) In some setting and display units, there may be screens that cannot display all eight digits.

(2) L system

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





The 6-digit (or 7-digit) tool No. code data and start signal will be output to the PLC.
All the other details are the same as in M system.

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	O	O	O	O	O	O
L system	O	O	O	O	O	O

Miscellaneous function, or M function, is used to command auxiliary functions for NC, such as rotating the spindle forward/backward or stopping it, as well as turning the cooling oil ON/OFF.
This can be designated with an 8-digit number following address M (0 to 99999999). Up to four sets of M commands can be issued in one block. Note that the number M commands to be issued within the same block is determined by parameter.
BCD output or binary output can be selected by parameter.
Output signal is 8-digit BCD code and start signal, signed 32-bit binary data and start signal, or non-signed 32-bit binary data and start signal.
(Example) G00 Xx1 Mm1 Mm2 Mm3 Mm4 ;

(1) When 5 or more sets are commanded in one block, only the last 4 sets are valid.
M00, M01, M02, M30, M98 and M99 are used only for some specific purposes and cannot be assigned as regular M functions.

(2) Processing and completion sequences are required for all M functions except M98 and M99.
Refer to user PLC specification for the relationship between values and functions.
As for M00, M01, M02 and M30, next block is not read into pre-reading buffer due to ban on pre-reading processing.

(3) Although M00, M01, M02, and M30 output an independent signal for each, the independent output of M00, M01, M02 and M30 will be reset upon pressing a reset key.

(4) M command can be designated with other commands within the same block. In the case where a movement command is in the same block, two different command sequences are available.
Depending on user PLC specifications, either one of the following two will be applied.
(a) M function is executed after the movement is completed.
(b) M function is executed at the same time as when the movement command is issued.
Processing and completion sequences are required for all the M commands except M98 and M99.

(Note 1) In some setting and display units, there may be screens that cannot display all eight digits.

8.3.2 Multiple M Codes in 1 Block

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	O	O	O	O	O	O
L system	O	O	O	O	O	O

Up to four sets of M commands can be issued in a block.
Respective processing and completion sequences are required for all M commands included in a block (except M98 and M99).
(Note 1) This function requires a built-in PLC. In this case, the code data and start 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

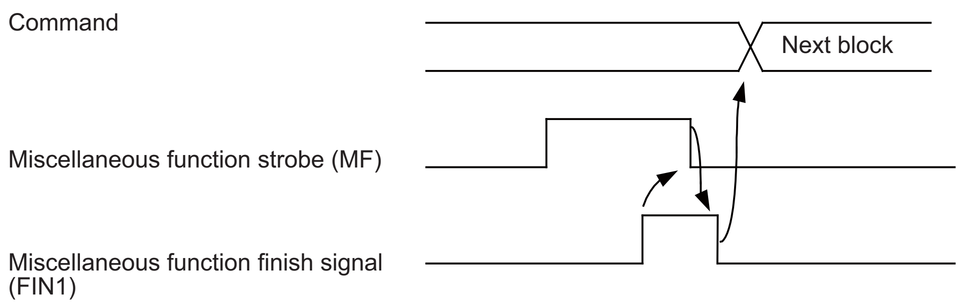
If movement or dwell command exists in the same block as these M commands, this signal is output upon completion of the movement or dwell command.

8.3.4 Miscellaneous Function Finish

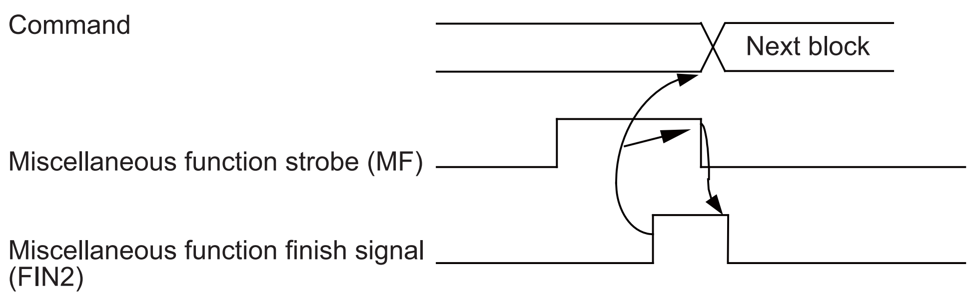
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

Miscellaneous function finish signal 1 (FIN1)
When the controller checks that FIN1 is ON, it sets the function strobes OFF. Furthermore, when the PLC checks that the function strobes are OFF, it sets FIN1 OFF. The controller checks that FIN1 is OFF and advances to the next block.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



Miscellaneous function finish signal 2 (FIN2)
When the controller checks that FIN2 is ON, it sets the function strobes OFF and simultaneously advances to the next block. The PLC checks that the strobe signals are OFF and sets FIN2 OFF.
Below is an example of a time chart applying when a miscellaneous function has been assigned.



8.3.5 M Code Output during Axis Traveling

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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 traveling
Xx1,Zz1,Cc1	: Movement start points
□□□□	: Miscellaneous function

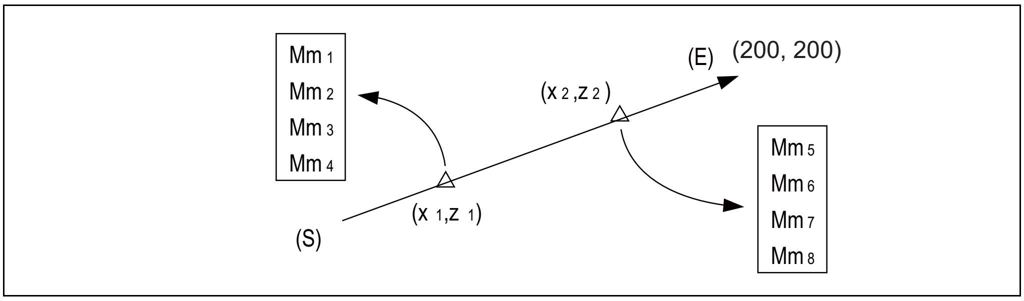
(1) This command is issued independently immediately before the block with the movement command that activates the miscellaneous function.

(2) Single block stop does not apply to this command.

(3) The maximum number of groups to which the miscellaneous functions in the G117 block can be issued is as follows:

M commands	4 sets
S commands	2 sets
T commands	1 set
2nd miscellaneous function	1 set

(4) This command can be issued in up to two consecutive blocks.
When issued in three or more consecutive blocks, the last two blocks will be valid.
(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 ;
:



(E) End point
(S) Start point

8.4 2nd Miscellaneous Functions (B)

8.4.1 2nd Miscellaneous Functions

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The code data and start signals are output when an 8-digit number is assigned following the address code A, B or C - whichever does not duplicate the axis name being used.
Processing and complete sequences must be incorporated on the PLC side for all 2nd miscellaneous commands.
(Note 1) This function requires a built-in PLC.
(Note 2) There are some screens in the setting and display unit that cannot display all eight digits.

9

Tool Compensation

9.1 Tool Length/Tool Position

9.1.1 Tool Length Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(1) M system

G43 Zz1 Hh1 ;
G44 Zz1 Hh1 ;

G43	: Tool length compensation command + direction (z1+h1)
G44	: Tool length compensation command + direction (z1-h1)
Zz1	: Compensation axis. Tool length compensation can be provided not only for the Z axis but for all the other axes (X, Y, etc.) which can be controlled in the system.
Hh1	: Compensation No.

Compensation can be canceled by the following G commands.

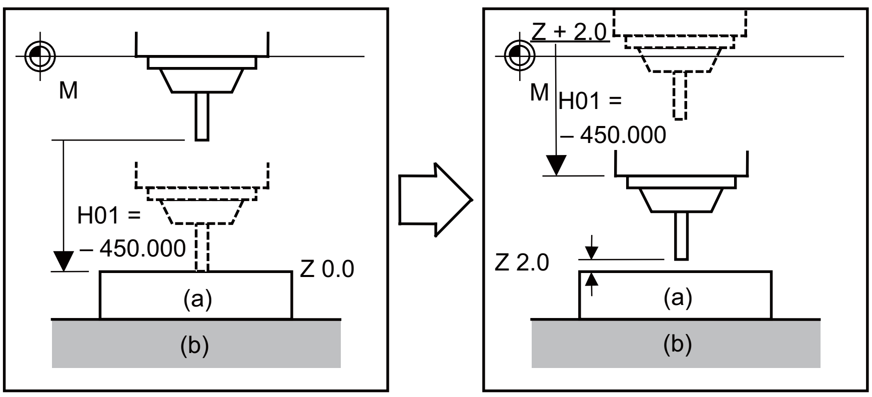
G49 ;
G43 H0 ;
G44 H0 ;

(Note) When the tool length compensation axis is returned to the reference position, the compensation of that axis is canceled.

(Example) Example of tool length compensation 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 compensation amount is set as a negative value such as H01 = -450.000.

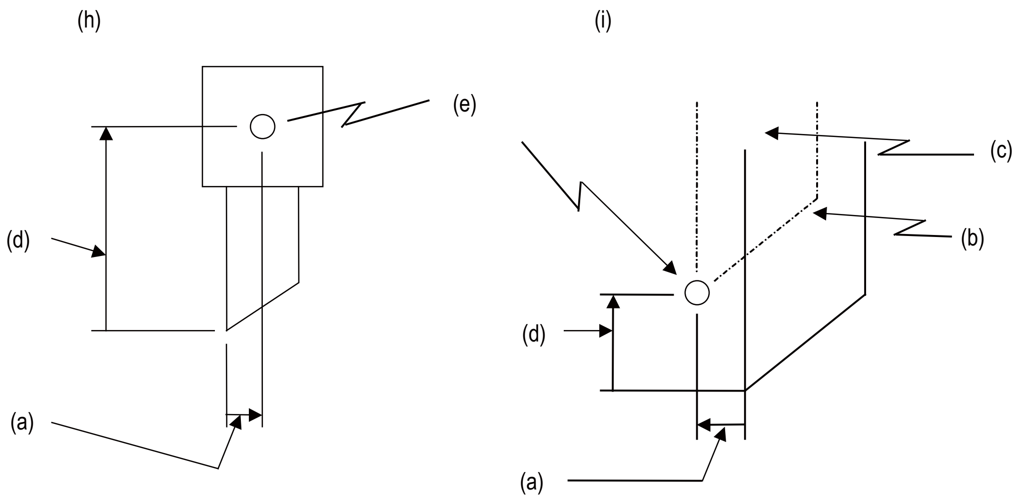


(a) Workpiece
(b) Table

(2) L system

(a) Shape compensation

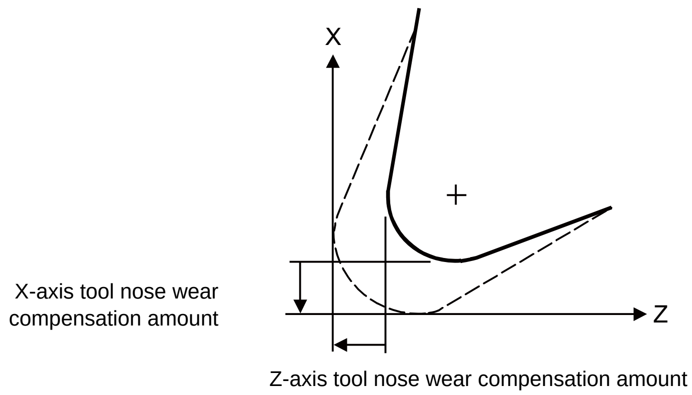
Tool length is compensated in reference to the programmed base position. The programmed base position is usually the center of the tool rest or the nose position of the base tool.



(a) Z-axis tool length compensation
(b) Tool used for machining
(c) Base tool
(d) X-axis tool length compensation
(e) Base position (base point)
(h) The programmed base position is the center of the tool rest:
(i) The programmed base position is the nose of the base tool:

(b) Wear compensation

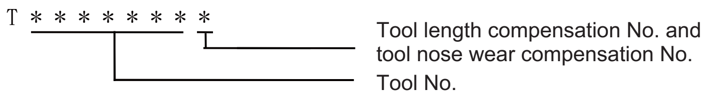
The wear of a tool nose can be compensated.

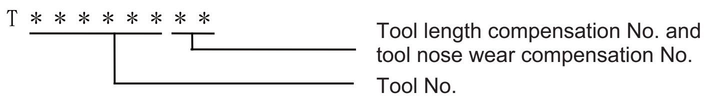


(c) Command format

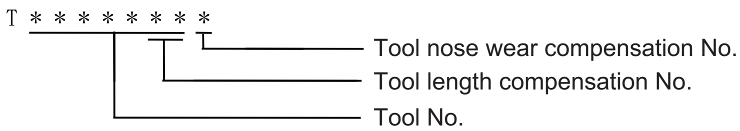
Tool compensation is performed by a T command. It is specified in eight digits following address T. Tool compensation is divided into two types: tool length compensation and tool nose wear compensation. The Nos. of such two types of compensations are specified by a parameter. Also a parameter is used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

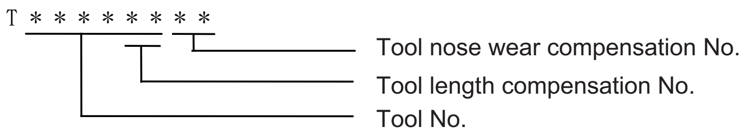
(i) Specifying tool length and wear compensation Nos. together using one or two low-order digits of the T command





(ii) Specifying tool length and wear compensation Nos. separately





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

9.1.2 Tool Position Offset

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

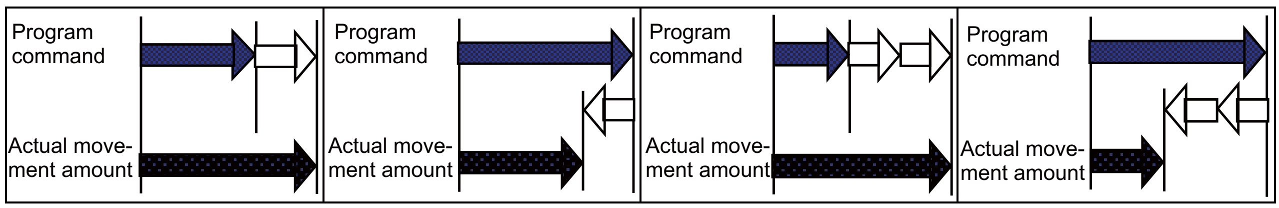
This function uses commands to control the movement by changing the positions of the end points of the movement commands to positions which have been extended or reduced by an amount equivalent to the tool compensation amount.
This function can be used to compensate for the difference in distance between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

G45 G00 Xx1 Yy1 Dd1 ;

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

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

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



(1) If the start and end points are on an axis, the radius can be extended or reduced only for one-quarter, one-half and three-quarter arcs.
(2) In the case of absolute commands, the position is extended or reduced in each axial direction from the end point of the previous block along the line of the movement toward the position commanded in the block containing the G45 (or G46, G47 or G48) command.
(3) In the case of simultaneous n axes command, the same amount of offset is applied to all the axes that have the command within the range of the number of the axes which can be simultaneously controlled. Tool position offset is also valid for additional axes.

9.1.3 Tool Compensation for Additional Axes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

The tool compensation for the lathe is valid for the X and Z axes. If an additional axis (Y axis) is added, the tool compensation will be validated for the additional axis.

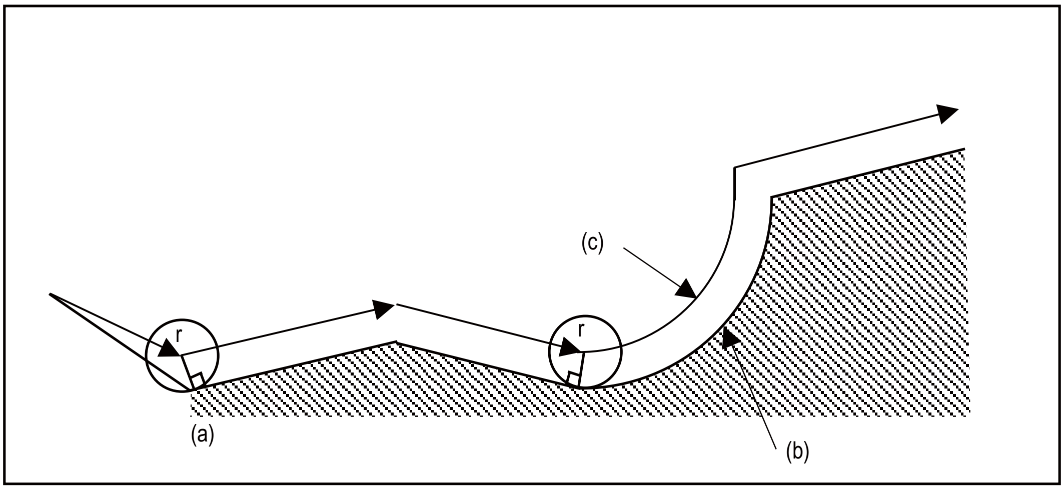
9.2 Tool Radius

9.2.1 Tool Radius Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

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

G code	Function
G38	Vector designation 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



(a) r: Tool radius compensation amount
(b) Programmed path
(c) Tool center path

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	: Tool radius compensation left command
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

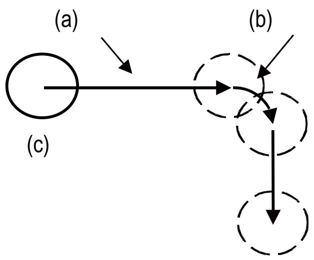
G18: ZX plane, Z, X, K, I

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 ;

G39	: Corner arc during tool radius compensation
Xx1,Yy1	: Movement amount



(a) Tool center path
(b) Arc inserted at corner
(c) Programmed path

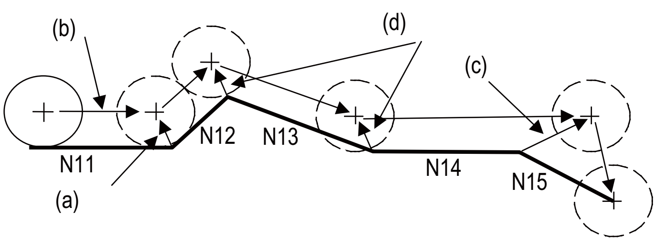
The compensation vector can be changed in following two ways.

G38 Xx1 Yy1 ;	
G38	: Vector designation for tool radius compensation
Xx1,Yy1	: Movement amount

The tool radius compensation vector amount and direction are retained.

G38 Xx1 Yy1 Ii1 Jj1 Dd1 ;	
G38	: Vector designation for tool radius compensation
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.



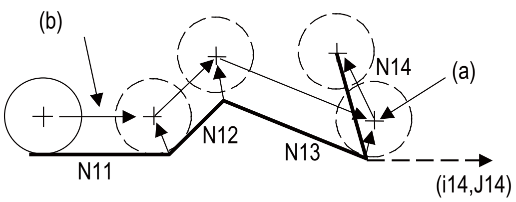
N11 G01 Xx11;
N12 G38 Xx12 Yy12;
N13 G38 Xx13 Yy13;
N14 G38 Xx14 Ii14 Ji14 Dd14;
N15 G40 Xx15 Yy15;

(a) Intersection point vector
(b) Tool center path
(c) Vector with length D (i14, j14)
(d) Holding of previous intersection point vector

The tool radius compensation is canceled by the following command.

G40 Xx1 Yy1 Ii1 Jj1 ;	
G40	: Tool radius compensation cancel
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.



N11 G01 Xx11 ;
N12 Xx12 Yy12 ;
N13 Xx13 Yy13 ;
N14 G40 Xx14 Ii14 Jj14 ;

(a) When i and j commands are assigned to G40
(b) Tool center path

9.2.2 3-dimensional Tool Radius Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This command serves the function of compensating for the spherical radius of ball end mills. It compensates for the actual tool center path to be either more outside or inside the programmed path by an amount equivalent to the tool radius amount in accordance with the 3-dimensional vectors.

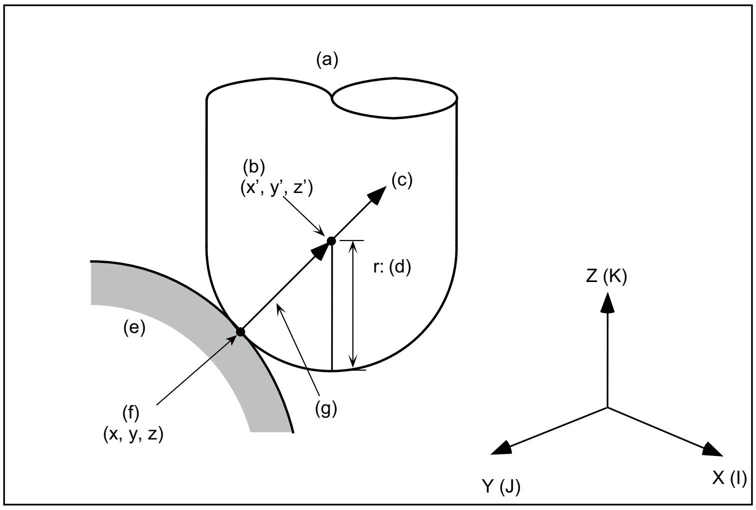
G code	Function
G40 G41 G42	Tool radius compensation cancel Tool radius compensation left command (compensation in the direction toward IJK) Tool radius compensation right command (compensation in the direction away from IJK)

With 3-dimensional tool radius compensation, the compensation is controlled from the block in which the block containing the G41 or G42 command has been designated.

G01 G41 Xx1 Yy1 Zz1 Ii1 Jj1 Kk1 Dd1 ;

G01	: Cutting
G41	: Tool radius compensation left command
Xx1,Yy1,Zz1	: Movement axis
Ii1,Jj1,Kk1	: Tool sphere center vectors
Dd1	: Compensation No.

The radius from the tool spherical center to the end nose serves as the compensation amount that corresponds to the tool compensation No. designated by d1. The compensation Nos. that can be used are limited by the "number of tool compensation sets." (Refer to the section "9.3.1 Number of Tool Compensation Sets".)
If the compensation vectors (tool spherical center vectors) are to be changed, the G41 and I, J and K commands must be assigned.



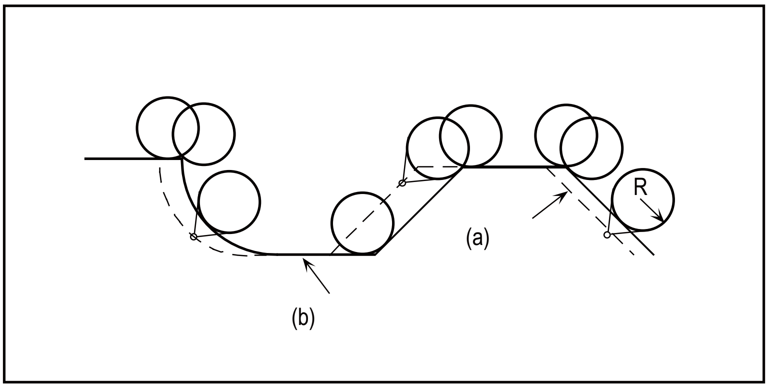
(a) Tool
(b) Tool center coordinates
(c) (I, J, K) Normal line vector
(d) Tool radius
(e) Work
(f) Programmed coordinates position
(g) 3-dimensional compensation vector

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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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

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



(a) Compensated path
(b) Programmed path

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.

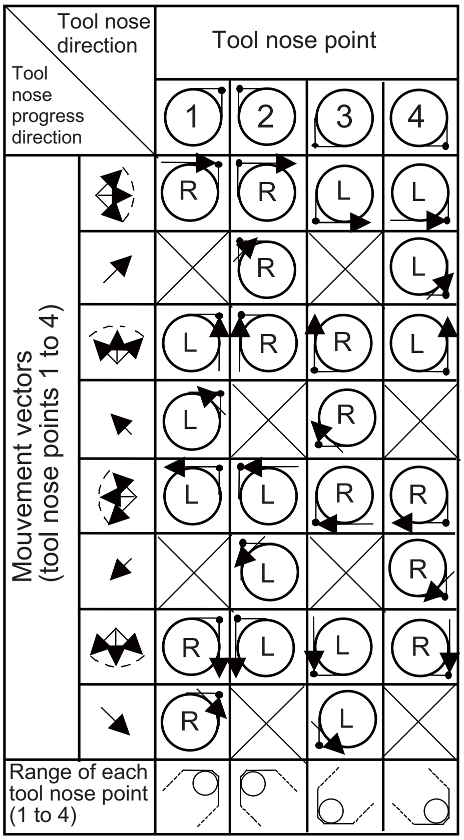
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/40)

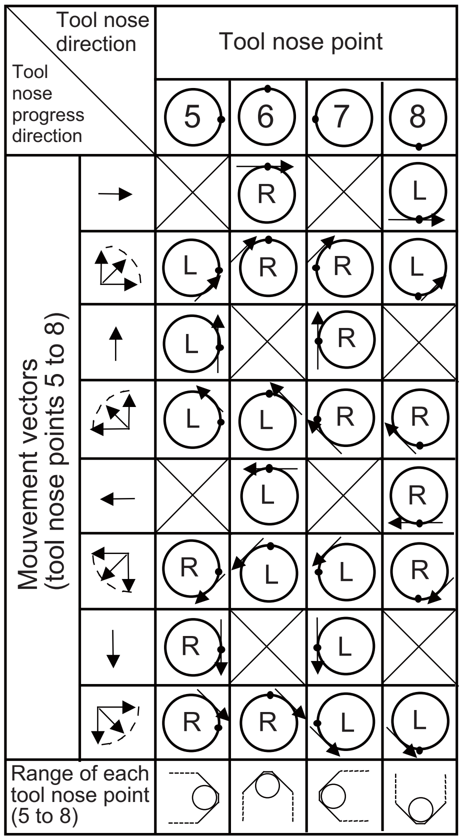
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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

G code	Function
G40 G46	Nose radius compensation cancel 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:





9.2.5 Tool Radius Compensation Diameter Designation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

Tool diameter designation handles the compensation amount as diameter value and compensates the amount set in the tool compensation amount screen when tool radius compensation is commanded. Whether compensation amount is handled in radius value or diameter value is switched by the parameter.

9.3 Tool Compensation Amount

9.3.1 Number of Tool Compensation Sets

The number of tool compensation sets is as follows.

<M system>

Function name	Common for part systems or 1-part system specification	Independent for part systems and multi-part system specification
Number of tool compensation sets (40 sets) (standard)	40 sets	20 sets/part system
Number of tool compensation sets (200 sets)	200 sets	100 sets/part system
Number of tool compensation sets (400 sets)	400 sets	200 sets/part system
Number of tool compensation sets (999 sets)	999 sets	500 sets/part system

<L system>

<table>
<tr><th colspan="2">Function name</th><th>Common for part systems or 1-part system specification</th><th>Independent for part systems and multi-part system specification</th></tr>
<tr><td colspan="2">Number of tool compensation sets (20 sets) (standard)</td><td>20 sets</td><td>10 sets/part system</td></tr>
<tr><td colspan="2">Number of tool compensation sets (80 sets)</td><td>80 sets</td><td>40 sets/part system</td></tr>
<tr><td rowspan="2">Number of tool compensation sets ((99 x number of part systems) sets)</td><td>#1096 T_Ltyp = 1,3</td><td>99 sets</td><td>99 sets/part system</td></tr>
<tr><td>#1096 T_Ltyp = 2</td><td>80 sets</td><td>80 sets/part system</td></tr>
</table>

(Note) Whether the tool compensation memory is provided commonly for the part systems or independently for the part systems depends on the parameter settings.

9.3.1.1 20 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

9.3.1.2 40 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

9.3.1.3 80 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

9.3.1.4 200 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

9.3.1.5 400 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

9.3.1.6 999 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

9.3.1.7 (99× Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

9.3.2 Compensation Memory

9.3.2.1 Tool Shape/Wear Compensation Amount

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function registers the tool shape compensation and wear compensation amounts. Compensation may encompass two or more axes.

(1) Shape compensation amount

The tool length compensation amount, tool radius compensation amount, nose radius compensation amount, nose radius imaginary tool tip point or tool width can be set as the shape compensation amount.

The compensation amount that can be set and used differs depending on whether compensation amount setting type 1, 2 or 3 is used.

(2) Wear compensation amount

When the tip of the tool used has become worn, the wear compensation amount is used to compensate this wear. Types of wear compensation amounts include the tool length wear compensation amount, tool radius wear compensation amount, and nose radius wear compensation amount.

The wear compensation amount can be used with compensation amount setting types 2 and 3, and it is added to the shape compensation amount for compensation.

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

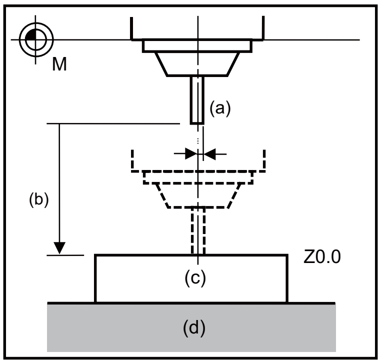
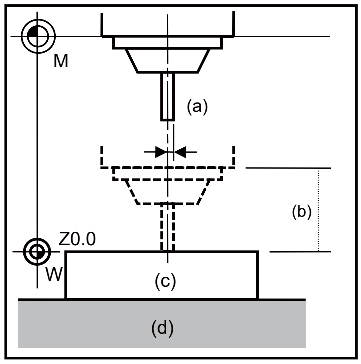
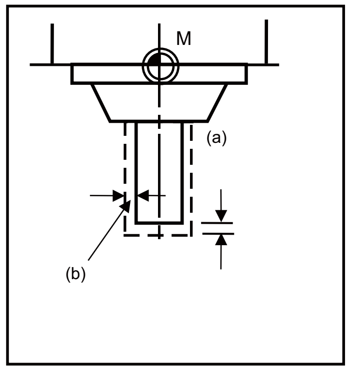
This is the value that is used by rotary tools.

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

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

One compensation amount data is registered in one compensation No., and the compensation Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, compensation 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 compensation amounts/with wear compensation [M system]
As with type 1, type 2 is for the compensation amounts used by rotary tools.
With type 2, four kinds of compensation amount data are registered in one compensation No.: the tool length compensation amount, tool length wear compensation amount, tool radius compensation amount, and tool radius wear compensation amount.
When a compensation No. is assigned by address D as the compensation 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 compensation using the amount obtained by adding the tool length compensation amount and tool length wear compensation amount.

<table>
<tr><td colspan="2">Figure: Example of how the compensation amount is handled when using the type 1 tool length compensation amount (Compensation types I and II are available for handling compensation amounts.)</td><td rowspan="2">Wear compensation amount when using type 2</td></tr>
<tr><td>Compensation types I</td><td>Compensation types I and II</td></tr>
<tr><td>
</td><td>
</td><td>
</td></tr>
</table>

(a) Tool radius compensation amount
(b) Tool length compensation amount
(c) Workpiece
(d) Table

(a) Tool radius compensation amount
(b) Tool length compensation amount
(c) Workpiece
(d) Table

(a) Tool length wear compensation amount
(b) Tool radius wear compensation amount

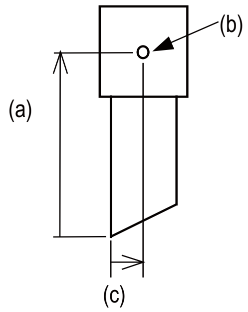
(c) Type 3: 2-axis compensation amounts [L system]

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

As the compensation 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.

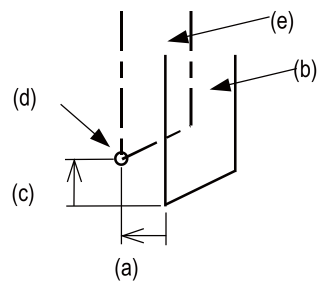
Compensation is carried out in the directions of the X, Y and Z axes from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

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



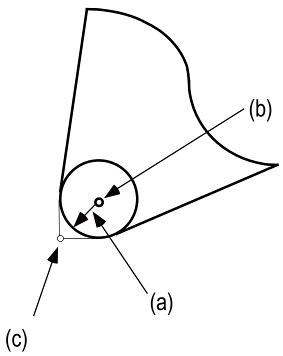
(a) X-axis tool length compensation amount
(b) Base position (base point)
(c) Z-axis tool length compensation amount

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

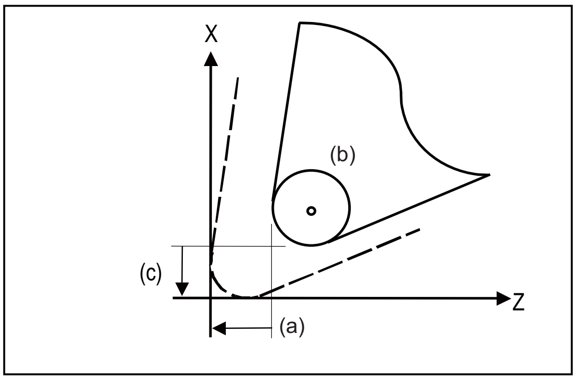


(a) Z-axis tool length compensation amount
(b)Tool used for machining
(c)X-axis tool length compensation amount
(d) Base position (base point)
(e) Base tool

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 compensation amount.



(a) Nose radius compensation amount
(b) Tool nose center
(c) Imaginary tool nose point



(a) Z-axis tool length wear compensation amount
(b) Tool nose
(c) X-axis tool length wear compensation amount

The X-axis tool length compensation amount, Z-axis tool length compensation amount and nose radius compensation amount are set as plus amounts.

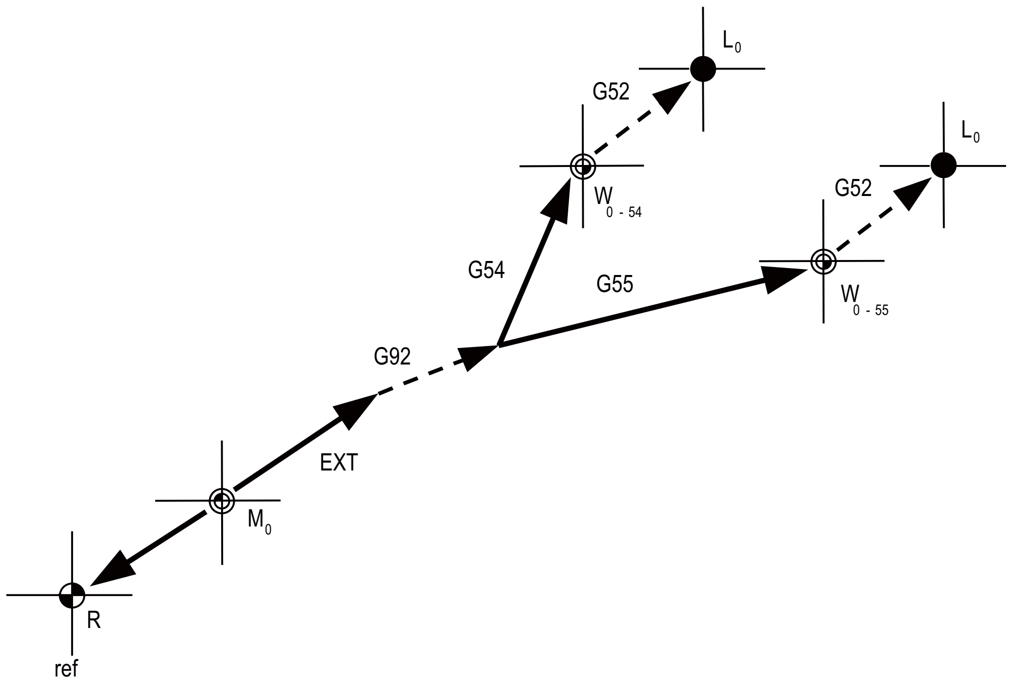
The compensation type (1, 2 or 3) is set using a parameter.

10

Coordinate System

10.1 Coordinate System Type and Setting

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



L0	**Local coordinate system zero point**	⟶	**Offset set with parameters**
G52	**Local coordinate system offset (*1)**	- - -⟶	**Offset set with program**
W0-54	**Workpiece coordinate system zero point (G54)**		**(0 when power is turned ON)**
W0-55	**Workpiece coordinate system zero point (G55)**		
G54	**Workpiece coordinate system (G54) offset (*1)**		**(*1) The G52 offset is available independently for G54 to G59.**
G55	**Workpiece coordinate system (G55) offset**		
G92	**G92 coordinate system shift**		
EXT	**External workpiece coordinate offset**		
M0	**Machine coordinate system zero point**		
ref	**Reference position**		

10.1.1 Machine Coordinate System

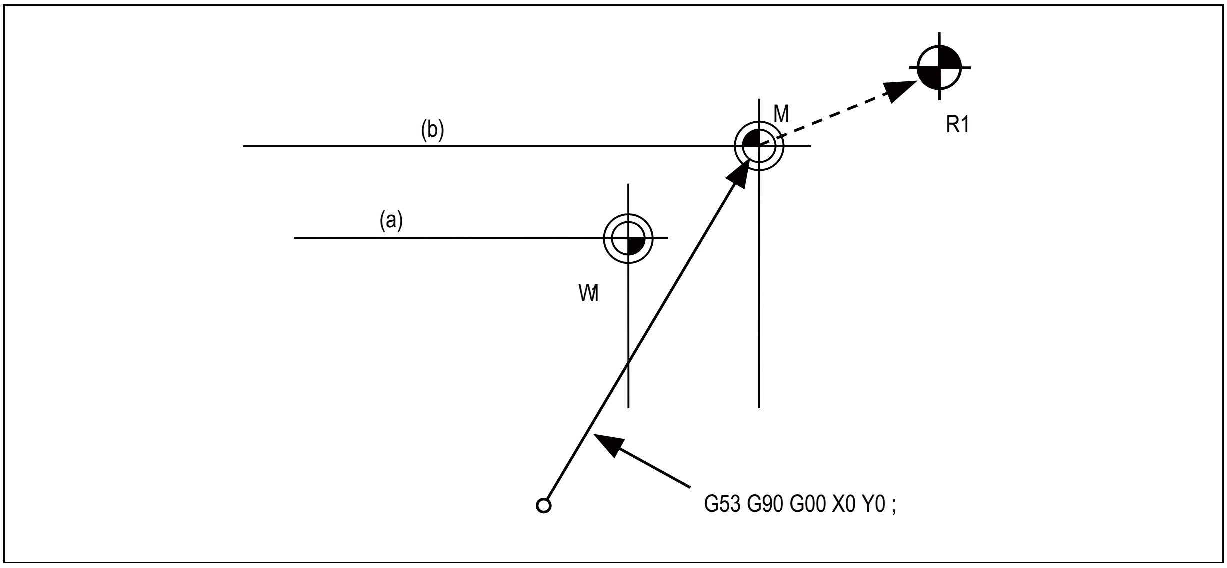
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

G53 (G90) (G00) Xx1 Yy1 Zz1 ;

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

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



(a) Workpiece coordinate system 1(G54)
(b) Machine coordinate system(G53)
R1: 1st reference position

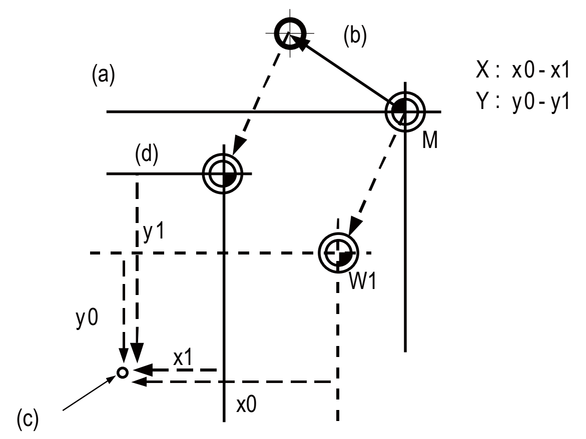
10.1.2 Coordinate System Setting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Among the workpiece coordinate systems with the G92, the program coordinate system (the programmed zero point) can be changed.
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 position 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

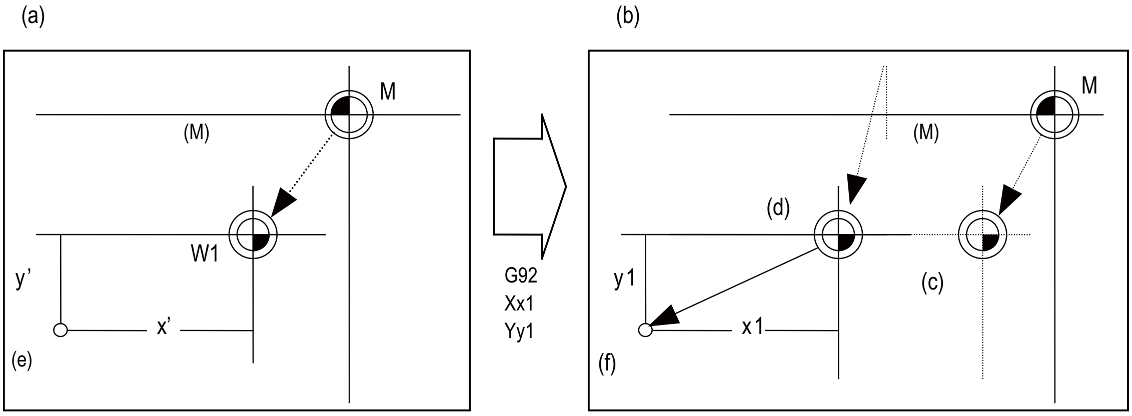
Example where W1 is shifted to new W1 when the machine was at the position (x0, y0) above W1 and the G92 Xx1 Yy1; command was assigned when the workpiece coordinate system W1 is modal
(external workpiece coordinate system offset = 0; interrupt amount offset = 0)



(a) Machine coordinate system
(b) G92 offset amount
(c) Machine position
(d) New W1

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

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



(a) Coordinate system created by automatic coordinate system setting
(b) Coordinate system after coordinate system setting by G92
(c) Old W1
(d) New W1
(e) Tool position
(f) G92 command position
(M) Machine coordinate system

(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 position return
  (b) Move to machine coordinate system zero point and assign G92 and G53 commands in same block to set the machine coordinate system.

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

10.1.3 Automatic Coordinate System Setting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

After the power is turned ON, the basic machine coordinate system and the workpiece coordinate system are automatically set without executing the zero point return.
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.

10.1.4 Workpiece Coordinate System Selection

10.1.4.1 Workpiece Coordinate System Selection (6 Sets)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When a multiple number of workpieces with the same shape are to be machined, these commands enable the same shape to be machined by executing a single machining program in the coordinate system of each workpiece.
Up to 6 workpiece coordinate systems can be selected.
The G54 workpiece coordinate system is 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 formats 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 command)
G54	: Coordinate system selection
G00	: Movement mode
Xx1,Yy1,Zz1	: Coordinate position of end point

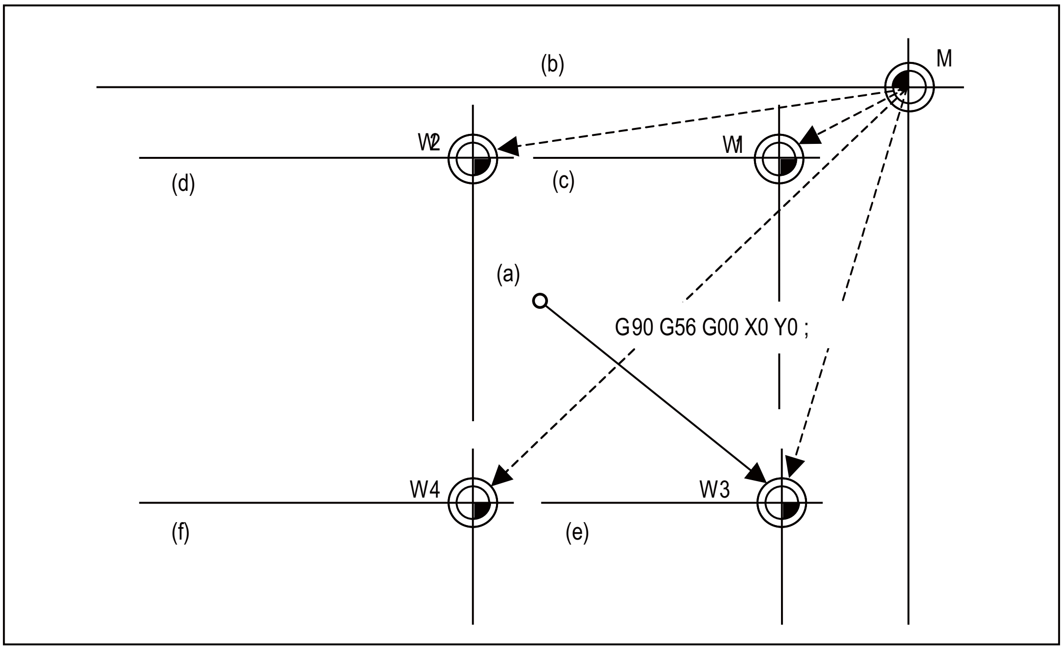
The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.

Settings can be performed in one of the following three ways:

(a) Setting using the setting and display unit

(b) Setting using commands assigned from the machining program

(c) Setting from the user PLC



(a) Start
(b) Machine coordinate system (G53)
(c) Workpiece coordinate system 1 (G54)
(d) Workpiece coordinate system 2 (G55)
(e) Workpiece coordinate system 3 (G56)
(f) Workpiece coordinate system 4 (G57)

10.1.4.2 Extended Workpiece Coordinate System Selection (48 Sets) G54.1P1 to P48

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

When a multiple number of workpieces with the same shape are to be machined, these commands enable the same shape to be machined by executing a single machining program in the coordinate system of each workpiece.
In addition to the six workpiece coordinate systems G54 to G59, 48 workpiece coordinate systems can be used by assigning G54.1Pn command.
The command format to select the workpiece coordinate system using the G54.1Pn command and to move on the workpiece coordinate system are given below.

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

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

The numerical value n of P following G54.1 indicates each workpiece coordinate system. Specify a value between 1 and 48.
The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following three ways:

(a) Setting using the setting and display unit
(b) Setting using commands assigned from the machining program
(c) 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.

10.1.4.3 Extended Workpiece Coordinate System Selection (96 Sets) G54.1P1 to P96

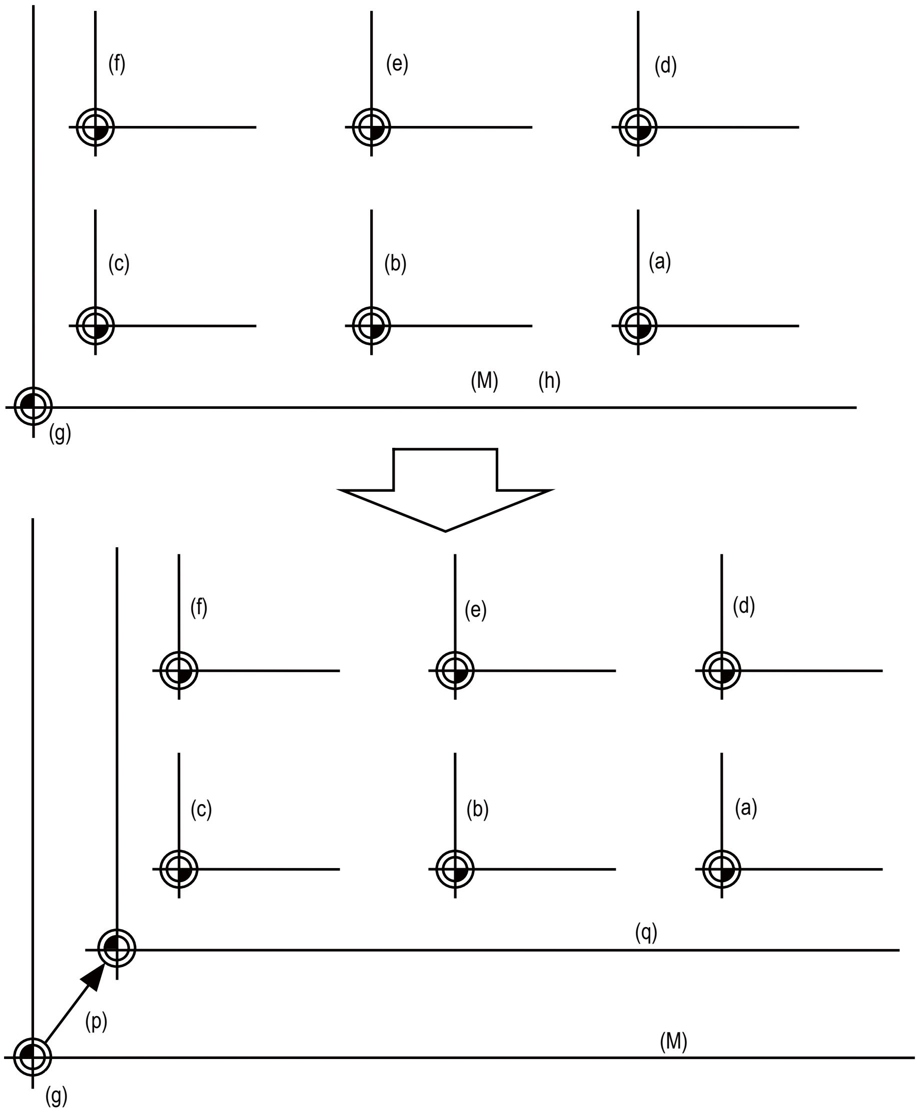
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

In addition to the six workpiece coordinate systems (G54 to G59), 96 workpiece coordinate systems can be used by assigning G54.1Pn command. Refer to "Extended workpiece coordinate system selection (48 sets) G54.1P1 to P48" for details.

10.1.5 External Workpiece Coordinate Offset

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

External workpiece coordinate offset that serves as the reference for all the workpiece coordinate systems is available outside the workpiece coordinates.
By setting the external workpiece coordinate offset, the external workpiece coordinate system can be shifted from the machine coordinate system, and all the workpiece coordinate systems can be simultaneously shifted by an amount equivalent to the offset.
When the external workpiece coordinate offset is zero, the external workpiece coordinate systems coincide with the machine coordinate system.
It is not possible to assign movement commands with the external workpiece coordinate selected.



(a) Workpiece coordinate 3 (G56)
(b) Workpiece coordinate 2 (G55)
(c) Workpiece coordinate 1 (G54)
(d) Workpiece coordinate 6 (G59)
(e) Workpiece coordinate 5 (G58)
(f) Workpiece coordinate 4 (G57)
(g) Machine coordinate zero point
(h) (= External workpiece coordinate system)
(q) External workpiece coordinate system
(M) Machine coordinate system

10.1.6 Workpiece Coordinate System Preset (G92.1)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

The workpiece coordinate system is shifted from the machine coordinate system when the such operations or the programmed commands as below have been performed.
- When manual intervention has occurred in the manual absolute OFF status
- When a movement command was performed in the machine lock status
- When movement was initiated by handle interrupt
- When a movement command was performed in the mirror image mode
- When a local coordinate system was set using the G52 command
- When a workpiece coordinate system was shifted using the G92 command

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

Command format

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

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

10.1.7 Local Coordinate System

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

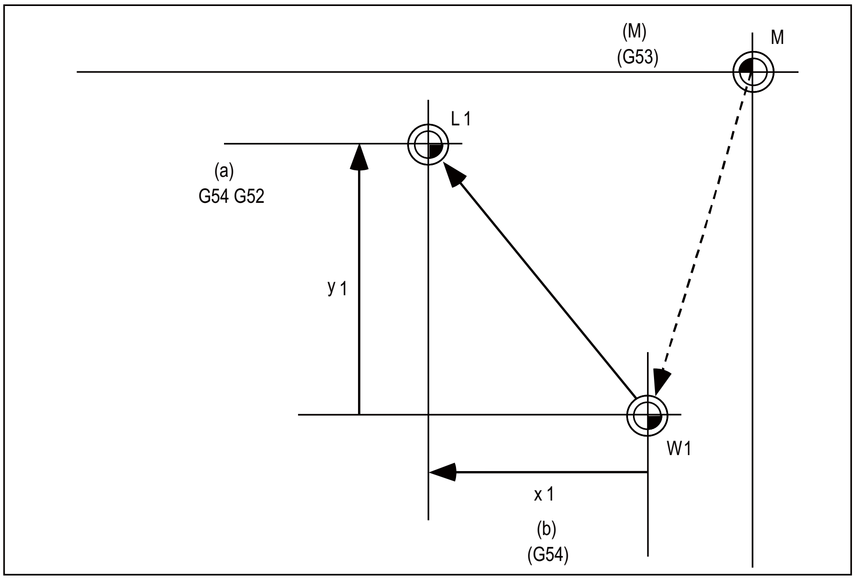
This function is for assigning a coordinate system on the workpiece coordinate system now being selected. This enables the workpiece coordinate system to be changed temporarily.
The local coordinate system can be selected independently on each workpiece coordinate system G54 to G59.

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

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

(G54) G52 Xx1 Yy1 Zz1 ;	
(G54)	: Workpiece coordinate 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 position setting mode, the position obtained by adding the local coordinate offset amount to the previously specified offset amount serves as the new local coordinate zero point.
If no workpiece coordinates are designated, the local coordinates will be created on the currently selected workpiece coordinates.
This command is unmodal but the local coordinate system created by G52 is valid until the next G52 command is issued.
The local coordinate system is canceled by the input of the reset signal or by manual or automatic dog-type reference position return.



(a) Local coordinate
(M) Machine coordinate system
(b) Workpiece coordinate 1

10.1.8 Coordinate System for Rotary Axis

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

<table>
<tr><th rowspan="3"></th><th colspan="4">Rotary axis</th><th rowspan="3">Linear axis</th></tr>
<tr><th colspan="2">Rotating type rotary axis</th><th colspan="2">Linear type rotary axis</th></tr>
<tr><th>Short-cut invalid</th><th>Short-cut valid</th><th>Workpiece coordinate position linear type</th><th>All coordinate position linear type</th></tr>
<tr><td>Workpiece coordinate position</td><td colspan="2">Displayed in the range of 0° to 359.999° .</td><td colspan="3">Displayed in the range of 0° to 99999.999° .</td></tr>
<tr><td>Machine position/ relative position</td><td colspan="3">Displayed in the range of 0° to 359.999° .</td><td colspan="2">Displayed in the range of 0° to 99999.999° .</td></tr>
<tr><td>ABS command</td><td>The incremental amount from the end point to the current position is divided by 360, and the axis moves by the remainder amount according to the sign.</td><td>Moves with a short-cut to the end point.</td><td colspan="3">In the same manner as the normal linear axis, moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees.).</td></tr>
<tr><td>INC command</td><td colspan="5">Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.</td></tr>
<tr><td rowspan="2">Reference position return</td><td colspan="5">Follows the absolute/relative command for a movement to the interim position.</td></tr>
<tr><td colspan="3">Returns to the reference position from the interim position within a 360 degree movement.</td><td colspan="2">Moves and returns in the reference position direction for the difference from the current position to the reference position.</td></tr>
</table>

10.1.9 Plane Selection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

G17, G18, and G19 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)

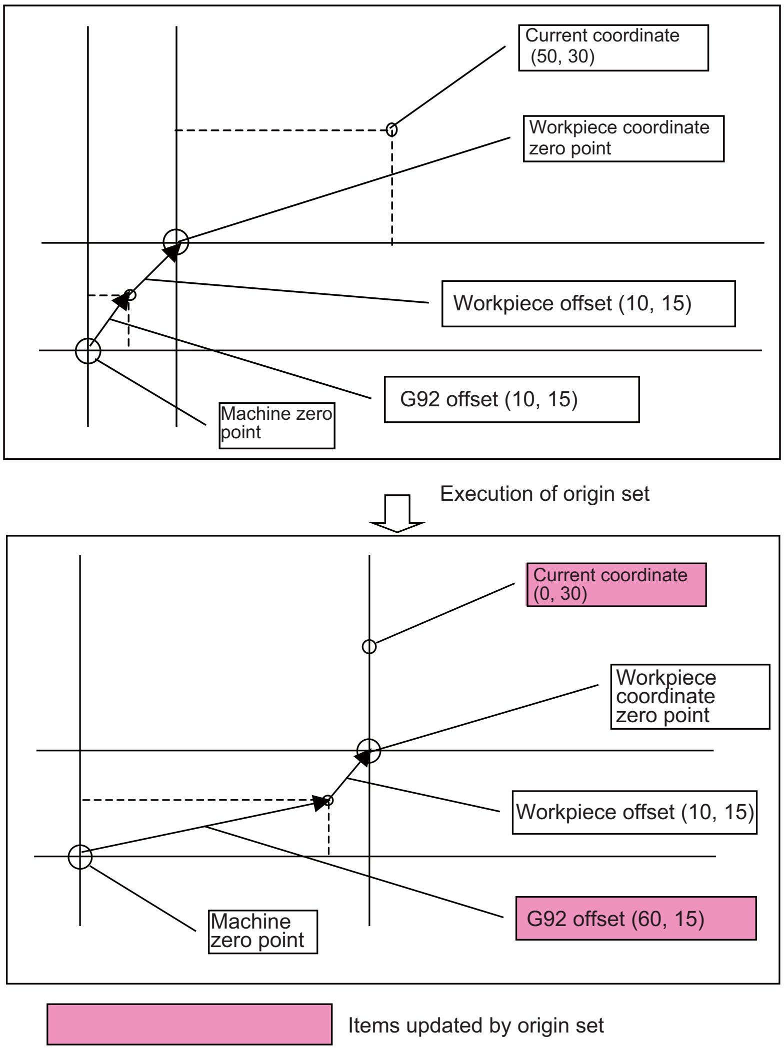
G19 X100. ;	With these program commands, X100. is the axis which does not exist on the G19 (Yp, Zp) plane, Yp-Zp plane is selected by G19 and the X axis moves by 100. mm separately from the plane selection.
G17 X100 . R50. ;	With these program commands, the Xp-Yp plane is selected by G17 and the arc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

10.1.10 Origin Set/Origin Cancel

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

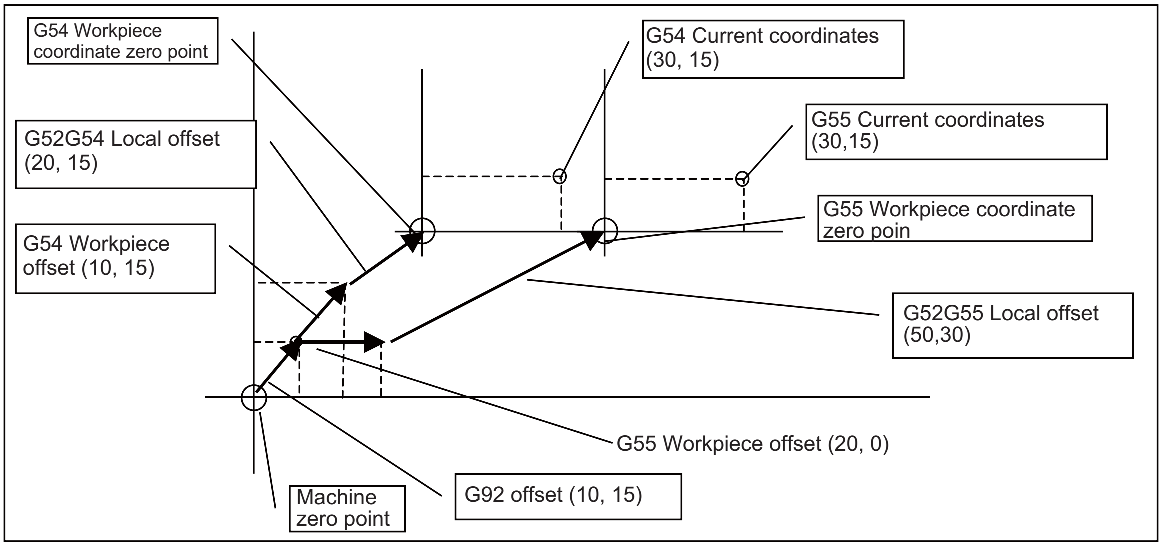
(1) Origin set

Origin set is a function that shifts the coordinate system so that the current position is the zero point on the workpiece coordinate system containing the workpiece coordinate system's offset value. The relative position counter and workpiece coordinate counter are set to "0" with this operation. In other words, this is the same as the coordinate system setting command "G92 X0;". (For target axis: X)

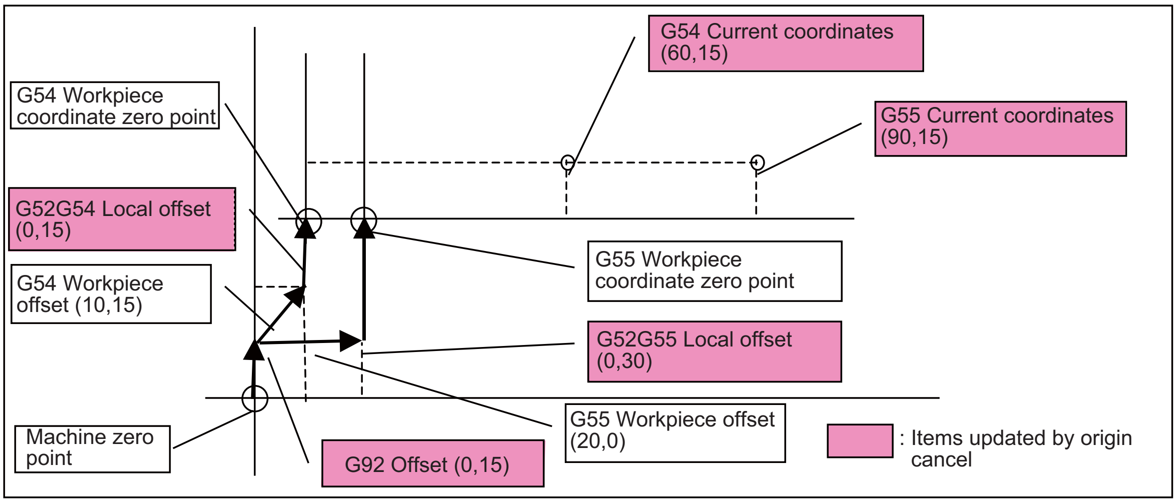


(2) Origin cancel

Origin cancel is a function that manually cancels all deviated amounts, and shifts to the designated zero point with the workpiece offset.
The relative position counter and machine position counter are set to "0" with this operation. In other words, this is the same "G92 G53 X0 ;". (For target axis: X)



Execution of origin cancel



10.1.11 Counter Set

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The relative position counter can be set to an arbitrary value by operating the setting and display unit screens.

Select the axis and then input a value. Finally, press the INPUT key.

Only the [Relative Position] display field will change to the set value. The other coordinate positions displayed will not change.
Up to 9 digits can be input in the integer section. The number of digits after the decimal point depends on the parameter setting.

10.2 Return

10.2.1 Manual Reference Position Return

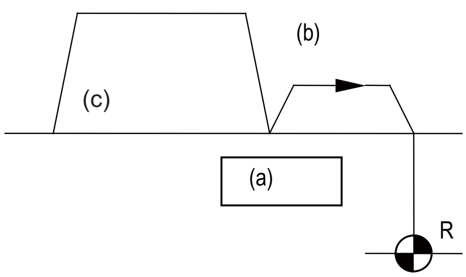
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(1) Return pattern to reference position

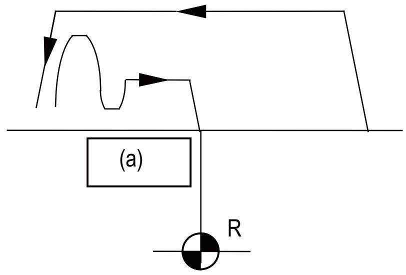
(a) Dog type

When starting in same direction as final advance direction



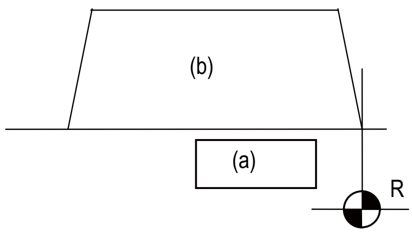
(a) Dog
(b) Creep speed
(c) Reference position return speed

When starting in opposite direction as final advance direction



(a) Dog

(b) High-speed type



(a) Dog
(b) Rapid traverse rate

(2) Differences according to detection method

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

10.2.2 Automatic 1st Reference Position Return

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

The G28 programming format is given below.

G28 Xx1 Yy1 Zz1 ;

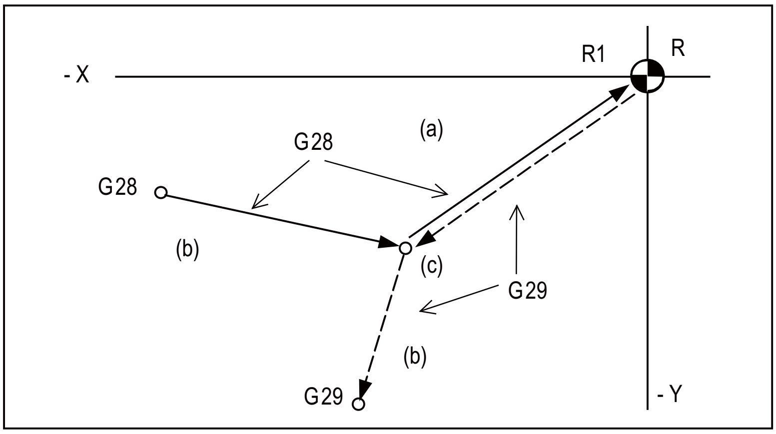
G28	: Return command
Xx1,Yy1,Zz1	: Return control axes (coordinate of interim point)

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

The G29 programming format is given below.

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

The tool is first moved by rapid traverse for each axis 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.



(a) Non - interpolation movement
(b) Interpolation or non - interpolation can be selected
(c) Interim point
R1 1st reference position

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

(Note 1) The automatic 1st reference position return pattern is the same as for manual reference position return.
(Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
(Note 3) If, at the time of the first reference position return, the tool radius compensation or nose radius compensation has not been canceled, it will be temporarily canceled during the movement to the interim point. The compensation is restored at the next movement after the return.
(Note 4) If, at the time of the reference position return, the tool length compensation has not been canceled, it will be canceled and the compensation amount also cleared upon completion of reference position return. The tool length compensation can also be canceled temporarily using a parameter. In this case, however, the tool compensation is restored by the next movement command.
(Note 5) Interpolation or non-interpolation can be selected using a parameter for the movement up to the G28 interim point or for the movement from the G29 interim point to the command point. Non-interpolation applies for movement from the G28 interim point to the reference position and movement up to the G29 interim point.
(Note 6) The machine will not stop at the interim point even when a single block is selected.

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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

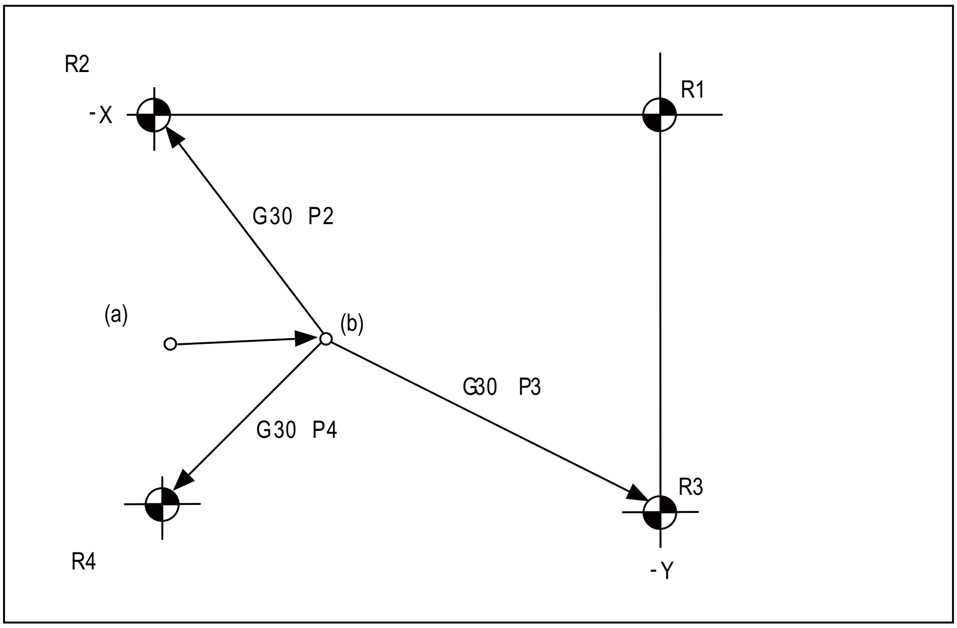
As with automatic 1st reference position return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference positions) characteristic to the machine. The 2nd, 3rd and 4th reference positions can be set by parameters.

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

The G30 programming format is given below.

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

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



(a) Start point
(b) Interim point
R1: 1st reference position
R2: 2nd reference position
R3: 3rd reference position
R4: 4th reference position

(Note 1) The second reference position return is performed if the P address is omitted.
(Note 2) The number of axes for which reference position return can be performed simultaneously depends on the number of simultaneously controlled axes.
(Note 3) If, at the time of the reference position return, the tool radius compensation or nose R compensation has not been canceled, it will be temporarily canceled during the movement up to the interim point. The compensation is restored at the next movement command after the return.
(Note 4) If, at the time of the reference position return, the tool length compensation has not been canceled, it will be canceled and the compensation amount also cleared upon completion of reference position return. The tool length compensation can also be canceled temporarily using a parameter. In this case, however, the tool compensation is restored by the next movement command.
(Note 5) Whether interpolation or non-interpolation is to apply to the movement up to the interim point can be selected using a parameter. Non-interpolation applies for movement from the interim point to each of the reference positions.
(Note 6) The machine will not stop at the interim point even when a single block is selected.

10.2.4 Reference Position Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

Command format

G27 Xx1 Yy1 Zz1 Pp1 ;	
G27	: Check command
Xx1,Yy1,Zz1	: Return control axes
Pp1	: Check No.
	P1:1st reference position check P2: 2 P3: 3 P4: 4

The tool is first positioned by rapid traverse to the assigned position and then, if this is the reference position, the reference position arrival signal is output.
When the address P is omitted, the first reference position verification will be applied.

(Note 1) The number of axes for which reference position check can be performed simultaneously depends on the number of simultaneously controlled axes.
(Note 2) An alarm results unless the tool is positioned at the reference position upon completion of the command.
(Note 3) Whether interpolation or non-interpolation is to apply to the movement can be selected using a parameter.

10.2.5 Absolute Position Detection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

<table>
<tr><th colspan="2">Method</th><th>Details</th><th>Establishment of zero point</th><th>Adjustment of zero point position</th></tr>
<tr><td colspan="2">Dog-type</td><td>Same method as incremental detection dog-type</td><td>Zero point is established with dog-type reference position return completion.</td><td>The value is set in the parameter of zero point shift amount.</td></tr>
<tr><td rowspan="2">Dog-less type</td><td>Marked point method</td><td>The zero point position is set from the screen.</td><td>The zero point is established by input from the zero point initialization screen.</td><td>The value equivalent to the shift amount is set in the zero point initialization screen.</td></tr>
<tr><td>Machine stopper method</td><td>The zero point is established by pressing the machine against a set point on the machine.</td><td>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.</td><td>The value equivalent to the shift amount is set in the zero point initialization screen.</td></tr>
</table>

(Note) This function is valid for the NC axis and the PLC axis. This function cannot be used for the spindle and the auxiliary axis.

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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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
[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 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

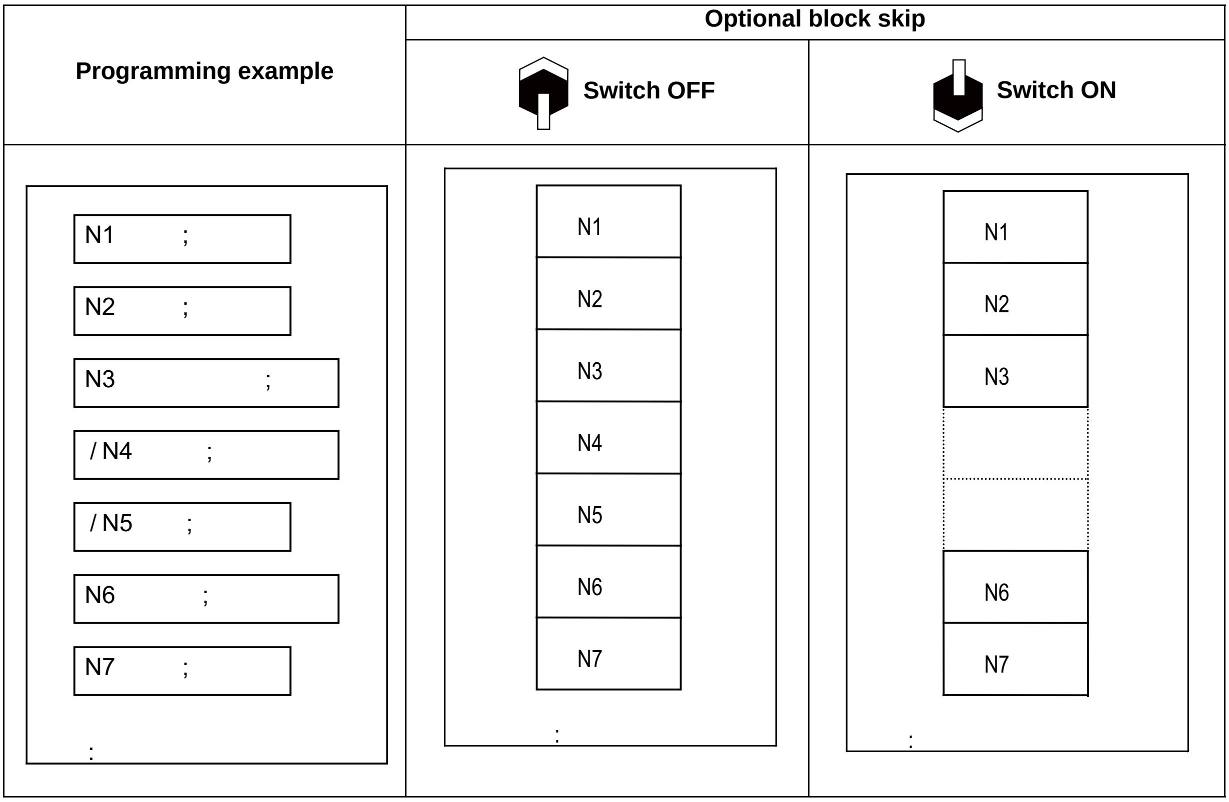
Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



11.1.2 Optional Block Skip Addition

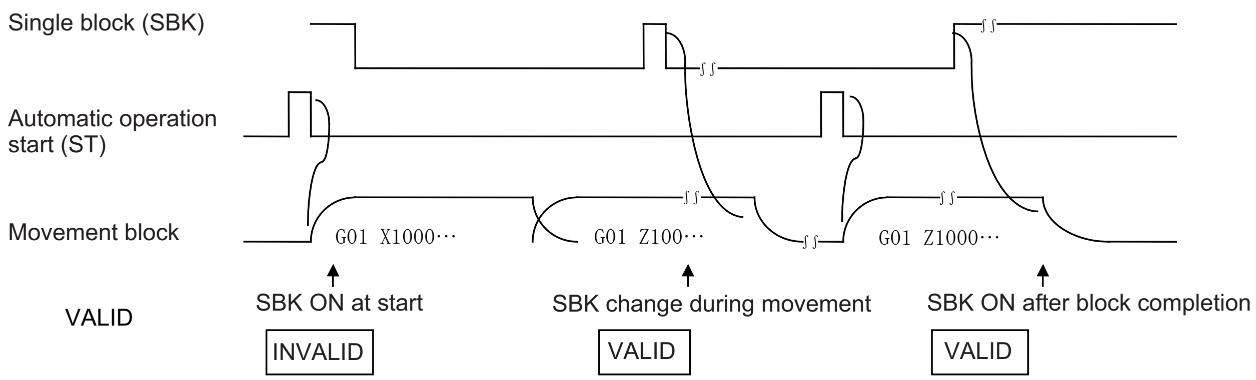
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

When "/n (n:1 to 9)" (slant code) is programmed at the head of a block, and the optional block skip n input signal from the external source is turned ON for automatic operation, the block with the "/n" code is skipped.
If the optional block skip n signal is turned OFF, the block with the "/n" code will be executed without being skipped.

11.1.3 Single Block

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



As with the multi-part system specification, the following function can be selected instead.

Multi-part system single block (L system)
This function is for executing single block operation while maintaining the synchronization between the part systems when two or more part systems are operated. When one part system has been stopped by single block stop, the other part systems are stopped by feed hold.

11.2 Program Test

11.2.1 Dry Run

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

11.2.2 Machine Lock

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When the machine lock input signal is set to ON, the NC operations can be executed without assigning commands to the NC axes.

The feed rate in the machine lock status is the command speed.
The M, S, T and B commands are executed as usual and operation is completed by returning the FIN signal.

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

11.2.3 Miscellaneous Function Lock

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

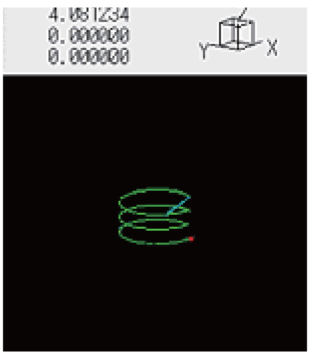
11.2.4 Graphic Check

11.2.4.1 Graphic Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function traces the machining program movement path without executing automatic operation, and draws results of the NC internal operation. The machining program can be checked while drawing the figure accurately at a high speed.
The drawing's viewpoint can be moved, enlarged and reduced while drawing. The three axes displayed are set with the parameters.

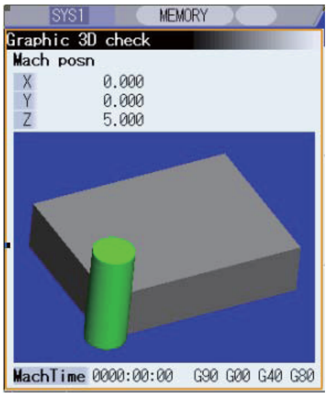
(Note 1) This function is required when performing program check for 2D and 3D.
(Note 2) Automatic operation is disabled during graphic check when using display unit FCU7-DA2xx, FUC7-DU1xx.



11.2.4.2 3D Solid Program Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function draws the workpiece shape and tool movement in the cutting process as a solid image without executing automatic operation. The machining program can be checked while drawing the figure accurately at a high speed.
The workpiece shape's viewpoint can be moved, enlarged and reduced while drawing. The three axes displayed are set with the parameters.
The workpiece shape and tool shape used for this function are set on this screen.
(Note 1) The option function of graphic check (2D) is also required to use this function.
(Note 2) In the 3D check of M700VS, the tool shape of the tap is not reflected.



11.2.4.3 Graphic Check Rotary Axis Drawing

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

When the rotary axis (C axis) is designated with the parameter in program check (2D), the path is drawn depending on the rotary axis. The rotary axis can be set for each part system.

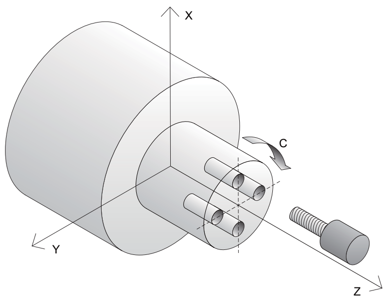
Refer to "11.2.4.1 Graphic Check" for details on the graphic check.

(Note 1) The option function of graphic check is also required to use this function.

(Note 2) When the control axis address other than C axis is designated with the parameter, the path corresponding to the rotary axis is not drawn.

(Note 3) The workpiece coordinate system must be set so that the center of the rotary axis becomes the coordinate system zero point.

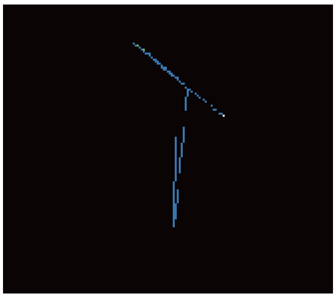
Image of actual tool and workpiece



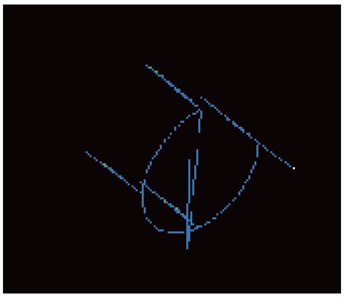
Example of machining program

```
G83Z-20.R-5.Q5000F300 ;
G0C90. ;
G83Z-20.R-5.Q5000F300 ;
G0C180. ;
G83Z-20.R-5.Q5000F300 ;
G0C270. ;
G83Z-20.R-5.Q5000F300 ;
G80 ;
```

Drawing image when the rotary axis drawing is invalid



Drawing image when the rotary axis drawing is valid



11.2.5 Graphic Trace

11.2.5.1 Graphic Trace

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function draws the machine tool's machine position. The movement path following the actual automatic operation or manual operation, or the tool tip movement path can be drawn, and the machine operation during machining can be monitored.
For the display mode, 1-plane, 2-plane and 3-dimensional display are provided. In the 3-dimensional display mode, cubic shapes can be rotated and tracing of the figure seen from the desired direction can be assigned.

Options related to 5-axis control (tool tip center control, tool length compensation along the tool axis, tool handle feed & interruption) are required for the trace of tool path.

11.2.5.2 Graphic Trace Rotary Axis Drawing

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

When the rotary axis (C axis) is designated with the parameter in program trace, the path is drawn depending on the rotary axis. The rotary axis can be set for each part system.

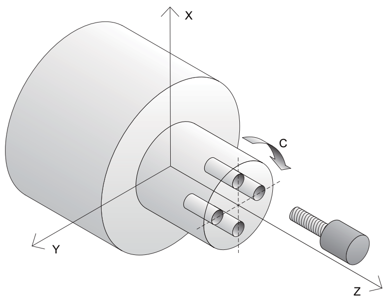
Refer to "11.2.5.1 Graphic Trace" for details on the graphic trace.

(Note 1) The option function of graphic trace is also required to use this function.

(Note 2) When the control axis address other than C axis is designated with the parameter, the path corresponding to the rotary axis is not drawn.

(Note 3) The workpiece coordinate system must be set so that the center of the rotary axis becomes the coordinate system zero point.

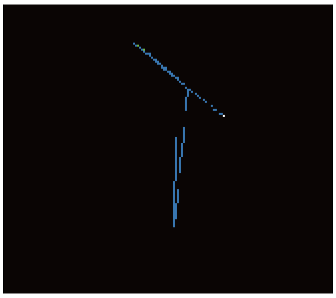
Image of actual tool and workpiece



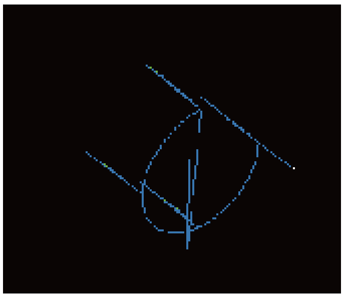
Example of machining program

G83Z-20.R-5.Q5000F300 ;
G0C90. ;
G83Z-20.R-5.Q5000F300 ;
G0C180. ;
G83Z-20.R-5.Q5000F300 ;
G0C270. ;
G83Z-20.R-5.Q5000F300 ;
G80 ;

Drawing image when the rotary axis drawing is invalid



Drawing image when the rotary axis drawing is valid



11.2.6 Machining Time Computation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function analyzes the machining program without moving the axis and calculates the approximate time required for machining.

The graphic check option is required to calculate the machining time.

The machining time is displayed as "hour: minute: second".

The miscellaneous function (M), spindle function (S), tool function (T) and 2nd miscellaneous function execution times differ according to the sequence and each function, and are not included in the calculated machining time.

11.3 Program Search/Start/Stop

11.3.1 Program Search

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

11.3.2 Sequence Number Search

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

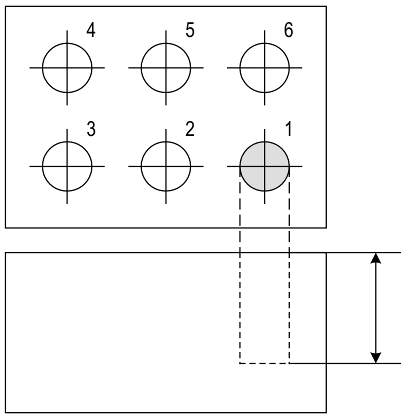
Blocks can be indexed by setting the program No., sequence No. and block No. of the program to be operated automatically.
The searched program is displayed upon completion of the search.
Machining programs are stored in the memory inside the NC system.

11.3.3 Verification Stop

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

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



Verification stop is executed when drilling of one hole is completed (N100 block is completed) as shown on the left.
The hole depth is measured to confirm that the dimensions are correct. If the dimensions are not correct, the tool compensation amount is changed.
If machining is started from the head of the program, accurate hole drilling can be carried out.

11.3.4 Program Restart

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

When a machining program is to be resumed after it has been suspended midway due to tool damage or for some other reason, this function searches the program and the block to be resumed and enables machining to be resumed from the block.
There are two types of restart, type 1 and type 2.

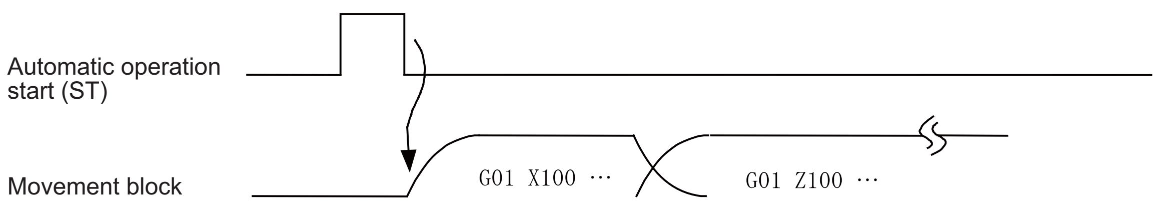
<table>
<tr><th>Restart method</th><th colspan="2">Details</th></tr>
<tr><td>Restart type 1</td><td colspan="2">After machining is reset due to a tool breakage, etc., machining is restarted from the designated sequence number and/or block number.
Only the program which had been executed just before can be restarted.
Even after the power is turned ON again, the program can be restarted if the program has been executed before the power supply is turned OFF.</td></tr>
<tr><td rowspan="3">Restart type 2</td><td colspan="2">After machining program is stopped due to a holiday, etc., and the power is turned OFF and ON, machining is restarted from the designated sequence number and/or block number.</td></tr>
<tr><td>Automatic top search OFF</td><td>A top search must be executed from the screen. Then, command a sequence No. and block No., and restart a program.</td></tr>
<tr><td>Automatic top search ON</td><td>A top search may not be executed from the screen. A top search is automatically executed, so it is not necessary to execute a top search from the screen. Command a sequence No. and block No., and restart a machining.
An arbitrary program can be restarted by designating the program No. When a program No. is omitted, the program currently searched is restarted.</td></tr>
</table>

The validity of the automatic top search can be changed with control parameter.

11.3.5 Automatic Operation Start

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function enables the controller to be reset.

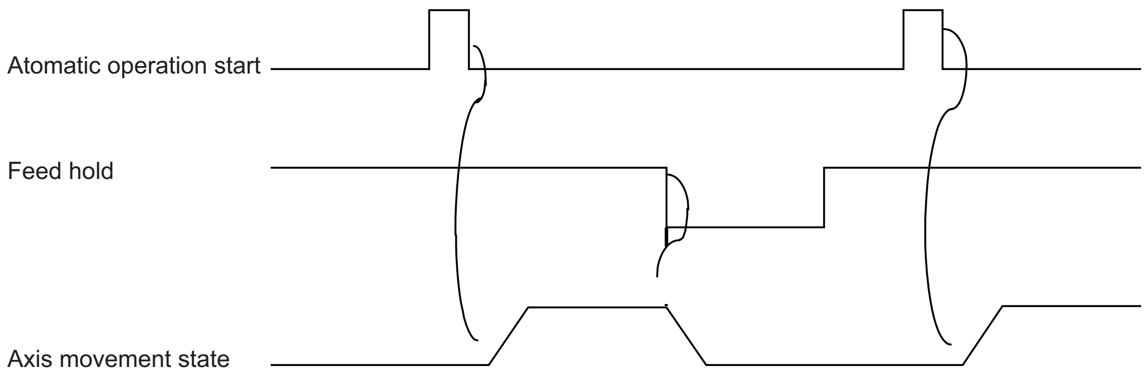
		Reset 1	Reset 2	Reset & Rewind
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	Retained
6	M code independent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

11.3.7 Feed Hold

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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



11.3.8 Search & Start

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

11.4 Interrupt Operation

11.4.1 Manual Interruption

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

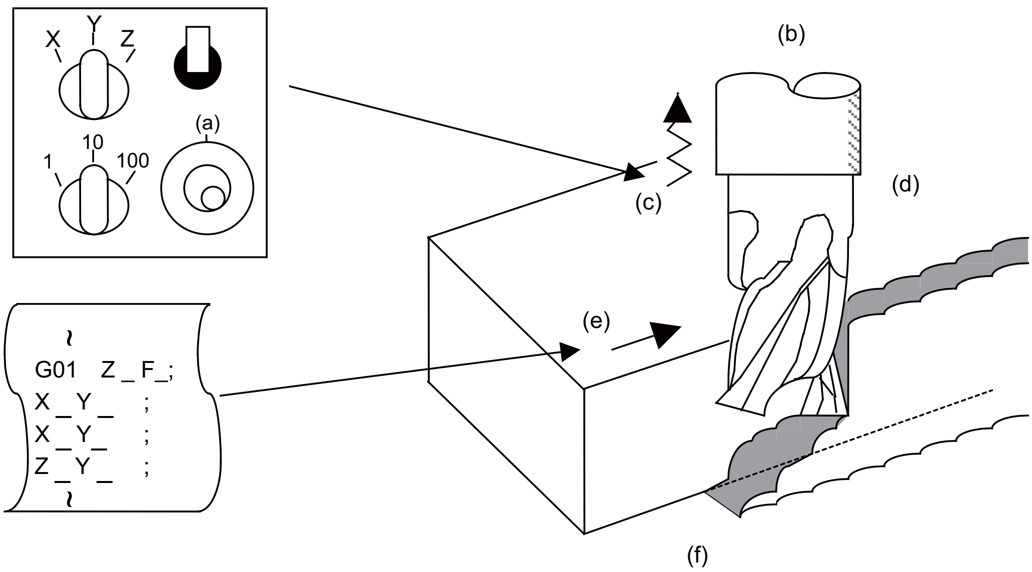
Manual interrupt is a function that enables manual operations to be performed during automatic operation.
The methods to select the operation mode are as follows:
- Initiates the interrupt by switching from the automatic mode to manual mode
- Initiates the interrupt by selecting the manual mode at the 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 Switch").

11.4.2 Automatic Operation Handle Interruption

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during 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, pulse scale factor and 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.

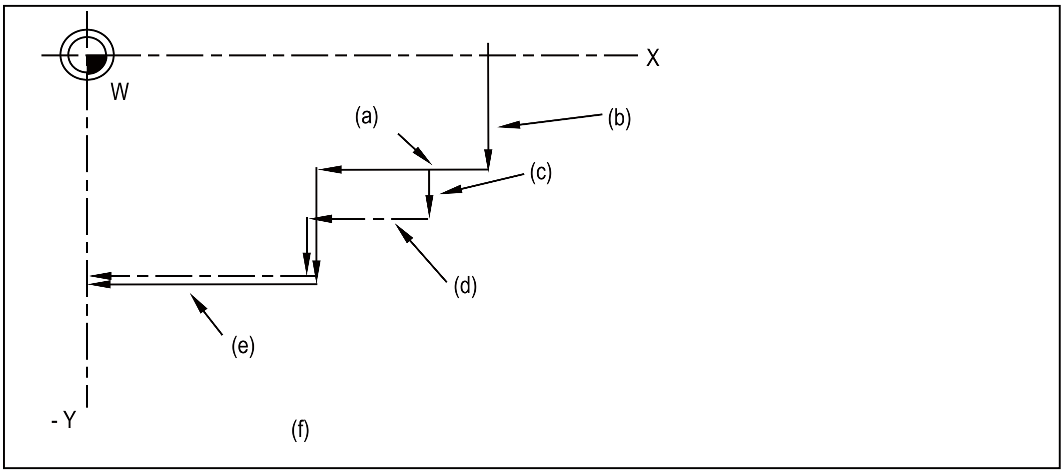


(a) Interrupt
(b) Tool
(c) Handle feed
(d) Workpiece
(e) Automatic feed
(f) Feed path with automatic feed and handle feed superimposed

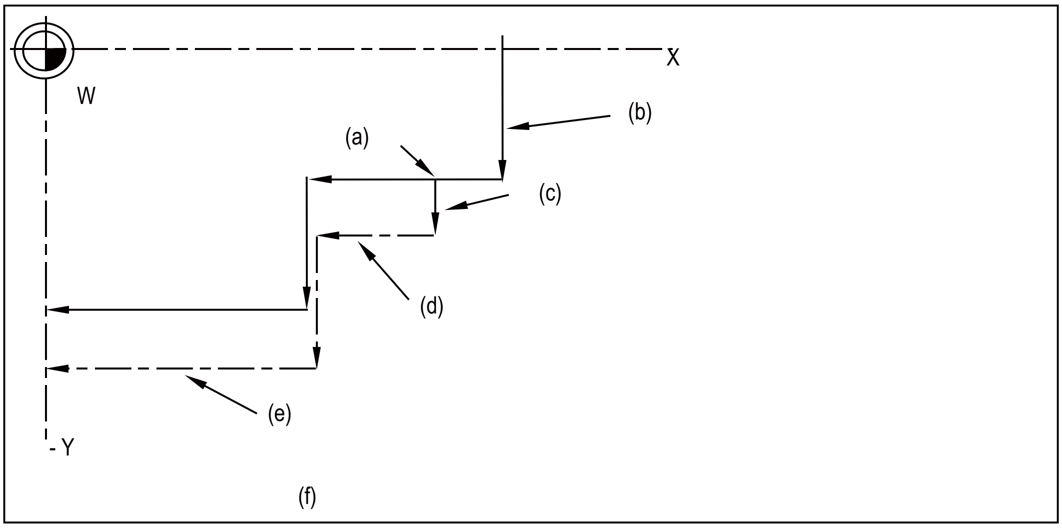
11.4.3 Manual Absolute Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The program absolute positions are updated by an amount equivalent to the distance by which the tool is moved by hand when the manual absolute switch 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.



(a) Feed hold stop
(b) Programmed path (absolute command)
(c) Manual interrupt (Program absolute position is updated by an amount equivalent to traveled value.)
(d) Path after manual interrupt
(e) Tool passes along same path as that programmed
(f) With manual absolute signal ON



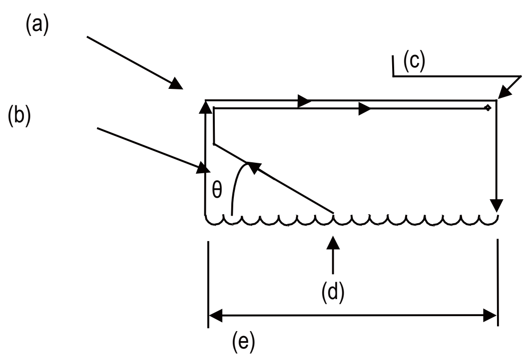
(a) Feed hold stop
(b) Programmed path (absolute command)
(c) Manual interrupt (Program absolute position is not updated even if axis moves)
(d) Path after manual interrupt
(e) Path is shifted by an amount equivalent to manual interrupt value (Zero point moves)
(f) With manual absolute signal OFF

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

11.4.4 Thread Cutting Cycle Retract

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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

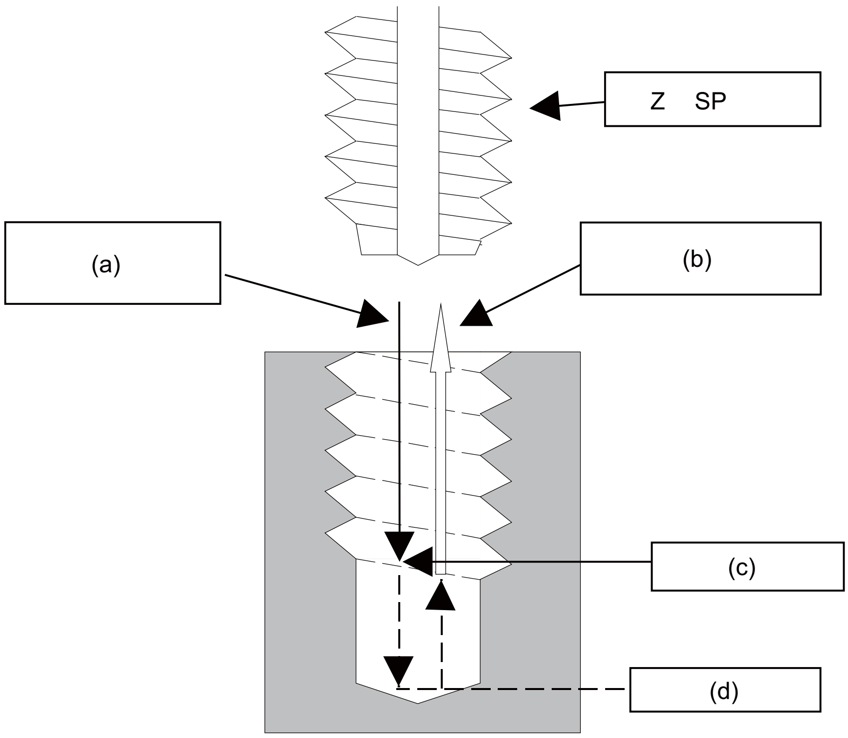


(a) Position where the block following the thread cutting is completed
(b) Chamfering angle
(c) Suspension position
(d) Feed hold
(e) Period when thread cutting is performed

11.4.5 Tapping Retract

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



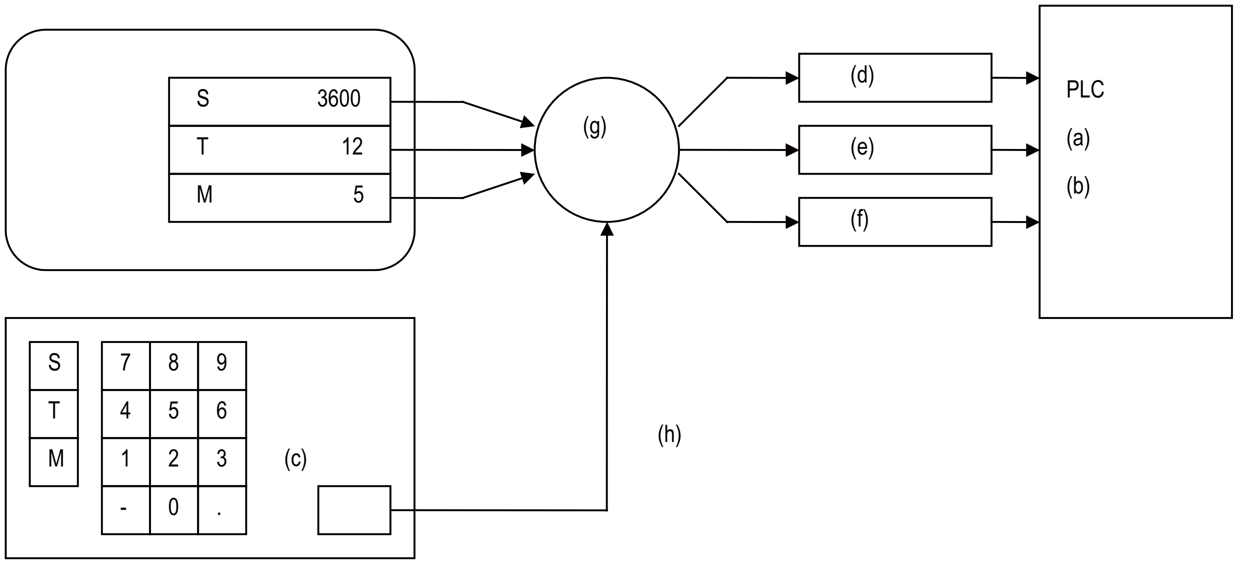
(a) Tap feed (spindle forward)
SP (spindle)
(b) Tap retract (spindle reverse)
(c) Retract signal
(d) Tap bottom

This function can be used when the machining was interrupted upon reset, emergency stop or power OFF during tapping cycle.
(Note that "power OFF" is applied only when the absolute position detection system is valid.)
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical Value Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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 compensation 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.



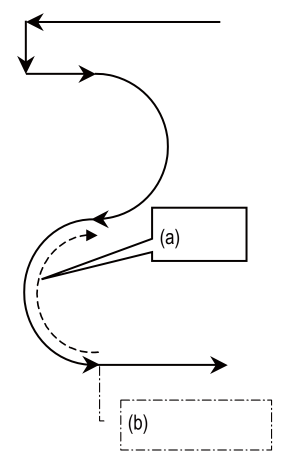
(a) sequence
(b) processing
(c) Input
(d) S command value
(e) T command value
(f) M command value
(g) Manual numerical value
(h) (Note) The input operation starts the execution of the M, S or T command.

11.4.7 Arbitrary Reverse Run

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function allows a program to run the executed blocks backward (hereinafter called "reverse run") after the block stop in the automatic operation.
The following steps are available:
- Execute the reverse run to the point to go back,
- Run the reversed blocks again following the program (hereinafter called "forward run") and
- Continue the remaining blocks from the point of the interruption.

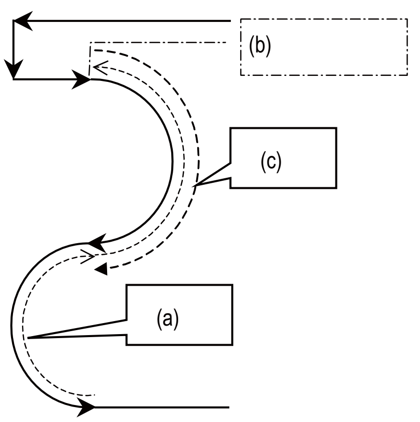
Maximum 200 blocks can be executed in the reverse run.
Only the 1st to 3rd axes of each part system can be used for this function.
(Example 1) Reverse run




```
N01 G90 G54 X0 Y0 ;
N02 G91 G01 X-50. F500 ;
N03 Y-20. ;
N04 X20. ;
N05 G02 Y-50. R25. ;
N06 G03 Y-50. R25. ;
N07 G01 X30. ;
M02 ;
```

(a) Reverse run
(b) Block stop

(Example 2) Forward run after the reverse run




```
N01 G90 G54 X0 Y0 ;
N02 G91 G01 X-50. F500 ;
N03 Y-20. ;
N04 X20. ;
N05 G02 Y-50. R25. ;
N06 G03 Y-50. R25. ;
N07 G01 X30. ;
M02 ;
```

(a) Reverse run
(b) Block stop
(c) Forward run

11.4.8 MDI Interruption

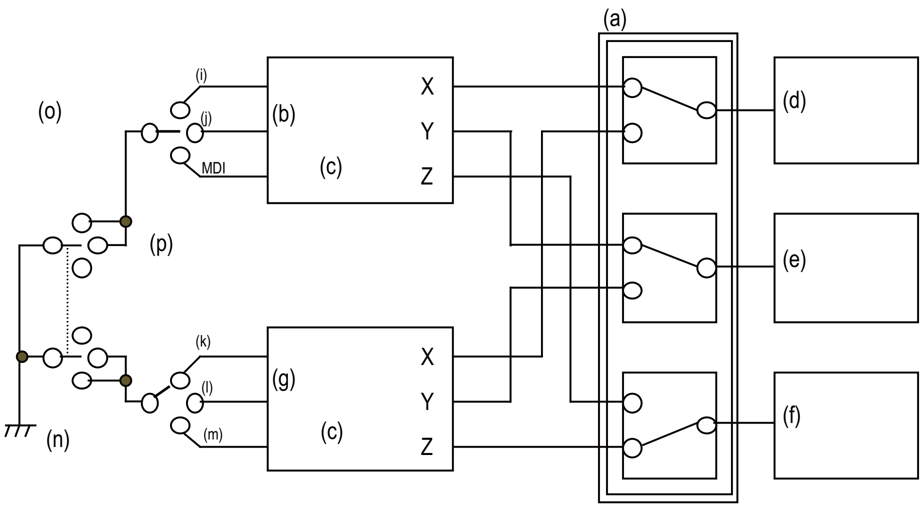
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

11.4.9 Simultaneous Operation of Manual and Automatic Modes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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



(a) Axis switching
(b) Automatic operation
(c) Axis control
(d) X-axis position control
(e) Y-axis position control
(f) Z-axis position control
(g) Manual operation
(i) Tape
(j) Memory
(k) Jog
(l) Handle
(m) Return
(n) Manual mode
(o) Automatic mode
(p) Simultaneous manual and automatic operation

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 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 applied to axes subject to automatic commands.

11.4.10 Simultaneous Operation of JOG and Handle Modes

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

11.4.11 Reference Position Retract

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

(1) Other operations

(a) When the retract signal is turned ON, the control unit is reset, the operation is interrupted, and the machining program is indexed.

(b) When the rapid traverse input signal is input, the rapid traverse rate is applied. When the rapid traverse input signal is not input, the manual feed rate is applied.

(c) If the retract signal is input during execution of a tapping cycle, the operation will be the tapping retract, and the normal reference position retract will be executed from the end point of tapping retract operation.

(d) Even if the retract signal is input during the thread cutting cycle, it will be invalid. However, if the retract signal is input in a block other than the thread cutting block, the retracting operation will be executed.

(e) If the retract signal is turned OFF midway during retracting, the operation will decelerate and stop. However, since the machining program is indexed, the block 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.

11.4.12 Tool Escape and Return

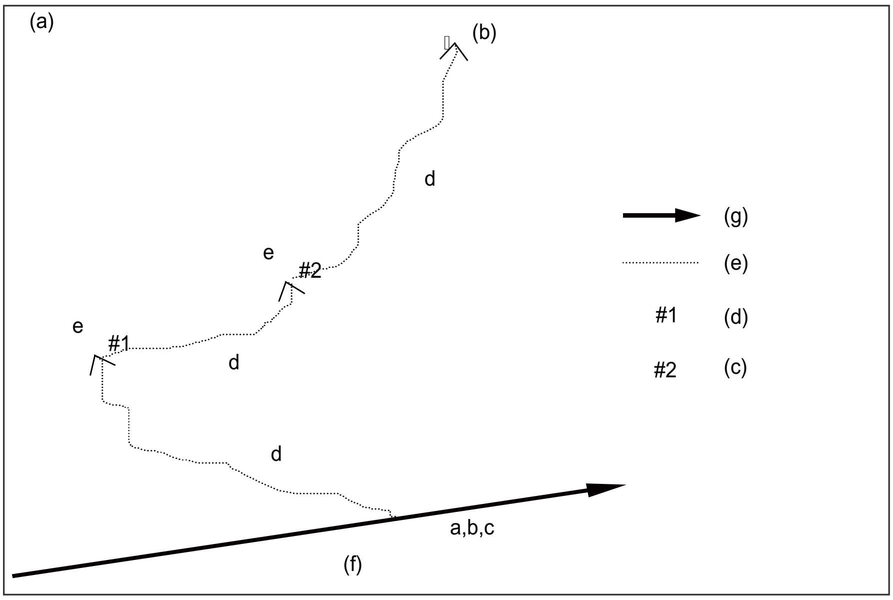
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

Even if the machining program's operation is halted and the tool is escaped to change the tool or check the workpiece, etc., the tool can be returned to the halted point (machining halted point) and machining resumed.

The two main functions are as follow.

- After the machining program is halted, the halt point is designated, and the tool is escaped manually.
  The transit points for returning are designated when escaping.
- The tool returns automatically.

When returning, the tool passes through the transit points designated when escaping and returns to the halt point. Then, machining restarts.



a. Machining program halt
b. Halt point designation
c. Operation mode changeover
d. Tool escape
e. Transit point designation
(a) Fig. 1
(b) Return start point
(c) No. 2 transit point
(d) No. 1 transit point
(e) Escape path
(f) Halt point
(g) Tool path

11.4.13 Skip Retract

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function is used to return to the opposite advance direction when the skip signal is input during G31 command.

11.4.14 PLC Interruption

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

12

Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

12.1.1.1 Subprogram Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ 8layers	○ 8layers	○ 8layers	○ 8layers	○ 8layers	○ 8layers
L system	○ 8layers	○ 8layers	○ 8layers	○ 8layers	○ 8layers	○ 8layers

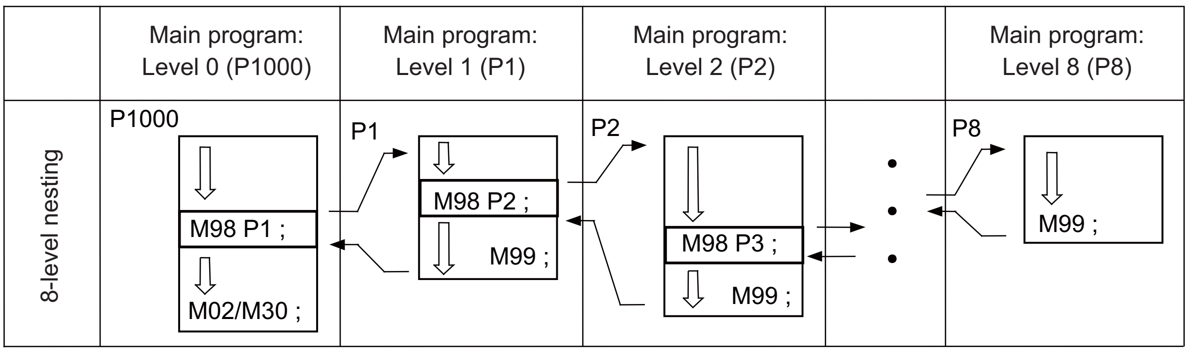
When the same pattern is repeated during machining, the machining pattern can be registered as one subprogram and the subprogram can be called from the main program as required, thereby realizing the same machining easily. Efficient use of program can be made. The subprogram is called with the program No. and sequence No. or the file name and sequence No.

(1) Calling the subprogram with M98 command

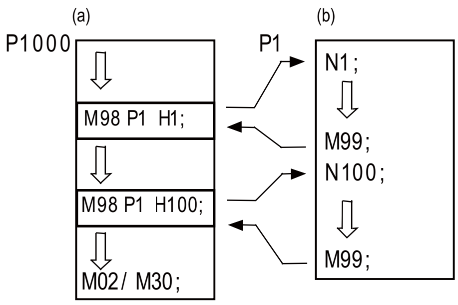
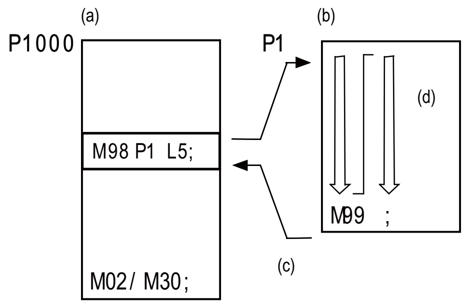
M98 Pp1 Hh1 Ll1 ,Dd1 ;
or, M98 <File name> Hh1 Ll1 ,Dd1 ;

M98	: Subprogram call command
Pp1	: Program No. in subprogram to be called. (Own program if omitted.) Note that P can be omitted only during memory operation and MDI operation. (Max. 8-digit value)
<File name>	: A file name can be designated instead of the program No. In this case, enclose the file name with brackets <>. (The file name can have up to 32 characters, including the extension.) (Example) M98 <PARTS-12.RAF>;
Hh1	: Sequence No. in subprogram to be called (Head block if omitted) (Max. 6-digit value)
Ll1	: Number of subprogram repetitions (If omitted, this is handled as L1. When L0 is set, the subprogram is not called.) (1 to 9999 times set with 4-digit value.)
,Dd1	: Device No. where subprogram is stored. (0 to 4) If ,D is omitted, the subprogram in the memory will be used. The device No. is set with the machining parameters.

Subprograms can be nested up to eight levels deep.



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

<table>
<tr><th>Specifying a subprogram branch destination</th><th>Specifying repetition of a subprogram</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

(2) Calling the subprogram with M198 command

A program registered in the data server (IC card in control unit) can be called out as a subprogram. To call a program in the data server as the subprogram, command as shown below in the main program.

M198 Pp1 Ll1 ;
or, M198 <File name> Ll1 ;

M198	: Call command
Pp1	: Program No. in data server to be called as subprogram (Max. 8 digits)
<File name>	: A file name can be designated instead of the program No. In this case, enclose the file name with brackets <>. (The file name can have up to 32 characters, including the extension.)
Ll1	: Number of subprogram repetitions (Max. 4 digits) This can be omitted. (If omitted, this is handled as L1.) When L0 is set, the subprogram is not called.

(Note 1) Sequence No. call (M198 H****) is not possible.

(Note 2) Calling of the subprogram with the M198 command is limited to once in the subprogram nest. The subprogram can be called only from the memory or MDI program.

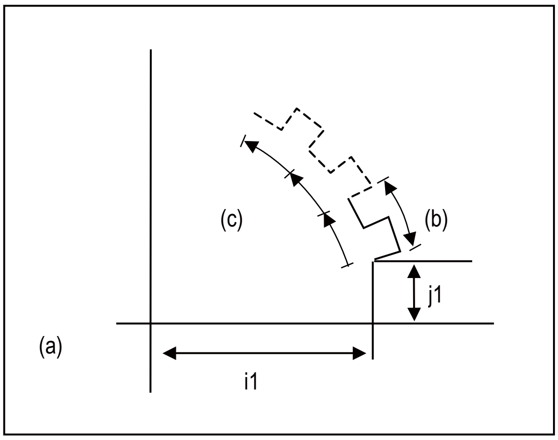
(Note 3) The data from the head of the program to the first LF (lien feed code. 0x0A as hexadecimal) is invalid, and cannot be run or displayed. Note that if the head starts from an O number (program No.), the program is valid from the head.

12.1.1.2 Figure Rotation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

If the same pattern is used repeatedly on a concentric circle, one of the rotating machining patterns can be registered as a subprogram. When the subprogram is called from the main program, if the rotation center is designated, a path similar to the rotary phase can be easily created on the concentric circle. This simplifies creation of the program.
The program format is shown below.

M98 Ii1 Jj1 (Kk1) Pp1 Hh1 Ll1 ,Dd1;	
or, M98 Ii1 Jj1 (Kk1) <File name> Hh1 Ll1 ,Dd1 ;	
M98	: Subprogram call command
Ii1 Jj1 (Kk1)	: Rotation center coordinates
Pp1	: Program No. in subprogram to be called. (Own program if omitted.) Note that P can be omitted only during memory operation and MDI operation. (Max. 8-digit value)
<File name>	: A file name can be designated instead of the program No. In this case, enclosed the file name with brackets <>. (The file name can have up to 32 characters, including the extension.) (Example) M98 <PARTS-12.RAF>;
Hh1	: Sequence No. in subprogram to be called (Head block if omitted) (Max. 6-digit value)
Ll1	: Number of subprogram repetitions (If omitted, this is handled as L1. When L0 is set, the subprogram is not called.) (1 to 9999 times set with 4-digit value.)
,Dd1	: Subprogram device No. (0 to 4) The subprogram in the memory can be used when ,D is omitted. The device No. is set with the machining parameters.



(a) Rotation center
(b) Basic figure
(c) l1 times

(1) The first subprogram called out with subprogram call is executed at a 0° rotation angle. The path is created as commanded.
(2) If the number of repetitions is set to two or more, the rotation angle is obtained from the called subprogram's start point, end point and rotation center coordinate. The path of the first subprogram is used as a basic figure and is rotated and arranged for the designated number of call repetitions, using the rotation center coordinates as a reference.
(3) During figure rotation, all blocks in the subprogram are treated as to be rotated.
(4) If the subprogram start point and end point are not on the same circle having the commanded figure rotation center coordinates as the center, the axis will interpolate using the subprogram's end point as the start point, and the end point in the first movement command block in the rotated subprogram as the end point.
(5) The figure is rotated on the workpiece coordinate system, and can be shifted with the G92, G52, G54 to G59 (workpiece coordinate system shift) command.
(6) Functions (reference position return, uni-direction positioning, etc.) on the machine coordinate system for the rotary plane axis cannot be used while the figure is rotated. However, the machine coordinate system functions can be used for axes other than the rotation plane.
(7) A program error will occur if figure rotation is commanded during figure rotation.
(8) Figure rotation and program coordinate rotation cannot be commanded simultaneously.

12.1.1.3 Scaling

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

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

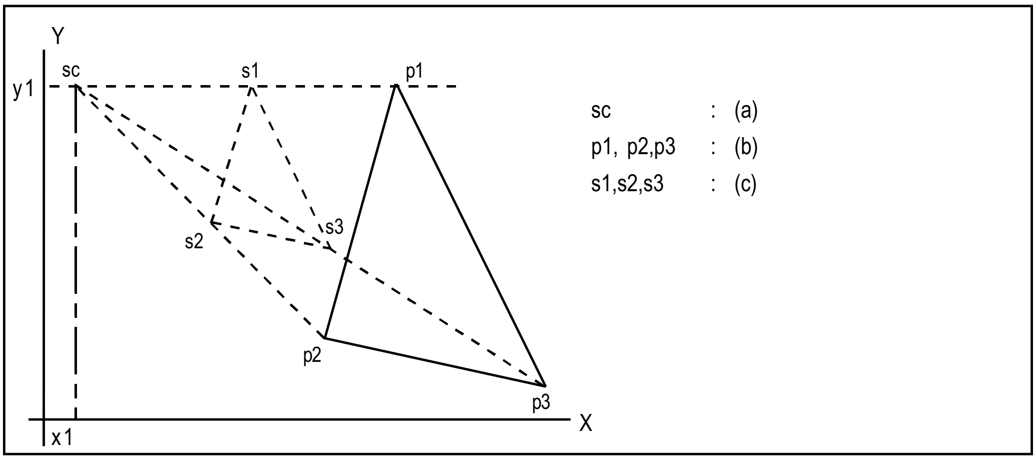
Gcode	Function
G50 G51	Scaling cancel Scaling ON

The program format is given below.

G51 Xx1 Yy1 Zz1 Pp1 ;

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

The scale factor ranges from 0.000001 to 99.999999 times.



(a) Scaling center
(b) Programmed shape
(c) Shape after scaling

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

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

12.1.1.4 Axis Name Switch

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

The axis name switch function switches the name of command axis and control axis.

G111 Axis name1 Axis name2 ;	
G111	: Axis name switch command
Axis name 1,2	: Axes to be performed the name switch command When the axis name switch command is canceled, assign only "G111;" command.

When switching the name of X-axis and Y-axis by axis name switch command, the machine performs as follow.

<Before axis name switch>			<After axis name switch>	
Command axis name	**Control axis**		**Command axis name**	**Control axis**
X	**Xaxis**	**->**	**X**	**Yaxis**
Y	**Yaxis**		**Y**	**Xaxis**

(Example)
G111 X Y ;
G01 X100. ; (Y axis moves to 100.mm)
G01 Y100. ; (X axis moves to 100.mm)

12.1.2 Macro Program

12.1.2.1 User Macro

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 4layers	△ 4layers	△ 4layers	△ 4layers	△ 4layers	△ 4layers
L system	△ 4layers	△ 4layers	△ 4layers	△ 4layers	△ 4layers	△ 4layers

(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--;or,G65 <File name> Ll1 --Argument--;

G65	: Call command
Pp1	: Program No.
<File name>	: File name When designating the file name, enclose the file name with brackets < >.
Ll1	: Number of repetitions
Argument	: Variable data assignment

The macro program is called immediately by this command

G66 Pp1 Ll1 --Argument--;or,G66 <File name> Ll1 --Argument--;

G66	: Call command
Pp1	: Program No.
<File name>	: File name When designating the file name, enclose the file name with brackets < >.
Ll1	: Number 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--;or, G66.1 <File name> Ll1 --Argument--;

G66.1	: Call command
Pp1	: Program No.
<File name>	: File name When designating the file name, enclose the file name with brackets < >.
Ll1	: Number 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 1) 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

G Argument;**	
G**	: G code for performing macro call
Argument	: Variable data designation

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

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

(Note 1) G101 to G110 and G200 to G202 are user macro I codes. However, if a parameter is set for the G code call code, the G code call will have the priority, and these cannot be used as the user macro I.

(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.)

Format

M (or S**;, T**;, B**;)**	
M** (S**, T**, B**)	: M (or S, T, B) code for performing macro call

The correspondence between the M** 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 other than the codes basically required by the machine and M codes of 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 (or S, T, B) code macro can be done independently for each part system.

12.1.2.2 Machine Tool Builder Macro

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function enables macro programs exclusively designed for use by the machine builders to be registered in addition to the regular user macro programs.
As opposed to the conventional macro program, the macro program display, editing and input/output can be protected with a password. This is suitable for maintaining the machine tool builder's original macro program confidentiality.
Machine tool builder macro programs are stored in a dedicated area which means that the user program registration area is not reduced in the process.
When registering a machine tool builder macro, secure its space by formatting.
128KB of registration area is secured in the machine tool builder macro program, and up to 500 programs can be registered there.
(For M720, registration area is 64KB and the registerable number of programs is 100.)
The macro program registered as a machine tool builder macro is called out as "G65 command" or "G code macro equivalent to G65". The macro program is described in the same manner as a conventional machining program.
Variables for the machine tool builder macros can be used within the machine tool builder macros. These variables can be used commonly within the machine tool builder macro programs.

Call format 1

G65 Pp Ll Argument ;	
p	: Machine tool builder macro program No.(O100010000 to O199999998)
l	: Number of repetitions

(Note) The machine tool builder macro program cannot be called with G66, G66.1 or M98. Note that a machine tool builder macro program can be called from a machine tool builder macro program using M98.

Call format2

G* Argument;**	
	: G code defined in macro definition program
***	: Variable data designation
Argument	Argument is used when a local variable must be delivered to the macro program. Designate a real value after the address.

(Note) When calling the macro with the G code, the call program is defined with the macro definition program or setup parameter. However, if the call program is defined in both the parameters and macro definition program, the parameter definition will have the priority.

12.1.2.3 Macro Interruption

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

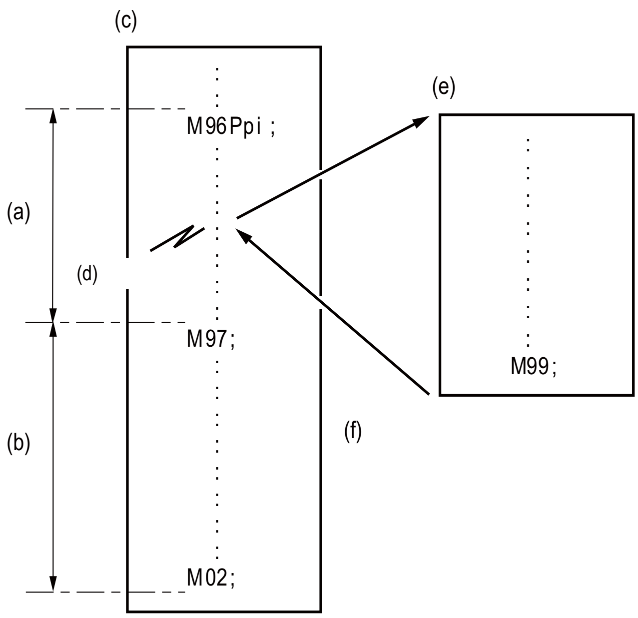
By inputting a user macro interrupt signal from the PLC, the program being currently executed is interrupted and other programs can be called instead.
Retract or return operations when tools have been damaged, for instance, and other kinds of restoration operations to be conducted when trouble has occurred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

[Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.
[Interrupt type 2] After the block being executed is complete, the interrupt program is executed.

The command format is given below.

M96 Pp1 Hh1 ; or, M96 <File name> Hh1 ;	**User macro interrupt valid**
M96	: Interrupt valid command
Pp1	: Interrupt program number
<File name>	: File name When designating a file name, enclose the file name in brackets <>.
Hh1	: Interrupt sequence No.

M97 ;	**User macro interrupt invalid**
M97	: User macro interrupt end command



(a) The user macro interrupt signal is accepted during this period.
(b) The user macro interrupt signal is not accepted during this period.
(c) Machining program Opm:
(d) Interrupt signal
(e) Interrupt program Opi
(f) The modal information is restored to the status applying before interrupt.

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

Type			Number		Function
Common variables			Common variables 1	Common variables 2	Can be used commonly for main, sub and each macro program.
	For 1-part system specifications	100 sets	500 to 549	100 to 149	
		200 sets	500 to 599	100 to 199	
		300 sets	500 to 699	100 to 199	
		600 sets	500 to 999	100 to 199	
		700 sets	400 to 999 100100 to 800199	100 to 199	
		8000 sets	400 to 999 100100 to 800199 900000 to 907399	100 to 199	
	For multi-part system specifications	50+ 50 sets	500 to 549	100 to 149 × number of part systems	
		100+100 sets	500 to 599	100 to 199 × number of part systems	
		200+100 sets	500 to 699	100 to 199 × number of part systems	
		500+100 sets	500 to 999 100100 to 800199	100 to 199 × number of part systems	
		600+100 sets	400 to 999 100100 to 800199	100 to 199 × number of part systems	
		7900+100 sets	400 to 999 100100 to 800199 900000 to 907399	100 to 199 × number of part systems	
Local variables			1 to 33		Can be used as local in macro program.
System variables			1000 to		Application is fixed in system.
Fixed cycle variables			1 to 32		Local variables in fixed cycle program.

(Note 1) All common variables are held even when the power is turned OFF.
(Note 2) The common variables can be emptied by resetting or turning the power OFF when the parameters are set accordingly.
(Note 3) The common variables are divided into the following two types.
Common variable 1: Variables that can be used commonly through the part systems.
Common variable 2: Variables that can be used commonly within that part system's program.
(Note 4) Variable names can be set for #500 to #519

Variable expressions
Variable : #Numerical value #100
(Numerical value: 1,2,3, ...)
: # [Expression] #100
Expression : Numerical value
: Variable
: Expression Operator Expression #100 + #101
: - (minus) Expression -#120
: [Expression] [#110]
: Function [Expression] SIN [#110]
Variable definition
Variable = expression
(Note1) Variables cannot be used with addresses "O" and "N".

12.1.2.4.1 100 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

12.1.2.4.2 200 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.3 300 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.4 600 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.5 700 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.6 8000 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.7 (50+50 × Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

12.1.2.4.8 (100+100 × Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.9 (200+100 × Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.10 (500+100 × Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.11 (600+100 × Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.2.4.12 (7900+100 x Number of Part Systems) Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

12.1.3 Fixed Cycle

List of fixed cycles

Type of fixed cycle	M system	L system						Remarks
	Gcode list 1	Gcode list 2	Gcode list 3	Gcode list 4	Gcode list 5	Gcode list 6	Gcode list 7	
Fixed cycle for drilling	G70	G80	G80	G80	G80	G80	G80	Refer to 12.1.3.1. Refer to 12.1.3.2 (Type II). Refer to 4.5.3.
	:	:	:	:	:	:	:	
	G89	G89	G89	G89	G89	G89	G89	
		G79	G83.2	G79	G83.2	G79	G83.2	
	G98	G98	G98	G98	G98	G98	G98	
	G99	G99	G99	G99	G99	G99	G99	
Special fixed cycles	G34							Refer to 12.1.3.3.
	G35	-	-	-	-	-	-	
	G36							
Fixed cycles for turning machining	-	G90	G77	G90	G77	G90	G77	Refer to 12.1.3.4.
		G92	G78	G92	G78	G92	G78	
		G94	G79	G94	G79	G94	G79	
Compound-type fixed cycles for turning machining	-	G70	G70	G70	G70	G70	G70	Refer to 12.1.3.5. Refer to 12.1.3.6 (Type II). Refer to 12.1.8.8.
		:	:	:	:	:	:	
		G76	G76	G76	G76	G76	G76	
		G76.1	G76.1	G76.1	G76.1	G76.1	G76.1	
		G76.2	G76.2	G76.2	G76.2	G76.2	G76.2	
Small-diameter deep-hole drilling cycle	G83	-	-	-	-	-	-	Refer to 12.1.3.7.

12.1.3.1 Fixed Cycle for Drilling

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(1) 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.

Gcode	Function
G70	
G71	
G72	
G73	Step cycle
G74	Reverse tapping cycle
G75	Perfect circle cutting cycle
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.

Gcode	Function
G98	Initial point level return
G99	R point level return

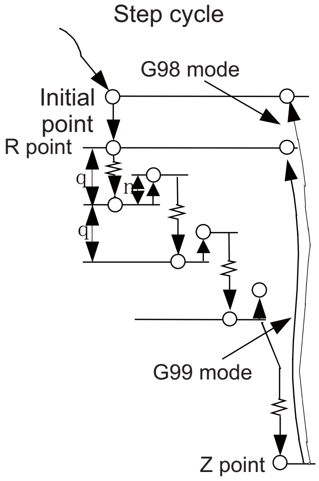
The basic program format for the fixed cycle commands is shown below.

G81 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ll1 Ff1 ;

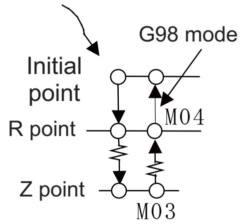
G81	: Hole drilling mode
Xx1,Yy1	: Hole position data; X-axis, Y-axis hole drilling position command (rapid traverse) (incremental/absolute)
Zz1	: Hole machining data; Hole bottom position designation (incremental/absolute)
Rr1	: Hole machining data; Hole R point designation (incremental/absolute)
Qq1	: 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 (Not executed when "0" is set.)
Ff1	: Cutting feed rate, Z-axis feed amount (tapping pitch) per spindle rotation

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

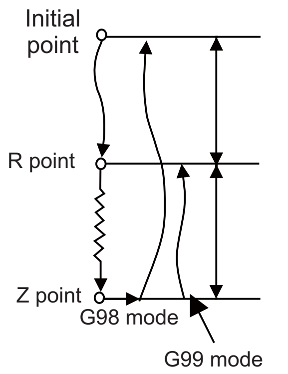
G73
Step cycle



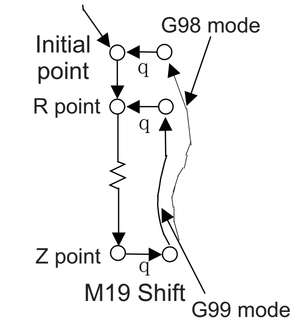
G74
Reverse tapping cycle



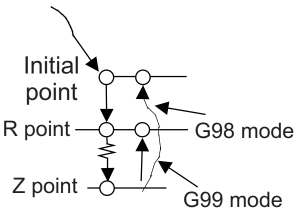
G75
Perfect circle cutting cycle



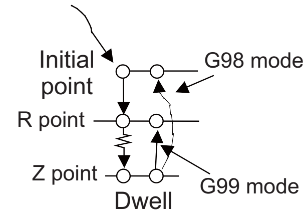
G76
Fine boring cycle



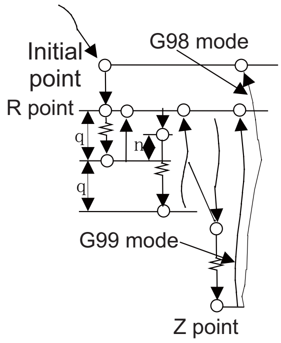
G81
Drilling, spot drilling cycle



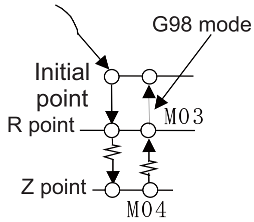
G82
Drilling, counterboring cycle



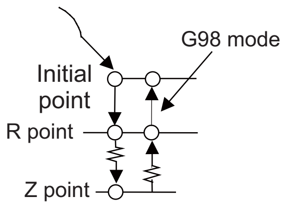
G83
Deep hole drilling cycle



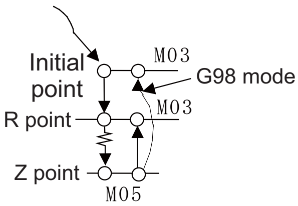
G84
Tapping cycle



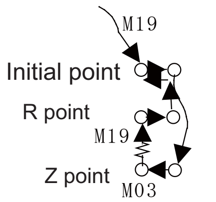
G85
Boring cycle



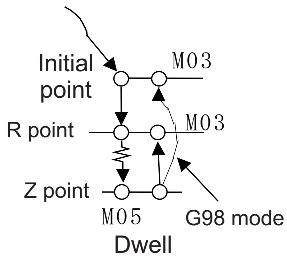
G86
Boring cycle



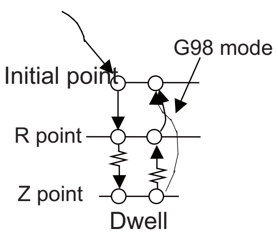
G87
Back boring cycle



G88
Boring cycle



G89
Boring cycle



(2) L system: G83 to G89, G80

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

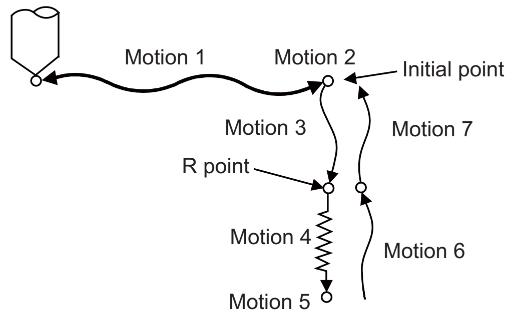
Gcode	Drilling axis	Drilling work start	Motion at hole bottom	Retract motion	Use
G80					Cancel
G83	Z	Cutting feed Intermittent feed	In-position check Dwell	Rapid traverse feed	Deep-hole drilling cycle1
G84	Z	Cutting feed	In-position check Dwell Spindle CCW	Cutting feed	Tapping cycle (Reverse tapping 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 G80 or some G command of the G01 group is issued. Data is also cleared to zero simultaneously.

Command format

G83/G84/G85 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1(Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ; **G87/G88/G89 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1(Mm1) Ss1 ,Ss1 Dd1 ,Rr1 ;**	
G83/G84/G85 G87/G88/G89	: Fixed cycle mode of drilling (G83, G87), tapping (G84, G88), or boring (G85, G89) These commands are modal. Once one of them is given, it is effective until another command of the same modal group or cancel command for drilling fixed cycle 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, Qq1, 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.
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 tapping 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.
(Motions 6 and 5 may be conducted as a single motion depending on the fixed cycle mode.)

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

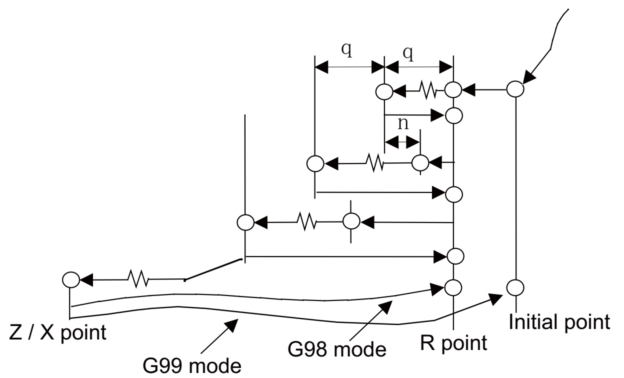
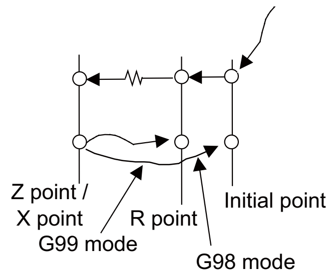
Whether the fixed cycle is complete with motion 6 or 7 can be specified by using either of the following G commands:

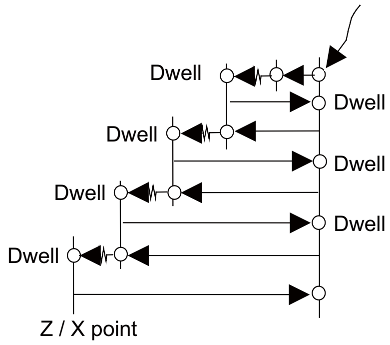
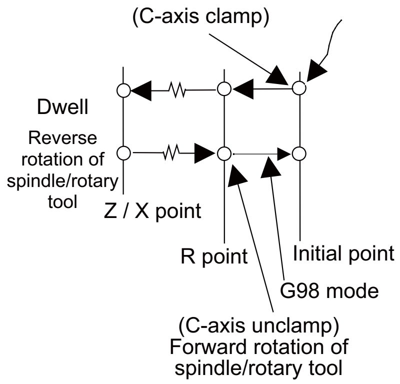
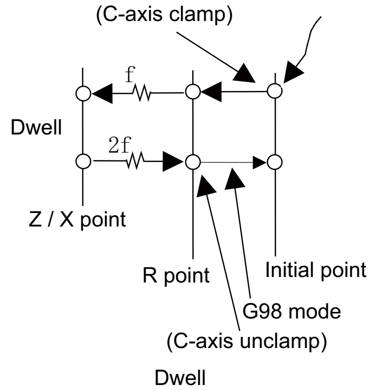
G98 : Initial level return
G99 : R point level return

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

Gcode	Function
G98	Initial point level return
G99	R point level return

12.1.3.2 Fixed Cycle for Drilling (Type II)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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

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

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

12.1.3.3 Special Fixed Cycle

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

These functions enable drilling, tapping and other hole machining cycles to be assigned in a simple 1-block program.
Special fixed cycles must always be used in combination with fixed cycles.

The special fixed cycles are as follows:

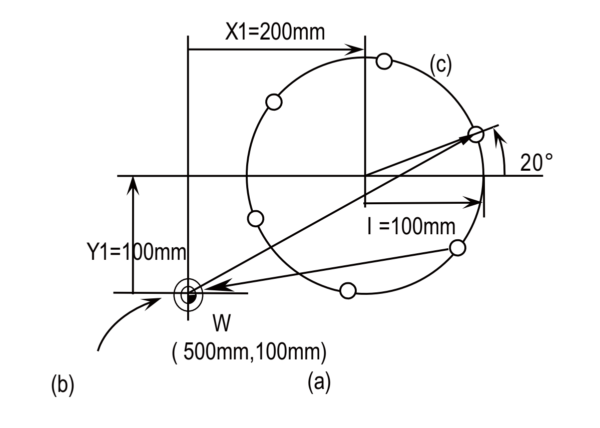
Gcode	Function
G34	Bolt hole circle
G35	Line at angle
G36	Arc
G37.1	Grid

(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 command 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 command 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 ;

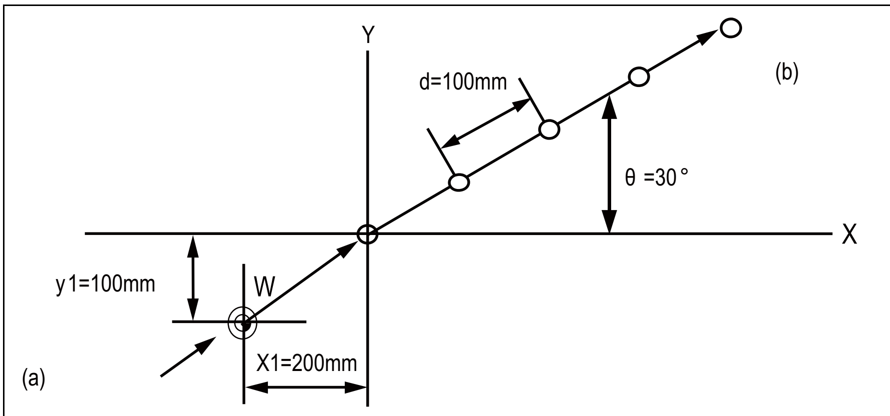
(a) G0 command in N005
(b) Position prior to execution of G34 command
(c) n = 6 holes

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, if the command is issued with incremental position, it will be necessary to calculate the coordinate position. Thus, it is convenient to issue the command with absolute position.

(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 command increment and when "d" is negative, drilling proceeds in the point symmetrical direction centered on the starting point.
Jθ	: Angle θ ; the counterclockwise direction is taken to be positive.
Kn	: Number "n" of holes to be drilled including the starting point; any number of holes from 1 through 9999 can be assigned.



(Example)
With 0.001 mm least command increment
G91;
G81 Z-10.000 R5.000 L0 F100 ;
G35 X200.000 Y100.000 I 100.000 J30.000 K5 ;

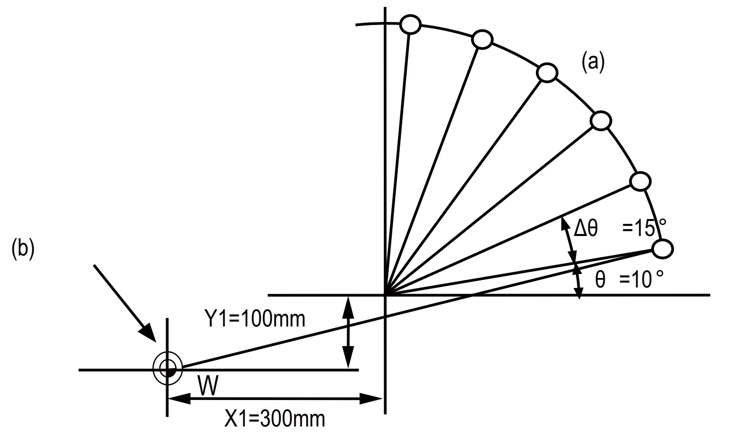
(a) Position prior to execution of G35 command
(b) N=5 holes

(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 $\triangle\theta$. 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 P△θ 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 command 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.
P△θ	: Angle interval △θ; 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.



(Exampl)
With 0.001 mm least command increment
N001 G91;
N002 G81 Z-10.000 R5.000 F100 ;
N003 G36 X300.000 Y100.000 I300.000 J10.000 P15.000 K6 ;

(a) n=6 holes

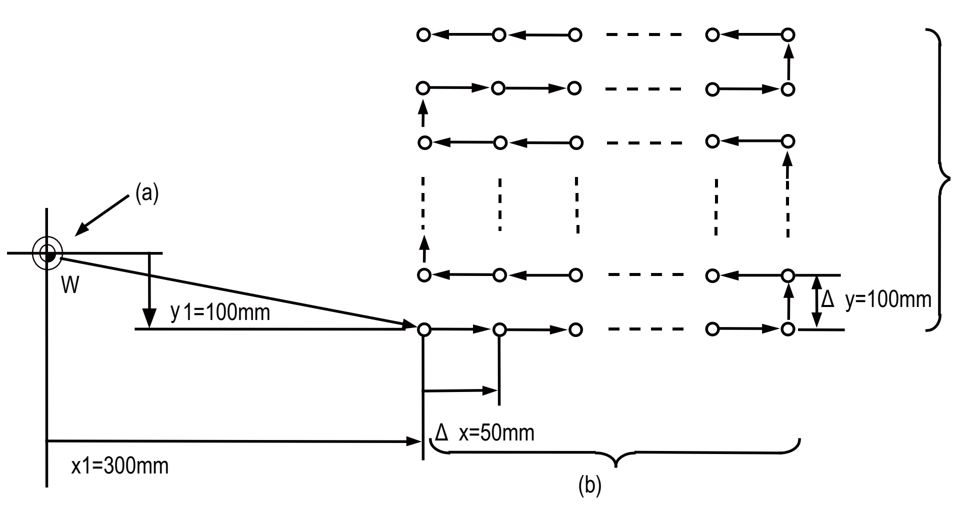
(b) Position prior to execution of G36 command

(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 △ x 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 Xx Yy I △ x Pnx J △ y Kny ;	
Xx,Yy	: The starting point coordinates; they are affected by the G90/G91 commands.
I △ x	: X-axis interval △ x; it is based on the least command increment; when △ x 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.
J △ y	: Y-axis interval △ y; it is based on the least command increment; when △ y 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 command increment
G91;
G81 Z-10.000 R5.000 F20 ;
G37.1 X300.000 Y-100.000 I50.000 P10
J100.000 K8 ;



(a) Position prior to execution of G37.1 command
(b) nx=10 holes
(c) ny=8 holes

12.1.3.4 Fixed Cycle for Turning Machining

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

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

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

Format

G △△ X/U __ Z/W __ I __ K __ R __ F __ (G18 plane)

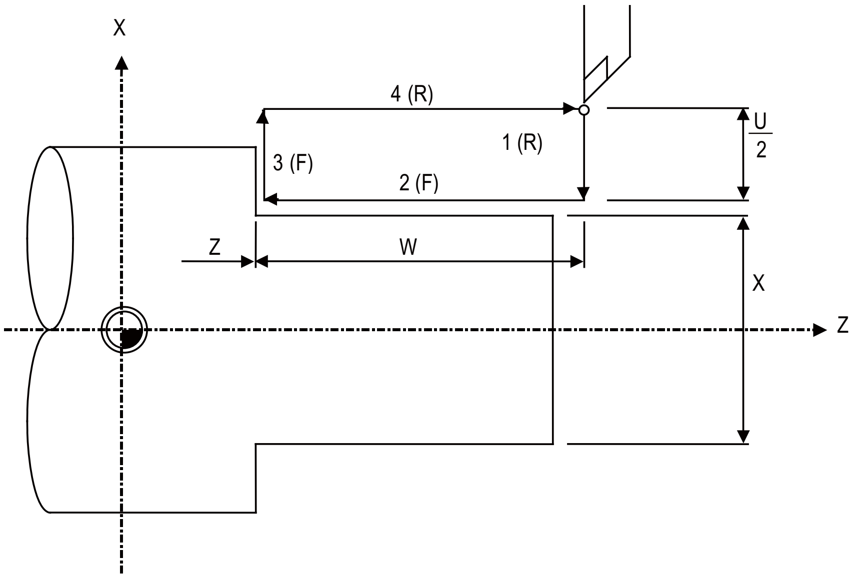
Each fixed cycle command for turning machining is a modal G code and is effective until another command of the same modal group or a cancel command is given.
The fixed cycle can be canceled by using any of the following G codes:
G00,G01,G02,G03
G09
G10,G11
G27,G28,G29,G30
G31
G33,G34
G37
G92
G52,G53
G65

(1) Longitudinal cutting cycle (G77)

(a) Straight cutting

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

```
G77 X/U_ Z/W_ F_ ;
```

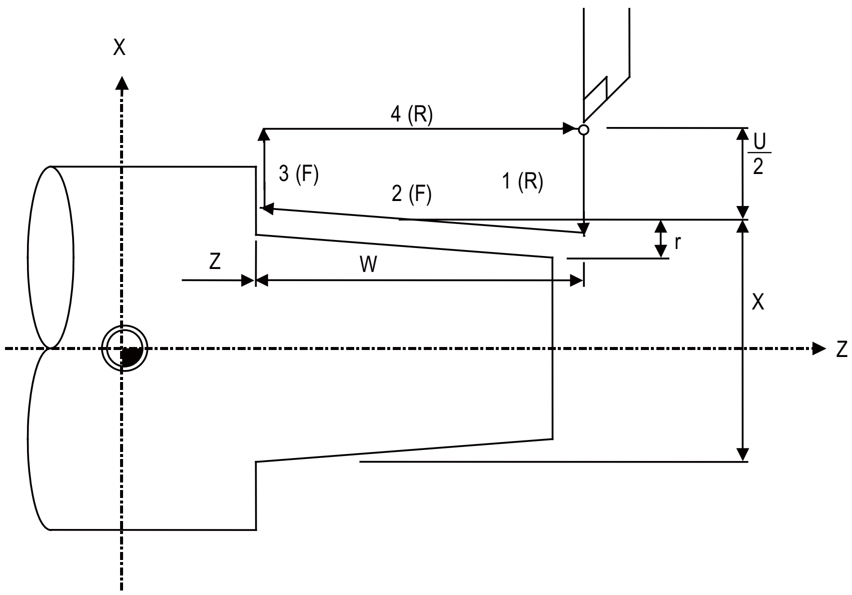


(F) Cutting feed
(R) Rapid traverse feed

(b) Taper cutting

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

```
G77 X/U_ Z/W_ R_ F_ ;
```



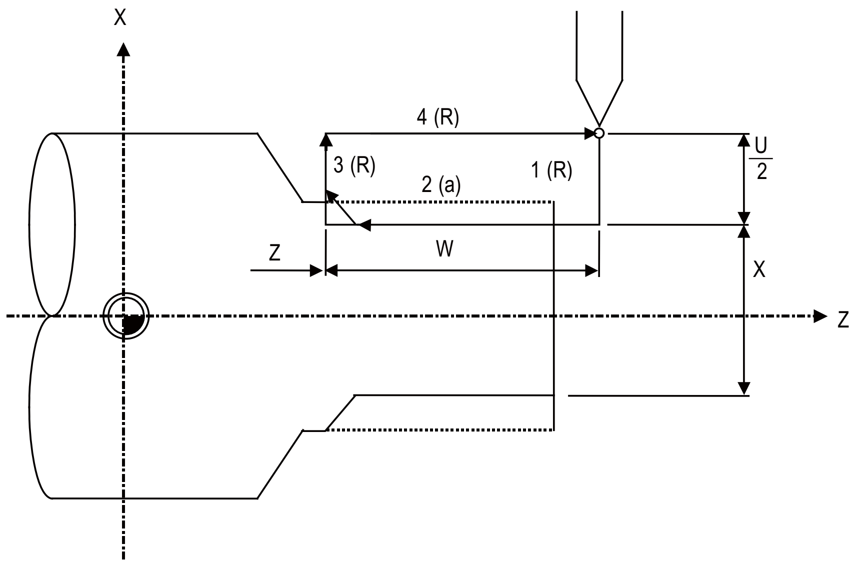
(F) Cutting feed
(R) Rapid traverse feed
r: Taper part depth (radius designation, incremental position, sign is required)

(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_ ;

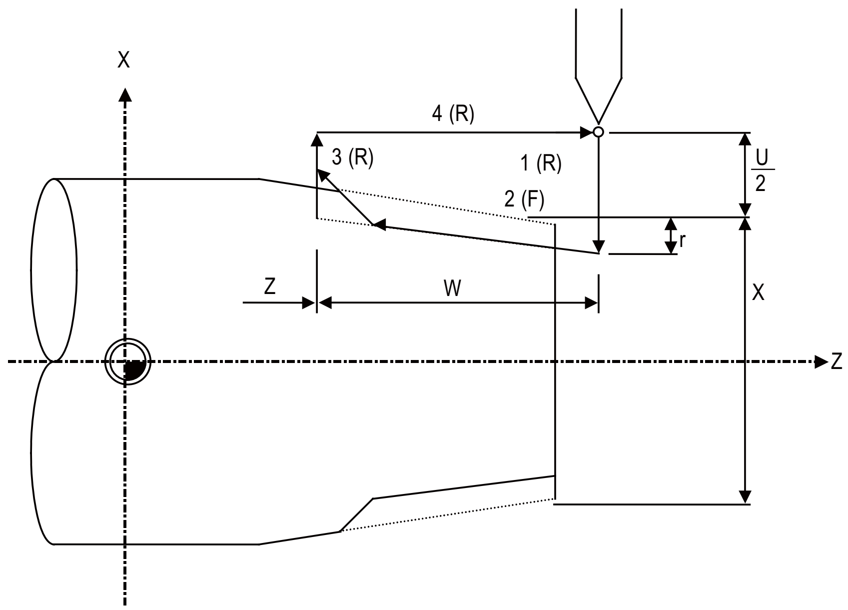


(a) F or E code designation
(R) Rapid traverse feed

(b) Taper thread cutting

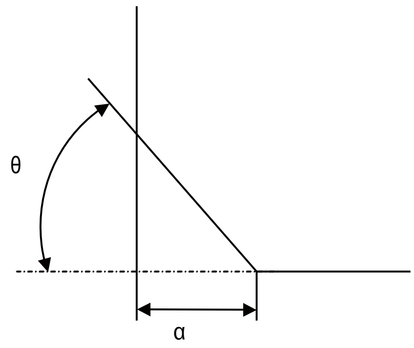
Taper thread cutting can be performed by the following block:

G78 X/U_ Z/W_ R_ F/E_ ;



(R) Rapid traverse feed
(F) F or E code designation
r : Taper part depth (radius designation, incremental position, 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**

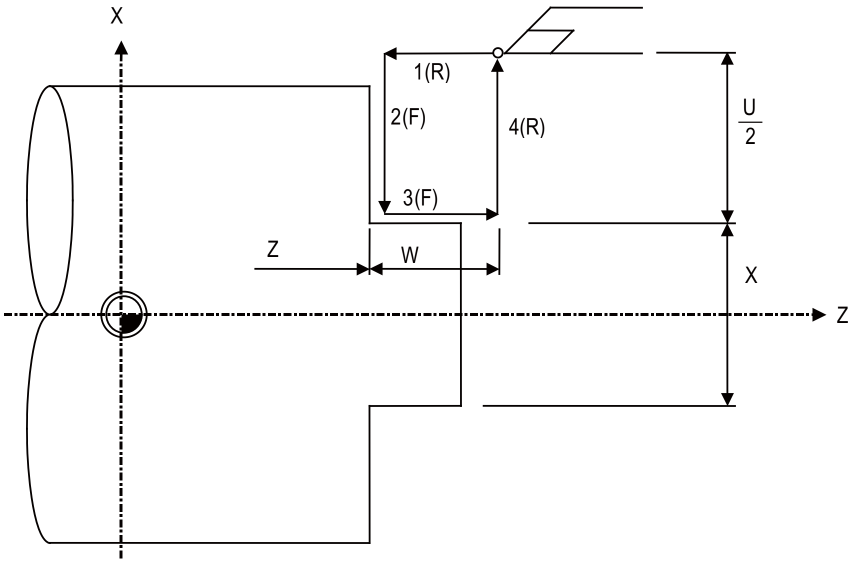
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°

(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_ ;

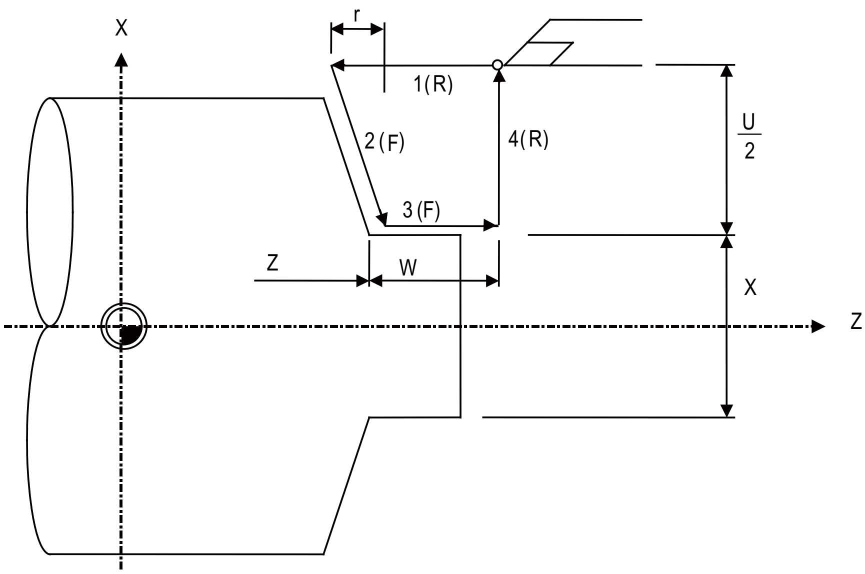


(R) Rapid traverse feed
(F) Cutting feed

(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) Rapid traverse feed
(F) Cutting feed
r: Taper part depth (radius designation, incremental position, sign is required)

12.1.3.5 Compound Type Fixed Cycle for Turning Machining

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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

G code	Function
G71	Longitudinal rough cutting cycle
G72	Face rough cutting cycle
G73	Molding material in rough cutting cycle
G70	Finish cycle
G74	Face cutting-off cycle
G75	Longitudinal cutting-off cycle
G76	Multiple repetitive thread cutting cycle

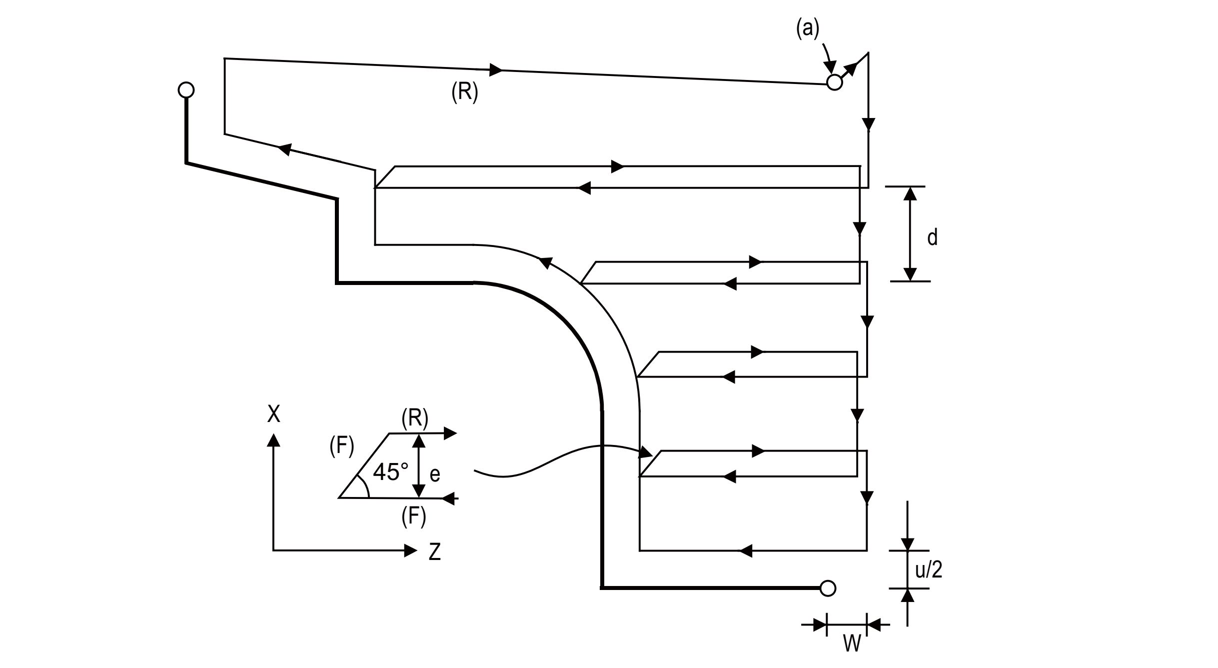
(1) Longitudinal rough cutting cycle (G71)

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	: Cutting amount (modal) .... Reversible parameter Increment : μ m or 1/10000inch ... Radius value command
Re	: Retract amount (modal) .... Reversible parameter Increment : μ m or 1/10000inch ... Radius value command
Aa	: Finish shape program No. (If omitted, the program being executed is designated.) If the A command is omitted, the program being executed are applied. If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.). A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
Pp	: Finish shape start sequence number (Head of program if omitted.)
Qq	: Finish shape end sequence number (To end of program if omitted.) If M99 precedes the Q command, up to M99.
Uu	: Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as 0.) Increment : μ m or 1/10000inch Diameter/radius value command follows changeover parameter.
Ww	: Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as 0.) Increment : μ m or 1/10000inch Radius value command
Ff	: Cutting speed (If omitted, cutting speed (modal) before G71 is applied.)
Ss, Tt	: Spindle command, tool command



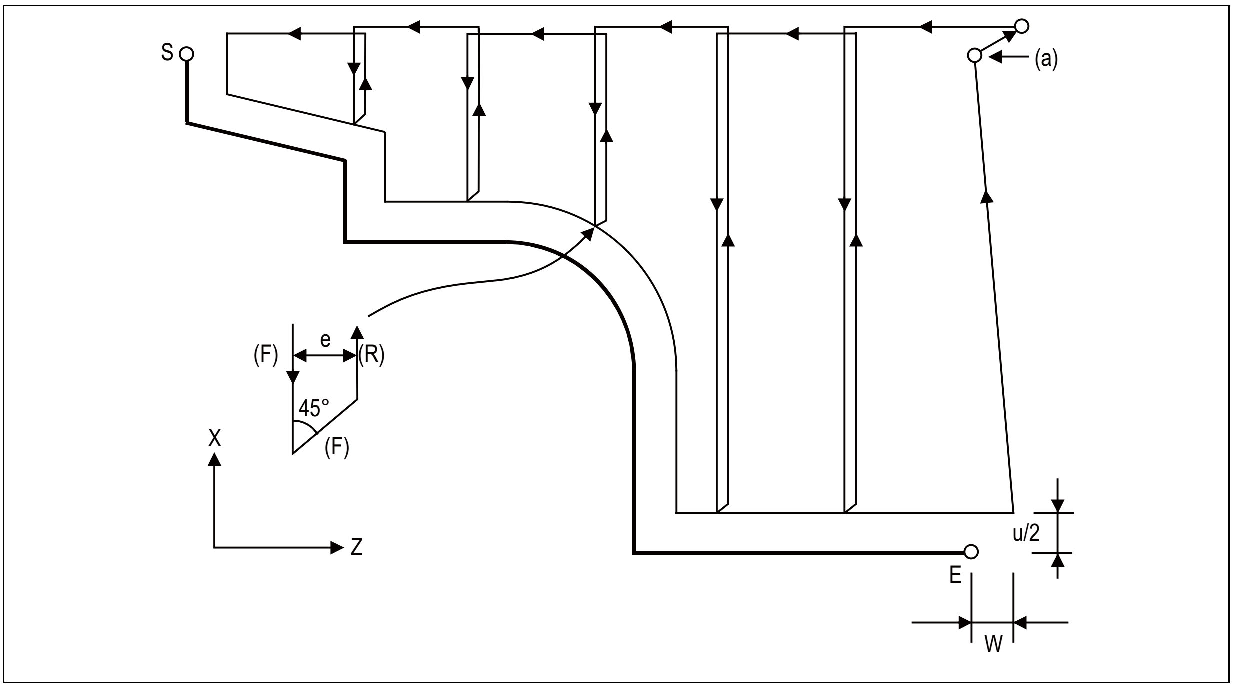
(a) (Cycle commanded point)
d Cutting amount
u/2 Finishing allowance
W Finishing allowance
(R) Rapid traverse feed
(F) Cutting feed

(2) Face rough cutting cycle (G72)

The finish shape program is called, and rough cutting 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 designated.) If the A command is omitted, the P, Q commands in the program being executed are applied. If the A command is omitted, the program being executed are applied. If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.). A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
Pp	: Finish shape start sequence number (Head of program if omitted.)
Qq	: Finish shape end sequence number (To end of program if omitted.) If M99 precedes the Q command, up to M99.
Uu	: Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as 0.)
Ww	: Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as 0.)
Ff	: Cutting feedrate (If omitted, cutting feedrate (modal) before G72 is applied.)
Ss, Tt	: Spindle command, tool command



(a) Cycle commanded point
d Cut depth
u/2 Finishing allowance
W Finishing allowance
(F) Cutting feed
(R) Rapid traverse feed

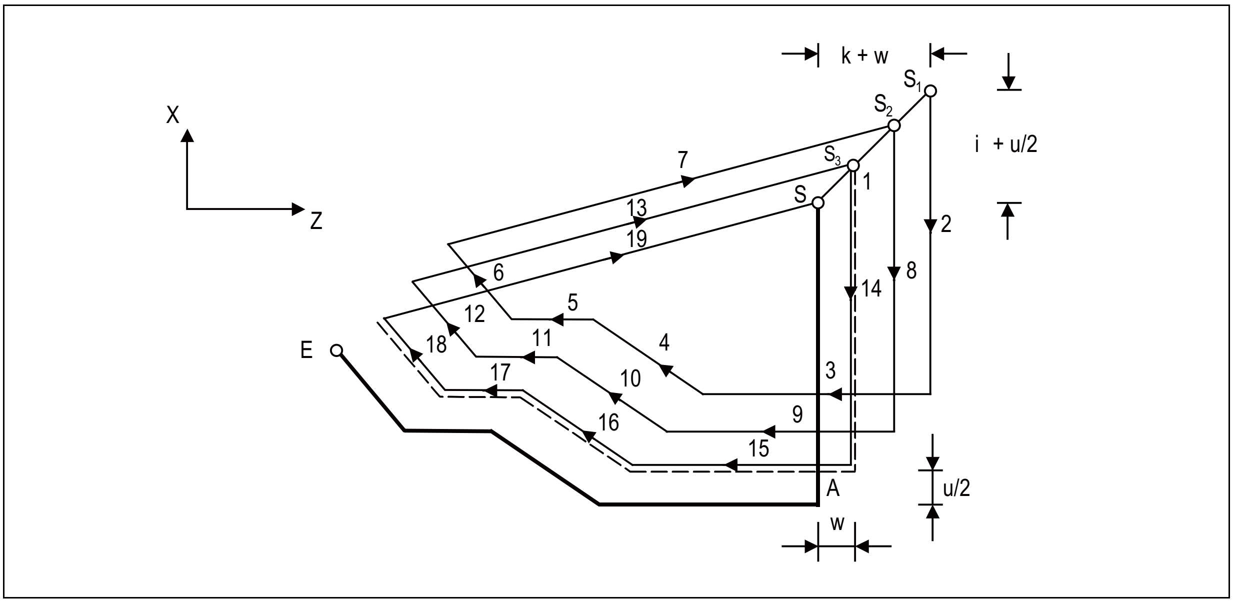
(3) Molding material in rough cutting cycle (G73)

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 Wk Rd	:Cutting allowance in the X axis direction i :Cutting allowance in the Z axis direction k :Split count d	- Cutting allowance when P, Q command is not given. - Modal data - Sign is ignored. - Cutting allowance is given with a radius designation.
Aa	:Finish shape program No.	(If it is omitted, the present program is assumed to be designated.) A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
Pp Qq	:Finish shape start sequence No. :Finish shape end sequence No.	(If it is omitted, the program top is assumed to be designated.) (If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Q command, up to M99.
Uu Ww	:Finishing allowance in the X axis direction u :Finishing allowance in the Z axis direction w	- Finishing allowance for the finish shape commanded by the address P or Q. - Sign is ignored. - Diameter or radius is designated according to the parameter. - The shift direction is determined by the shape
Ff Ss Tt	:Cutting feed rate (F function) :Spindle speed (S function) :Tool selection (T 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.



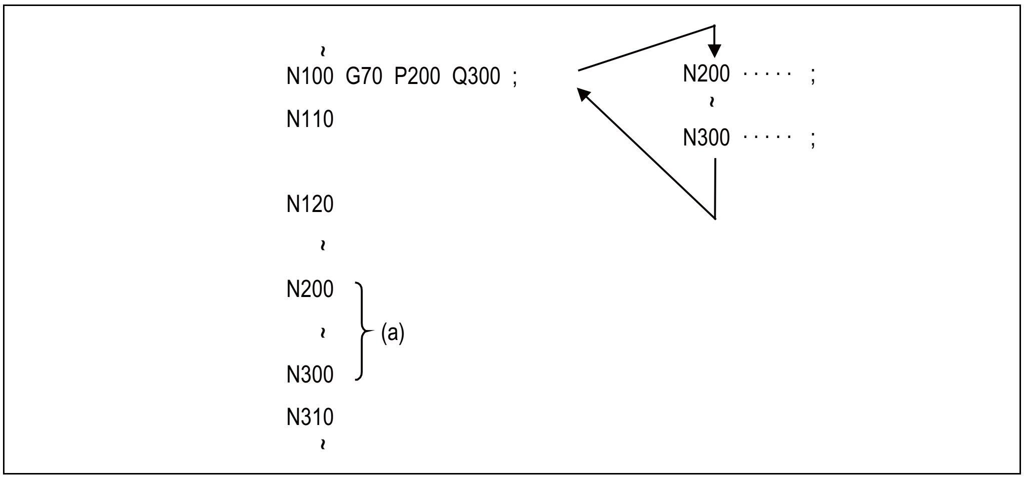
(4) Finish cycle (G70)

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

G70 Aa Pp Qq ;	
Aa	: Finish shape program No. (If it is omitted, the program being executed is assumed to be designated.) A file name can be designated instead of address A by enclosing the file name in brackets <>. (The file name can have up to 32 characters, including the extension.)
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.

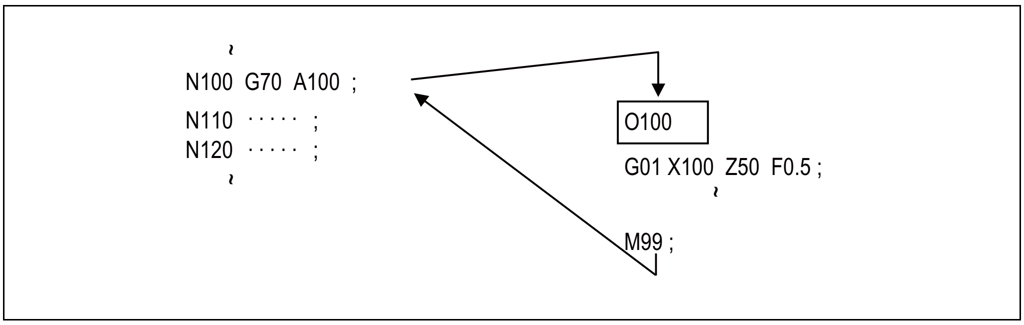
(a) The F, S, and T commands in the rough cutting cycle command G71 to G73 blocks are ignored, and the F, S, and T commands in the finish shape program become effective.
(b) The memory address of the finish shape program executed by G71 to G72 is not stored. Whenever G70 is executed, a program search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

(Example1) Sequence No. designation



(a) Finish shape program

(Example2) Program No. designation



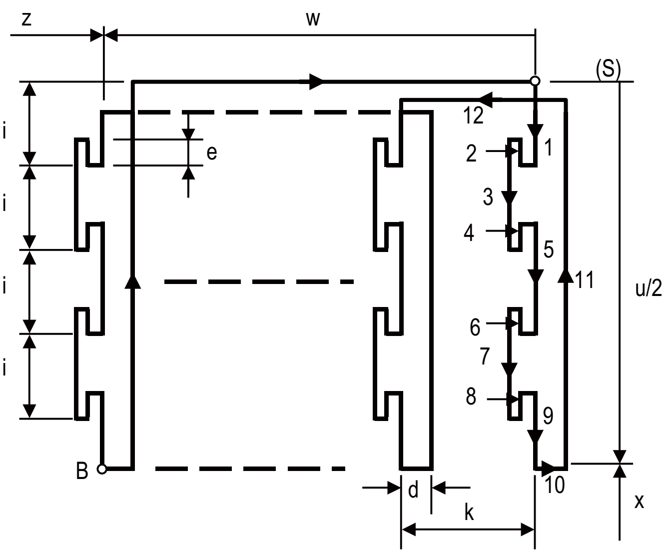
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

(5) 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 Xx1/(Uu1) Zz1/(Ww1) Pi Qk Rd Ff ;

Re	: Retract amount e (when X/U, Z/W command is not given) (Modal)	
Xx1/Uu1	: B point coordinate (absolute/incremental position)	
Zz1/Ww1	: B point coordinate (absolute/incremental position)	
Pi	: Tool shift amount (radius designation, incremental position, sign not required)	
Qk	: Cut depth k (radius designation, incremental position, 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 made 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.

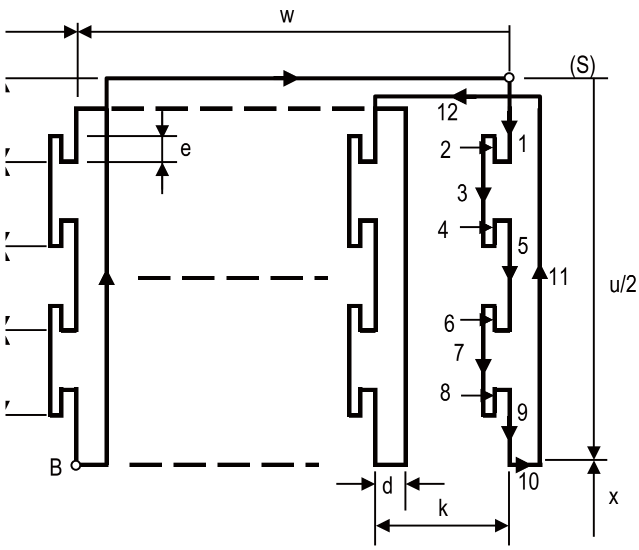
(S) Start point

(6) 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 Xx1/(Uu1) Zz1/(Ww1) Pi Qk Rd Ff ;

Re	: Retract amount e (when X/U, Z/W command is not given) (Modal)	
Xx1/Uu1	: B point coordinate (absolute/incremental position)	
Zz1/Ww1	: B point coordinate (absolute/incremental position)	
Pi	: Cut depth i (radius designation, incremental position, sign not required)	
Qk	: Tool shift amount k (radius designation, incremental position, 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 made 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.

(S) Start point

(7) Compound type thread cutting cycle (G76)

When the thread cutting start and end points are commanded, cut at any desired angle can be made by automatic 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

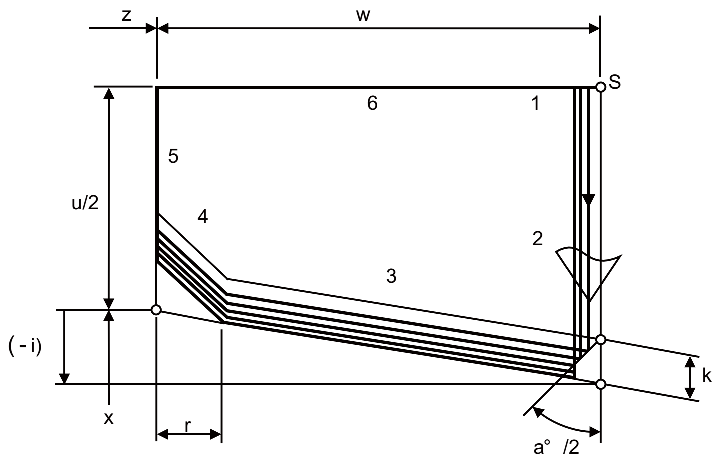
G76 Pmra Q △ dmin Rd ;
G76 Xx1/Uu1 Zz1/Ww1 Ri Pk Q △ d Fl ;

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)
△ dmin	: Least cut depth When the calculated cut depth becomes smaller than △ dmin, the cut depth is clamped at △ dmin.
Xx1/Uu1	: X axis end point coordinate of thread part. --- Absolute/Incremental position
Zz1/Ww1	: Z axis end point coordinate of thread part. --- Absolute/Incremental position
Ri	: Taper height constituent in thread part (radius value). When i = 0 is set, straight screw is made.
Pk	: Thread height. Designate the thread height in a positive radius value.
Q △ d	: Cut depth. Designate the first cut depth in a positive radius value.
Fl	: Thread lead

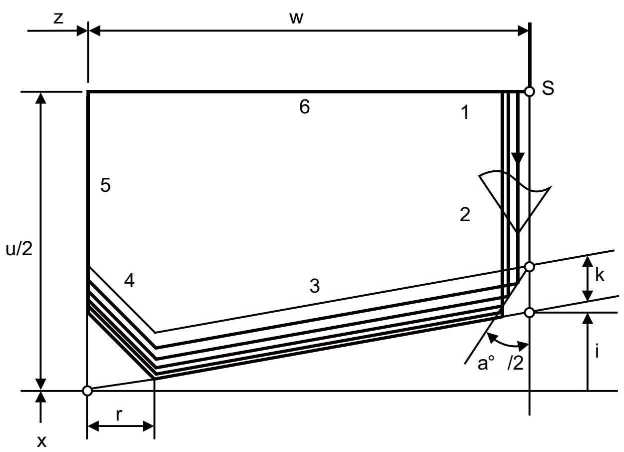
Configuration of one cycle

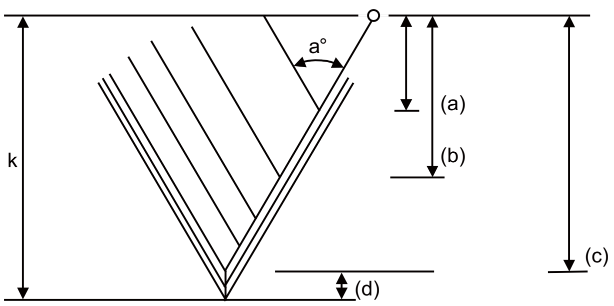
In one cycle, 1, 2, 5, and 6 move at rapid traverse feed and 3 and 4 move at cutting feed designated in F command.

When Ri is negative



When Ri is positive





(a) First time Δd

(b) Second time $\Delta d \times \sqrt{2}$

(c) nth time $\Delta d \times \sqrt{n}$

(d) Finishing allowanced (Cut "m" times at finishing)

12.1.3.6 Compound Type Fixed Cycle for Turning Machining (Type II)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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

The cutting method differs according to whether pocket machining is ON or OFF.

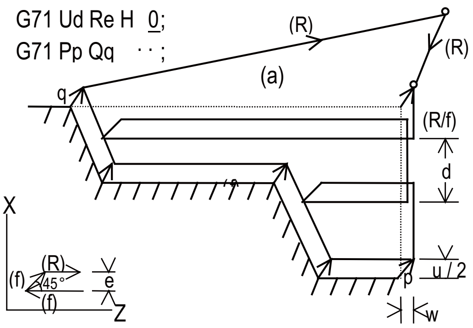
Pocket machining OFF .... Method to pull up the tool in a 45-degree direction from the workpiece
Pocket machining ON .... Method that traces the shape (After executing the last trace, the tool is pulled up in the X axis direction.)

Pocket machining is designated with the program (H address) or parameter.

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

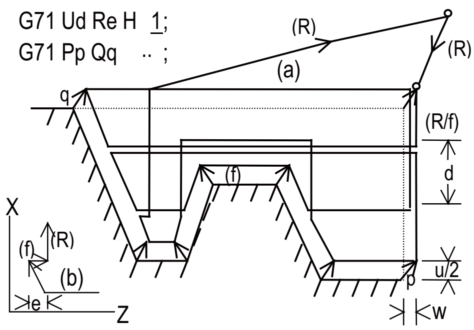
G71 Ud Re Hh ; <- (can be omitted when values set in parameters are used)
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

<H0:Used for finished shapes without pockets>



(a) Rough cutting start point

<H1:Mainly used for finished shapes with pockets>



(a) Rough cutting start point
(b) Hole bottom

Ud : Cut amount (modal) .... Reversible parameter
Increment : μm or 1/10000inch ..... Radius value command

Re : Retract amount (modal) .... Reversible parameter
Increment : μm or 1/10000inch .... Radius value command

Hh : Pocket machining (modal) .... Reversible parameter
0 : Select this only for finished shapes without hollow areas (pockets).
With the beginning of the pockets, the tool is pulled up in the 45-degree direction with each cycle until the finished shape is finally traced.
1 : This can be selected regardless of whether the finished shape has hollow (pocket) parts or not.
A method that traces the finished shape with each cycle is used for the beginning of the pockets.
Depending on the parameter setting, pocket machining ON/OFF is automatically determined by the number of axes in the finish shape start block.

Aa : Finish shape program No. (If omitted, the program being executed is designated.)
If the A command is omitted, the program being executed are applied.
If A is omitted, the program following the end of this cycle will be executed at the block after Qq (finish shape end sequence No.).
A file name can be designated instead of address A by enclosing the file name in brackets <>.
(The file name can have up to 32 characters, including the extension.)

Pp : Finish shape start sequence number (Head of program if omitted.)

Qq : Finish shape end sequence number (To end of program if omitted.)
If M99 precedes the Q command, up to M99.

Uu : Finishing allowance in X axis direction (If omitted, finishing allowance in X axis direction is handled as 0.)
Increment : μm or 1/10000inch .... Diameter/radius value command follows changeover parameter.

Ww : Finishing allowance in Z axis direction (If omitted, finishing allowance in Z axis direction is handled as 0.)
Increment : μm or 1/10000inch .... Radius value command

Ff : Cutting feed rate (If omitted, cutting feed rate (modal) before G73 is applied.)

Ss, Tt : Spindle command, tool command

12.1.3.7 Small-diameter Deep-hole Drilling Cycle

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

In deep hole drilling, cutting and retract are repeated and the workpiece is machined multiple times. In addition, when PLC signals are input during cutting, the cutting for the time concerned is skipped. In this way, this cycle reduces the load applied to the tool.

Command format

The small-diameter deep-hole drilling cycle mode is established by designating the M code command which was set in the parameter.

If the G83 command is designated in this mode, the small-diameter deep-hole drilling cycle is executed.

The mode is canceled by the following conditions.

- Designation of a fixed cycle cancel command (G80, G commands in Group 1)
- Resetting

The signal is not output in respect to the M command which changes the mode to the small-diameter deep-hole drilling mode.

G83 Xx1 Yy1 Zz1 Rr1 Qq1 Ff1 Ii1 Pp1 ;	
Xx1, Yy1	: Hole drilling position
Zz1	: Hole bottom position
Rr1	: R point position
Qq1	: Depth of cut in each pass (designated with incremental position)
Ff1	: Cutting feed rate
Ii1	: Feedrate from R point to the cutting start position, the speed for returning from hole bottom
Pp1	: Dwell time at hole bottom position

12.1.4 Mirror Image

12.1.4.1 Mirror Image by Parameter Setting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

12.1.4.2 Mirror Image by External Input

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

12.1.4.3 Mirror Image by G Code

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

Using a program for the left or right side of a shape, this function can machine the other side of the shape 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.

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

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

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

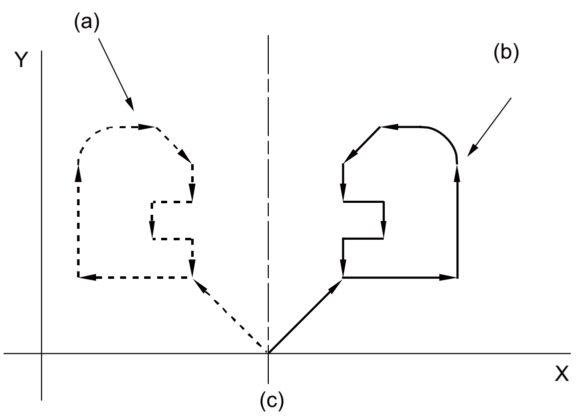
With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

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

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

The x1, y1 and z1 indicate the axes for which the mirror image function is to be canceled and the coordinate position is ignored.

In the case of G51.1 Xx1 ;



(a) Original shape (program)
(b) Shape achieved when machining program for the left side has been executed after the mirror command
(c) Mirroring axis

12.1.4.4 Mirror Image for Facing Tool Posts

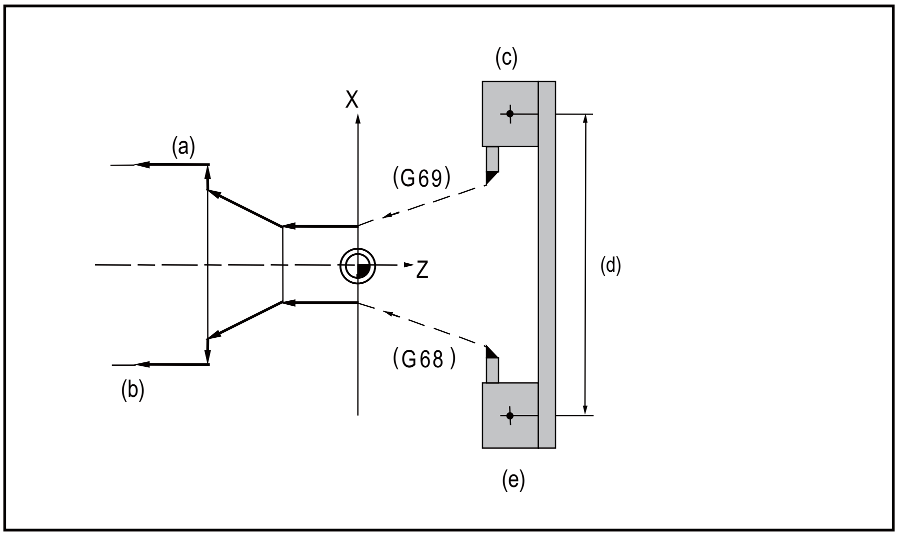
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

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.

Gcode	Function
G68 G69	Facing tool post mirror image ON 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.



(a) Programmed path
(b) Facing side path (mirror image ON)
(c) Base post
(d) Parameter for distance between posts (radial value, X axis only)
(e) Facing post

12.1.4.5 T Code Mirror Image for Facing Tool Posts

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

If, when tools that correspond to tool Nos. 1 to 64 are selected (T commands), these are tool Nos. for which the facing tool post mirror image function has already been designated with a parameter, the status equivalent to G68 (facing tool post mirror image ON) is established. When the commands apply to tool Nos. for which the facing tool post mirror image function is not designated, the status equivalent to G69 (facing tool post mirror image OFF) is established.

12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

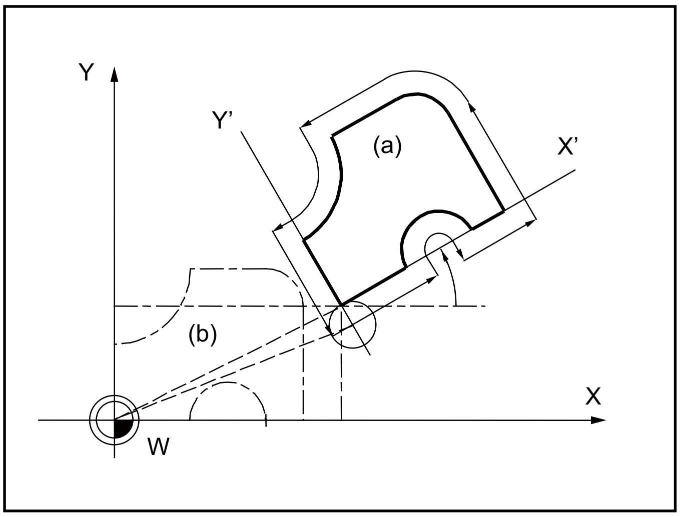
The program format for the coordinate rotation command is given below.

(1) M system

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

G68	: Call command
G69	: Cancel command
Xx1,Yy1	: Rotation center coordinates
Rr1	: Angle of rotation

(Example)

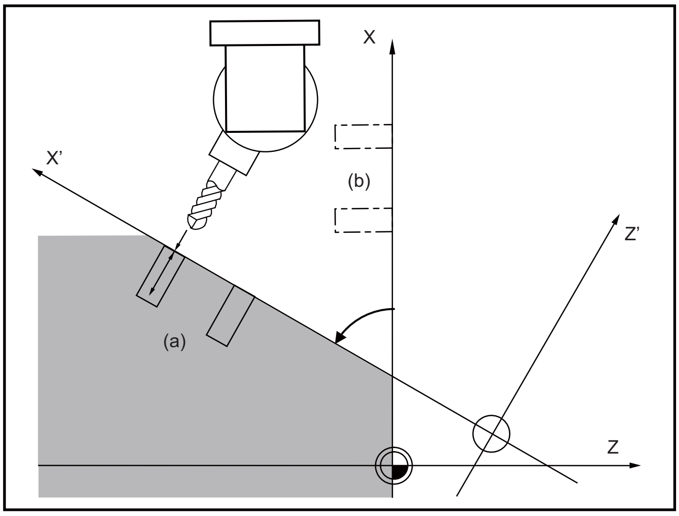


(a) Actual machining shape
(b) (Programmed coordinate)

(2) L system

G68.1 Xx1 Yy1 Rr1 ; Coordinate rotation ON	
G69.1 ; Coordinate rotation cancel	
G68.1	:Call instruction
Xx1,Zz1	:Rotation center coordinates
Rr1	:Angle of rotation

(Example)

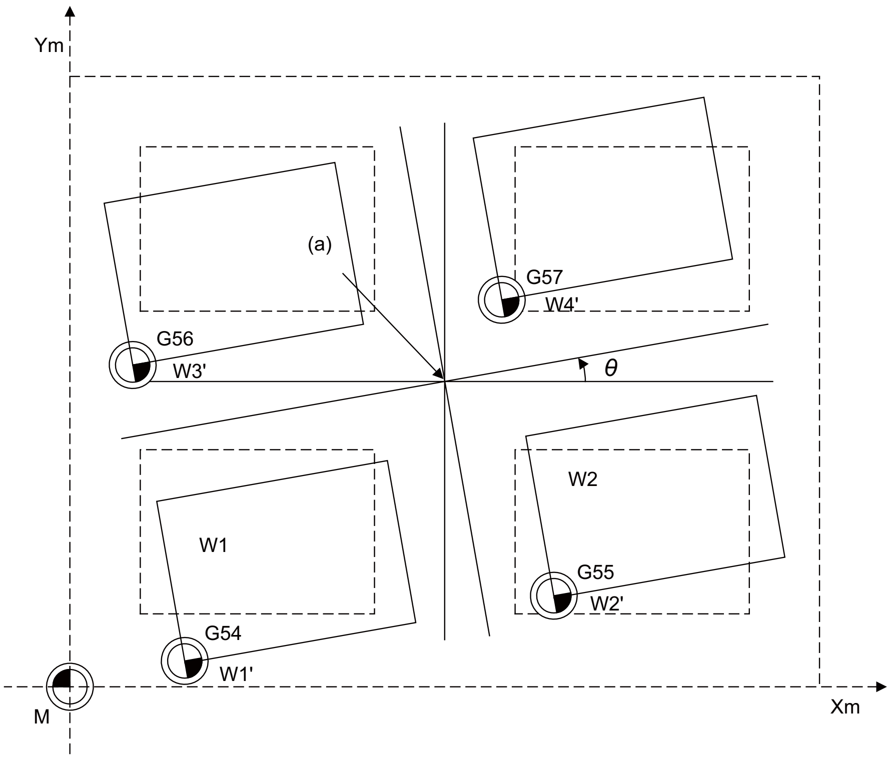


(a) Actual machining shape
(b) (Programmed coordinate)

12.1.5.2 Coordinate Rotation by Parameter

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

If a deviation occurs between the workpiece alignment line and machine coordinate system's coordinate axis when the workpiece is mounted, the machine can be controlled to rotate the machining program coordinates according to the workpiece alignment line deviation. The coordinate rotation amount is set with the parameters. The parameter can be set either on the parameter screen or with the G10 command.
Note that when the G10 command is used, the separate additional specification "programmable parameter input" is required.



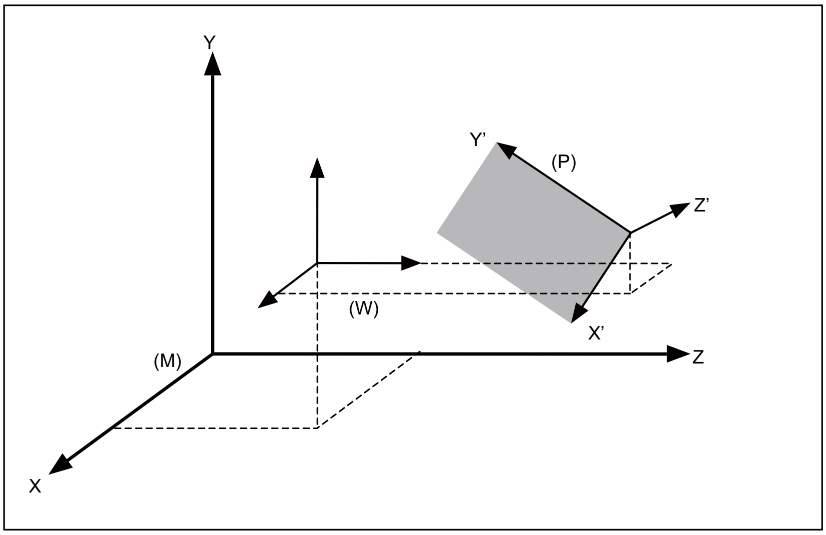
(a) Rotation center
θ **Rotation angle**

(1) As for the rotation center coordinate position, designate the position on the machine coordinate system.
(2) All workpiece coordinate systems from G54 to G59 rotate with the rotation command.
While the machine coordinate system does not rotate, it can be understood that there is a hypothetical machine coordinate system in the coordinate system after rotation.
(3) The coordinate position counter does not rotate. The position in the original workpiece coordinate system (non-rotated) is displayed.
(4) If the setting is made on the parameter screen, the setting is validated with the cycle start after the parameter is set. If set with the G10 command, settings are immediately validated.

12.1.5.3 3-dimensional Coordinate Conversion

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

With the 3-dimensional coordinate conversion function, a new coordinate system can be defined by rotating and moving in parallel the zero point in respect to the X, Y and Z axes of the currently set workpiece coordinate system. By using this function, an arbitrary spatial plane can be defined, and machining on that plane can be carried out with normal program commands.
An option is required to validate this function. If the 3-dimensional coordinate conversion is commanded when the option is not added, a program will occur.



(P) G68 program coordinate system
(W) Workpiece coordinate system
(M) Machine coordinate system

When the G68 command is issued, the zero point is shifted by the command value (x, y, z) in respect to the current local coordinate system. A new G68 program coordinate system rotated by the designated rotation angle r in respect to the commanded rotation center direction (i, j, k) is created.
The local coordinate system is the same as the workpiece coordinate system when the local coordinate system offset is not ON.

The program format is as follows.

G68 Xx Yy Zz Ii Jj Kk Rr ;

G68	: 3-dimensional coordinate conversion mode command
Xx,Yy,Zz	: Rotation center coordinates Designate with the absolute position of the local coordinate system.
Ii,Jj,Kk	: Rotation center axis direction 1:Designated、0:Not designated Note that "1" is designated for only one of the three axes. "0" is designated for the other two axes.
Rr	: Rotation angle The counterclockwise direction looking at the rotation center from the rotation center axis direction is positive (+). The setting range is -360 to 360°, and the increment follows the least command increment.

G69 ;

G69	: 3-dimensional coordinate conversion mode cancel command

12.1.6 Dimension Input

12.1.6.1 Corner Chamfering/Corner R

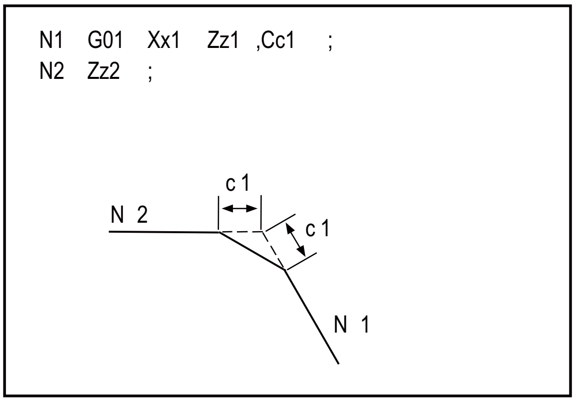
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function executes corner processing by automatically inserting a straight line or an arc by 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.

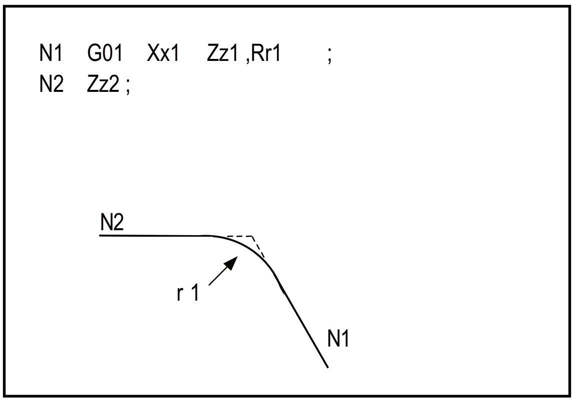
(1) Corner chamfering / Corner R I

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 ;



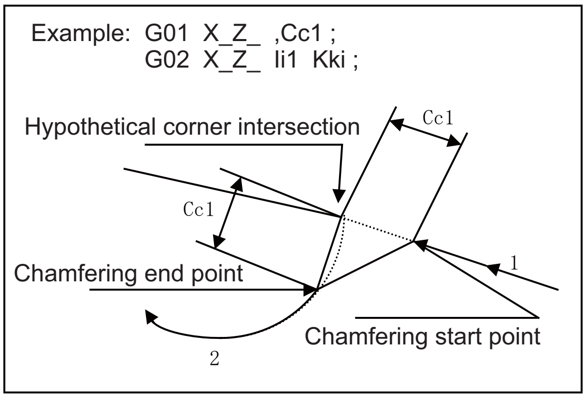
(Note 1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

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

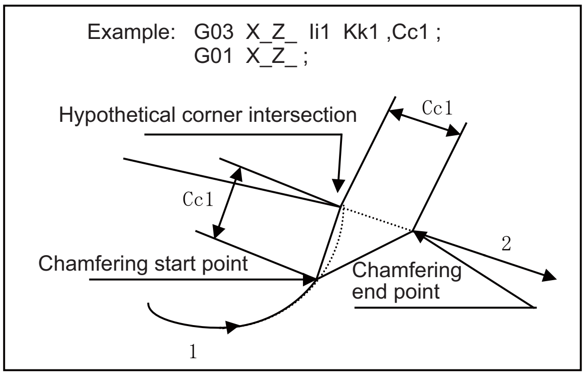
When ",C" or ",R" is commanded in a program between linear-circular, corner chamfering or corner R can be inserted between blocks.

When the parameters are set, "I_", "K_", "C_" can be used to command chamfering instead of ",C", and "R_" can be used to command rounding instead of ",R_".

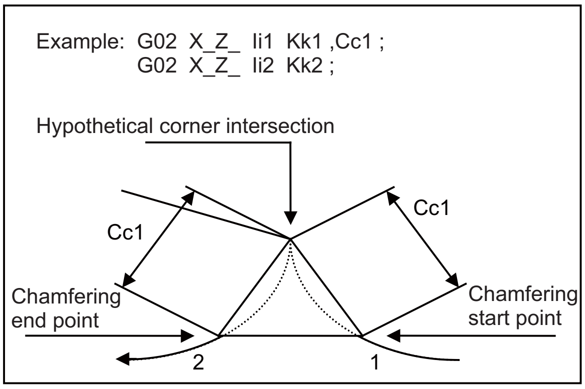
(a) Corner chamfering II (Linear - circular)



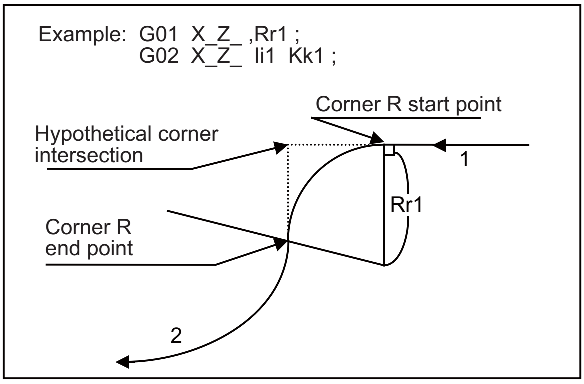
(b) Corner chamfering II (Circular - linear)



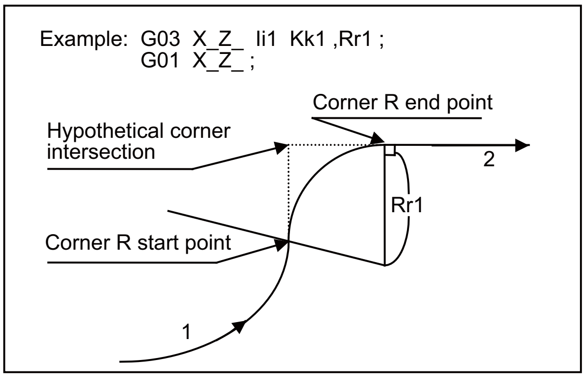
(c) Corner chamfering II (Circular - circular)



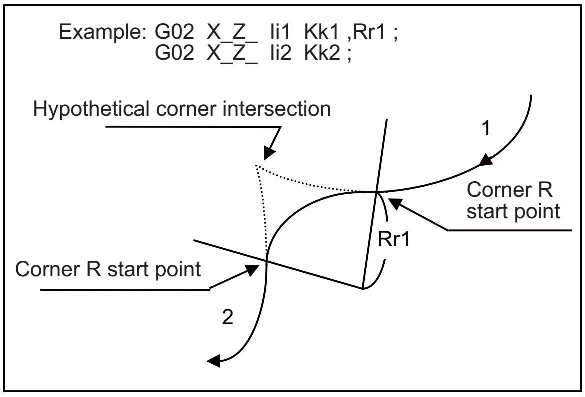
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular - linear)



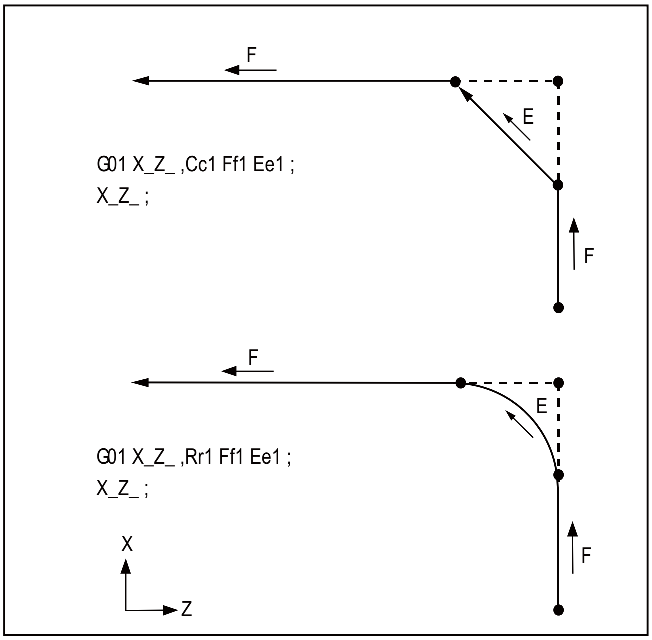
(f) Corner R II (Circular - circular)



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

An E address 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 address is a modal and remains effective for feeding in next corner chamfering or corner R. An E address 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 address 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 address modal at the time of resetting. It is cleared when the power is turned OFF (as it is with an F command).

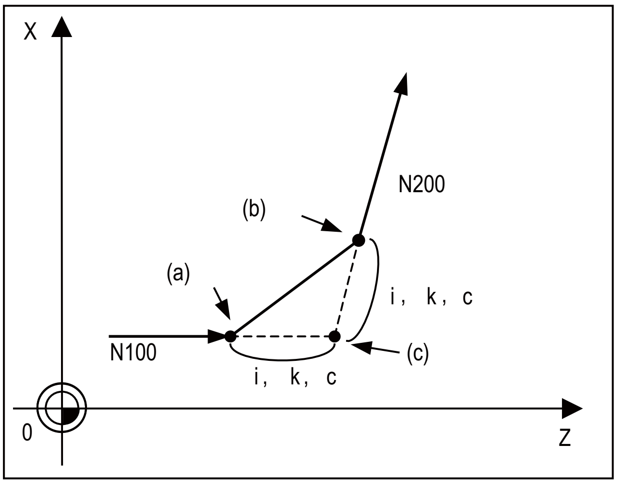
(4) Corner chamfering / corner R (I, K designation)

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

(a) Corner chamfering (I, K designation)

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

N100 Xx1/Uu1 Zz1/Ww1 Ii1/Kk1/Cc1 ;	
N200 Xx1/Uu1 Zz1/Ww1 ;	
Xx1/Uu1	: X-axis end point coordinate
Zz1/Ww1	: Z-axis end point coordinate
Ii1/Kk1/Cc1	: The length from the hypothetical corner intersection to the chamfering start point or chamfering end point is designated using the I, K or C address.



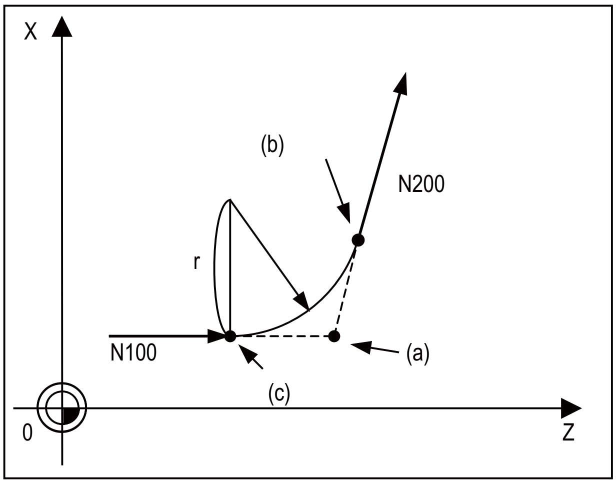
(a) Chamfering start point
(b) Chamfering end point
(c) Hypothetical corner intersection

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

(b) Corner R (I, K designation)
Corners are rounded using the "R_" address with no comma.
Corners can be rounded to any angle.
Signs, if they are provided for the corner R commands, are ignored.
Command format

N100 Xx1/Uu1 Zz1/Ww1 Rr1 ;
N200 Xx1/Uu1 Zz1/Ww1 ;

Xx1/Uu1	: X-axis end point coordinate
Zz1/Ww1	: Z-axis end point coordinate
Rr1	: Radius of corner R arc



(a) Hypothetical corner intersection
(b) Corner R end point
(c) Corner R start point

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

12.1.6.2 Linear Angle Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

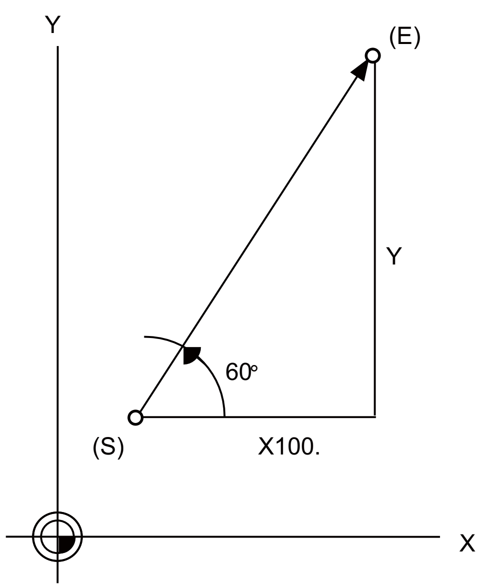
The end point coordinates are automatically calculated by assigning one element (one component of the selected plane) of the end point coordinates and the linear angle.

G17 Xx1 Aa1 ; or G17 Yy1 Aa1 ;

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

Example

G17 G91 X100. A60. ;



(E) End point (Automatically calculated with operation)
(S) Start point

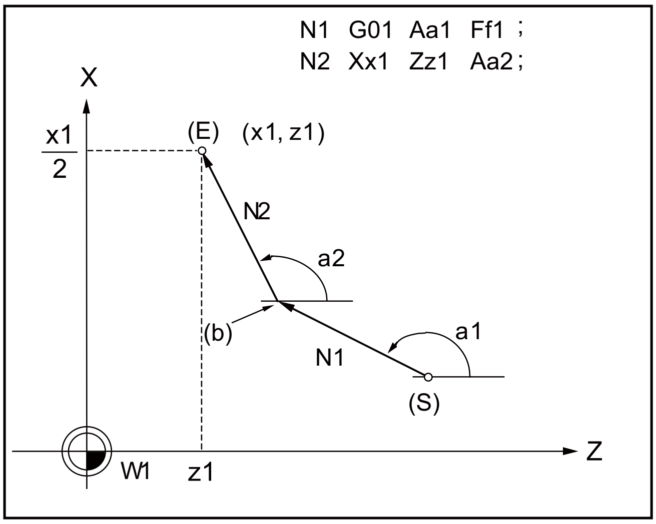
(Note 1) If the axis "A" or 2nd miscellaneous function "A" is used, address "A" is treated as the axis "A" command or the 2nd miscellaneous function, respectively.

12.1.6.3 Geometric Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	○	○	○	○	○	○

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

Example



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

(a) Automatic intersection point calculation
(E) End point
(S) Start point

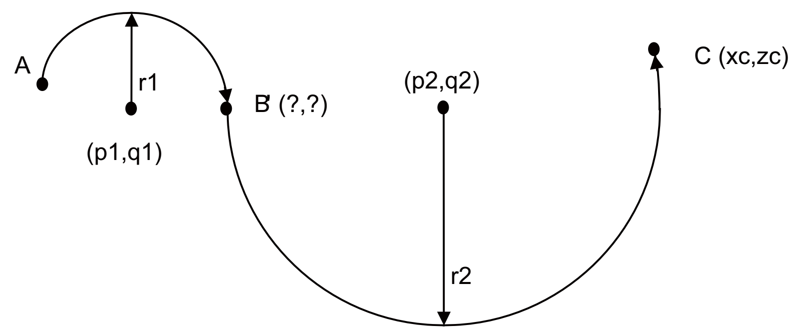
(Note1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(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 position or radius of the first circular arc and the end point (absolute position) and center position or radius of the second circular arc.

Example

G18 G02 Ii1 Kk1 Ff1 ;
G03 Xxc Zzc 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 : Incremental position from arc end point
P and Q : Arc center position (absolute position)

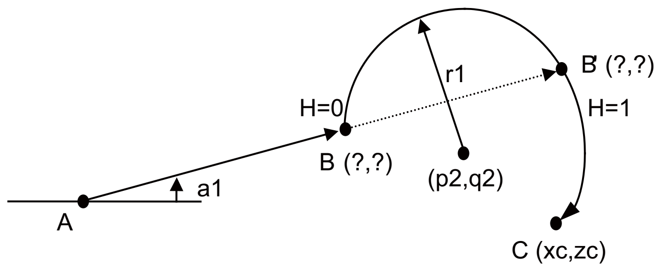
I and K are the arc center position (incremental position); distances from the start point in the first block or distances from the end point in the second block. P and Q (X, Z arc center position (absolute position)) can be commanded instead of I and K commands.

(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 (,Hh2) Ff2 ;



I and K : Incremental position from arc end point
P and Q : Arc center position (absolute position)
H = 0 : Intersection with shorter line (B point)
H = 1 : Intersection with longer line (B' point)

The p2 and q2 can be commanded instead of Ii2 and Kk2.
The linear - arc contact is automatically calculated by designating R instead of I and K (P, Q).

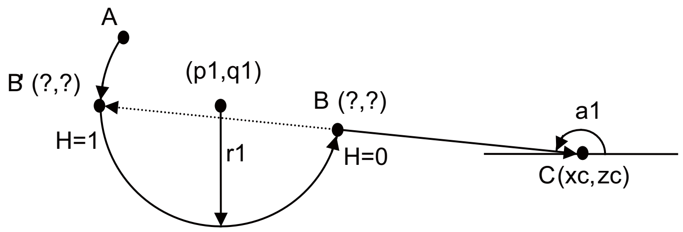
(3) Automatic calculation of arc-linear intersection

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 Ff1 ;



I and K : Incremental position from arc end point
P and Q : Arc center position (absolute position)
H = 0 : Intersection with shorter line (B point)
H = 1 : Intersection with longer line (B' point)

The p1 and q1 can be commanded instead of Ii1 and Kk1.
The arc - linear contact is automatically calculated by designating R instead of I and K (P, Q).

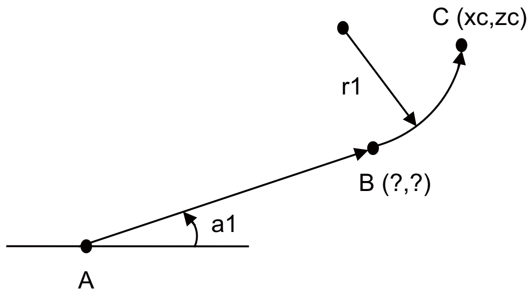
(4) Automatic calculation of linear-arc contact

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 Ff1 ;



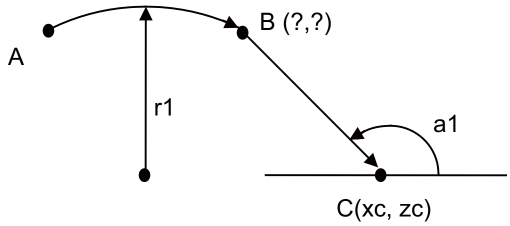
The linear - arc intersection is automatically calculated by designating R instead of P and Q (I, K).

(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   Ff1    ;
```



The arc - linear intersection is automatically calculated by designating R instead of P and Q (I, K).

12.1.6.4 Polar Coordinate Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

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

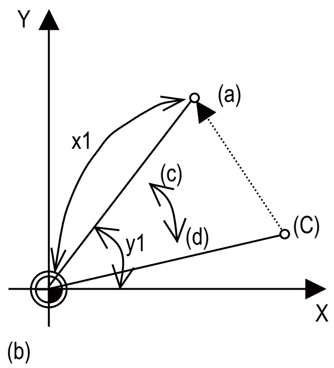
Command format

G16 ;	**Polar coordinate command mode ON**

G15 ;	**Polar coordinate command mode OFF**

Example of program

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



(a) Commanded position (end point)
(b) For G90/G17(X-Y plane)
(c) Plus
(d) Minus

G15 ;	Polar coordinate command mode OFF

12.1.7 Axis Control

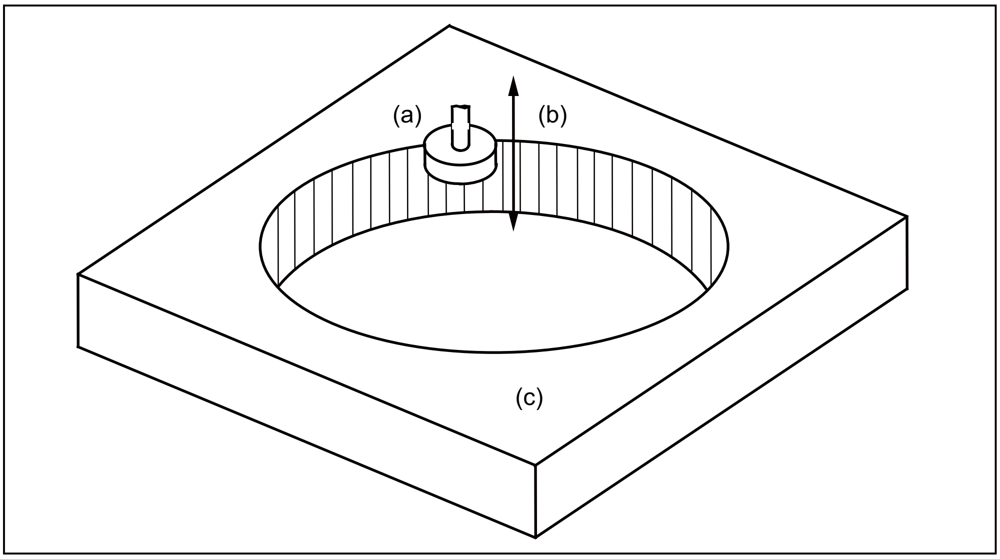
12.1.7.1 Chopping

12.1.7.1.1 Chopping

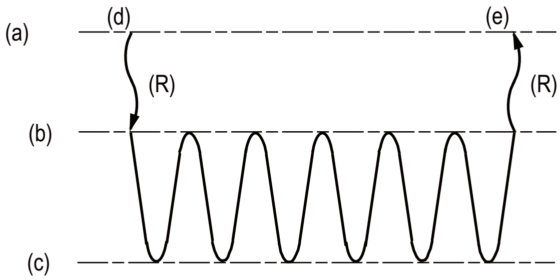
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

(1) Chopping action



(a) Grindstone
(b) Chopping action
(c) Workpiece



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

(a) Base position
(b) Upper dead point
(c) Lower dead point
(d) Start
(e) Stop
(R) Rapid traverse

(Note 1) The upper dead point position, lower dead point position and number of cycles are set and the start and stop commands are designated by input signals from the user PLC.
(Note 2) The setting for the number of cycles differs according to the motor, inertia and other factors.

The chopping operation is performed as follows.

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

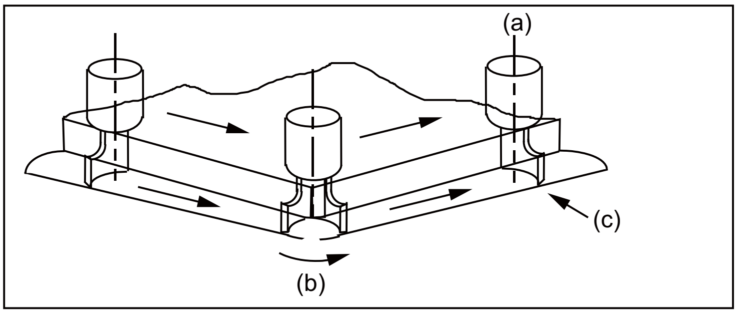
Chopping override

Override in 1% increments from 0% to 100% can be applied to the chopping operation.
(Note) Bear in mind that the override increment differs according to the machine specifications.

12.1.7.2 Normal Line Control

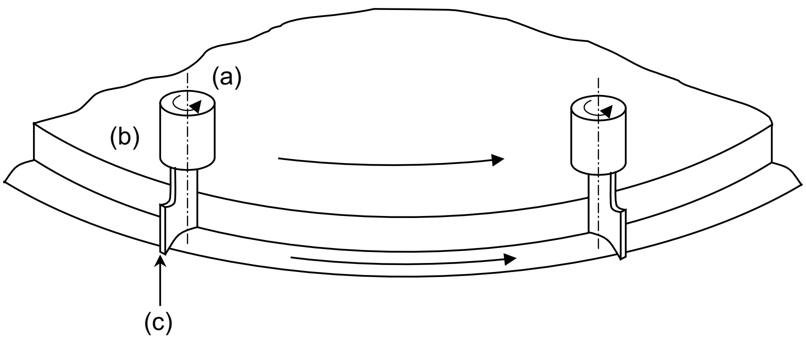
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This function controls the swiveling of the C axis (rotary axis) so that the tool is always pointing in the normal line direction for the X and Y axes movement commands during program operation. It can be used for machining hale, for sewing operations involving sewing machines, carpets, etc. It is valid for only one C axis (rotary axis).
At the block seams, the C axis turning is controlled so that the tool faces the normal line direction at the next block's start point.



(a) C-axis center (rotary axis)
(b) C axis swiveling
(c) Tool tip position

During arc interpolation, the C axis turning is controlled in synchronization with the operation of arc interpolation.



(a) C axis center (rotation axis)
(b) Tool
(c) Tool end position

Gcode	Function
G40.1 G41.1 G42.1	Normal line direction control cancel Normal line direction control left ON Normal line direction control right ON

12.1.7.3 Circular Cutting

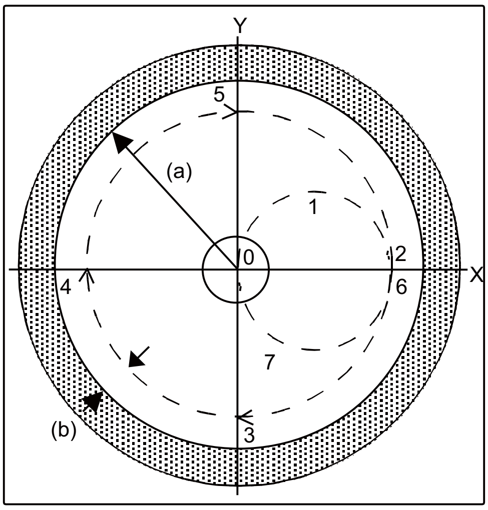
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed serves as the center of the circle.

The program format is given below.

G12/(G13) Ii Dd Ff ;	
G12	: Circular cutting command CW (clockwise)
G13	: Circular cutting command CCW (counter clockwise)
Ii	: Radius of complete circle
Dd	: Compensation No.
Ff	: Feed rate

When the G12 command is used (path of tool center)
0 → 1 → 2 → 3 → 4 → 5 → 6 → 7 → 0
When the G13 command is used (path of tool center)
0 → 7 → 6 → 5 → 4 → 3 → 2 → 1 → 0



(a) Radius of circle
(b) Compensation amount

(Note1) Circular cutting is carried out on the plane which has been currently selected (G17, G18 or G19).
(Note2) The (+) and (-) signs for the compensation amount denote reduction and expansion respectively.

12.1.8 Multi-part System Control

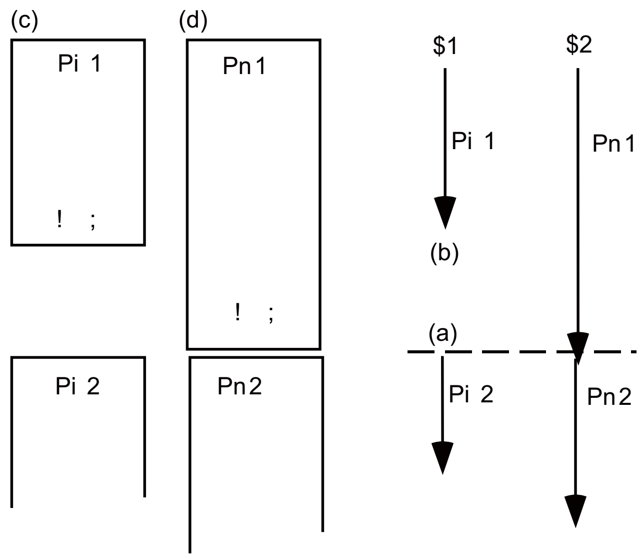
12.1.8.1 Timing Synchronization between Part Systems

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Timing synchronization between part systems (block start synchronization) is possible by programming.

! ; Part system 1 and part system 2 are waited for a synchronous operation.	When the part system number is to be omitted

This timing synchronization command makes it possible to control the simultaneous operation of two part systems and the independent operation of part systems.



(a) Simultaneously start
(b) Wait
(c) Part system 1 program
(d) Part system 2 program
$1 Part system 1
$2 Part system 2

(Note 1) When a part system to be waited for a simultaneous operation is not operating, the currently operating part system will move on to the next block without executing timing synchronization. This function is useful when conducting a program check on each part system.

(Note 2) The timing synchronization command is normally assigned as a separate command. However, if a movement command and M, S or T command have been assigned in the same block, a parameter is set to decide whether timing synchronization is executed upon executing of the movement command and M, S or T command, or the movement command and M, S or T command are executed upon completion of timing synchronization.

12.1.8.2 Start Point Designation Timing Synchronization

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

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

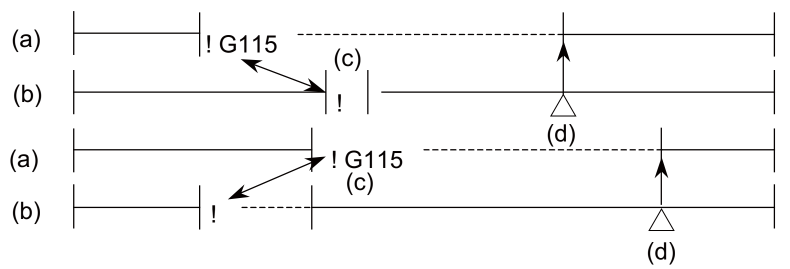
Command format

!Ll G115 Xx1 Zz1 Cc1 ;

!Ll	: Timing synchronization command
G115	: G command
Xx1, Zz1, Cc1	: Own start point (designate other part system's coordinate position)

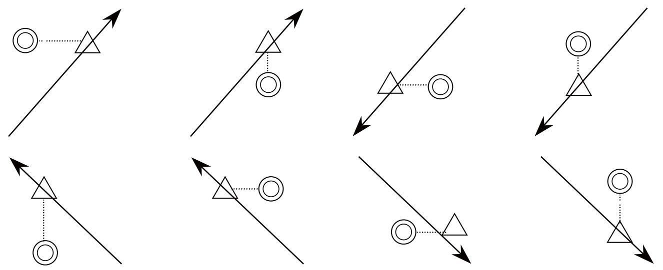
(a) The other part system starts first when timing synchronization is executed.

(b) The own part system waits for the other part system to move and reach the designated start point, and then starts.



(a) Own part system
(b) Other part system
(c) Timing synchronization
(d) Designated start point

(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 each start point axis coordinate position.



← : Movement
◎ : Command point
△ : Actual start point

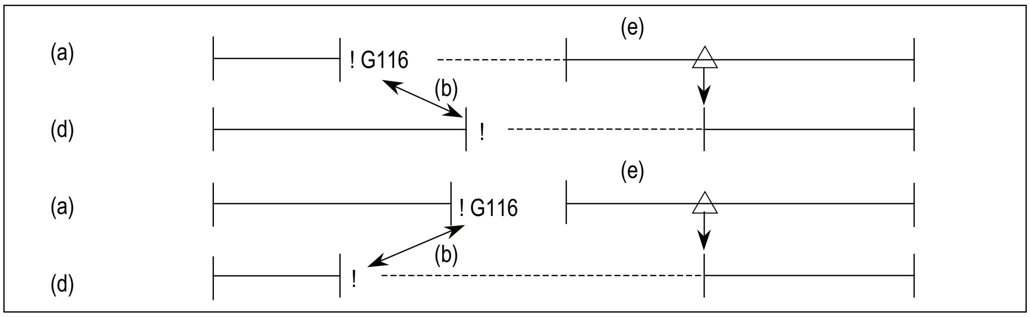
(2) Start point designation timing synchronization Type 2 (G116)

Command format

!L*l* G116 Xx1 Zz1 Cc1 ;	
!L*l*	: Timing synchronization command
G116	: G command
Xx1, Zz1, Cc1	: Other start point (designate own part system's coordinate position)

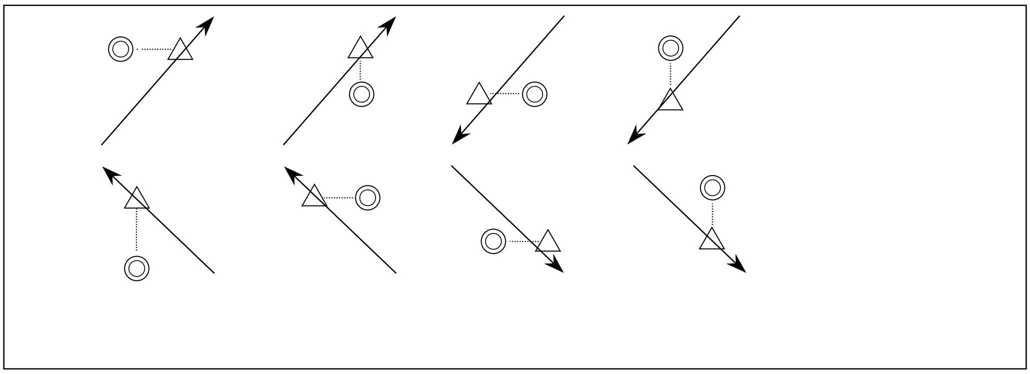
(a) The own part system starts first when timing synchronization is executed.

(b) The other part system waits for the own part system to move and reach the designated start point, and then starts.



(a) Own part system
(b) Timing synchronization
(d) Other part system
(e) Designated start point

(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 each start point coordinate position.



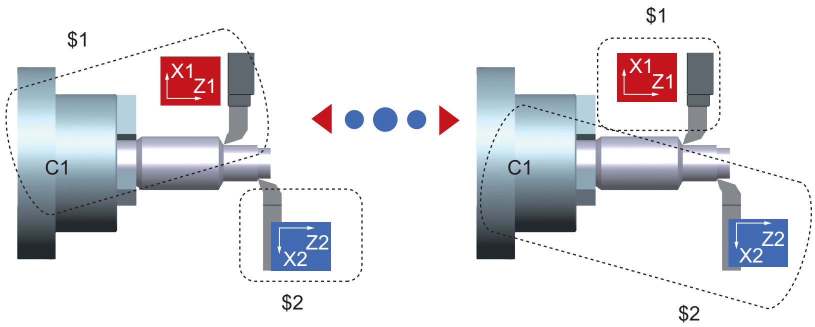
← : Movement
◎ : Command point
△ : Actual start point

12.1.8.3 Mixed Control (Cross Axis Control)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

This function enables any axis to be replaced by another axis between part systems.
This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on part system 1 can be used for machining on part system 2.
This manual contains descriptions where part system Nos. have been added after the axis names in order to identify which part system the axes belong to (such as X1). In terms of designating the program commands, however, a single letter is used to designate the axis address just as before.
The mixed control (cross axis control) has two types depending on the command method:
mixed control (cross axis control) I (G command) and mixed control (cross axis control) II (command with PLC signal).

Switching C1 axis control from the 1st part system to the 2nd part system



12.1.8.4 Control Axis Superimposition

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

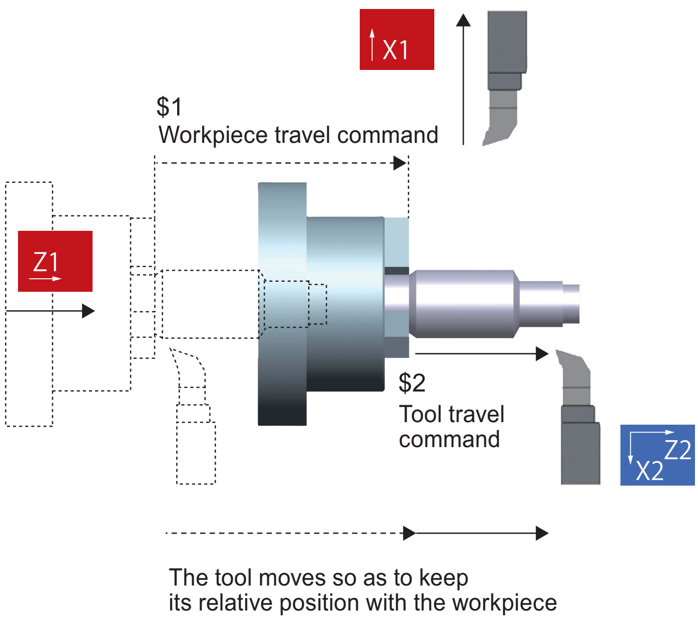
This function enables to superimpose on and control an axis in a part system with an axis in another part system.
This is effective when machining by the 1st part system (X1, Z1) and the 2nd part system (X2, Z2) are executed simultaneously for the machine configuration where a workpiece moves by a movement command to the Z axis direction. This function eliminate the need of fixing the workpiece position for machining in the superimposed part system (the 2nd part system) and allows the simultaneous machining of the reference axis part system (the 1st part system) and the superimposed axis part system (the 2nd part system).

Reference axis and superimposition axis are defined as follows.

Reference axis	The basic in the control axis superimposition function (moves only by its own axis command)
Superimposed axis	The axis which moves including the reference axis movement in the control axis superimposition function (moves by reference axis or/and own axis command).

While using the control axis superimposition function, the workpiece zero point of the superimposed axis moves according to the movement amount of the reference axis. Because the superimposed axis tries to maintain the position in the workpiece coordinate system, it moves as much as the movement amount of the reference axis.
As long as the reference axis and the superimposed axis do not overlap, the number of control axis superimposition set has no limitation.

(Example)
Reference axis: Z1
Superimposed axis: Z2
The zero point in the figure indicates the 2nd part system workpiece coordinate.



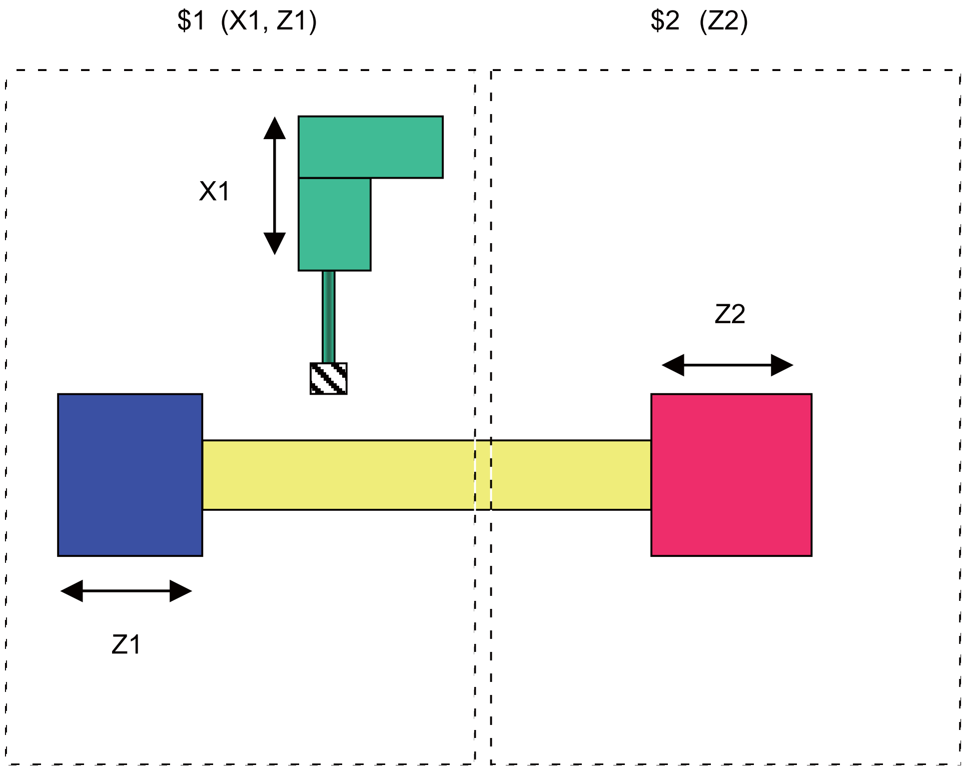
12.1.8.5 Control Axis Synchronization across Part Systems

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

Synchronization control enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.
The control axis synchronization across part systems has two types depending on the command method: control axis synchronization across part systems I (G command) and control axis synchronization across part systems II (command with PLC signal). It can be selected with the parameters.
The direction in which the axis is to move synchronously can also be reversed using a parameter.

Base axis････Axis to which movement command to synchronize axes is assigned.
(X1 in the following figure)

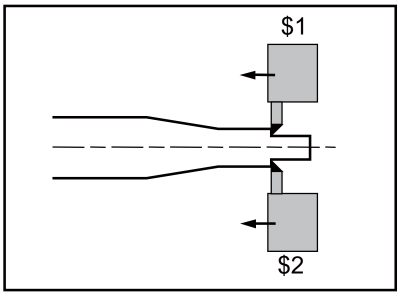
Synchronization axis････Axis whose movement is synchronized with base axis.
(Z2 in the following figure)



12.1.8.6 Balance Cut

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L 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.

(a)

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

(b)

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

(a) Part system 1 program
(b) Part system 2 program

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

12.1.8.7 Common Memory for Part Systems

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

For a machine with multiple part systems, the common variables and tool compensation memory which exist for each part system can be made common for all part systems by setting the parameters.

(1) Common variables

In the multi-part system, normally the common variables #100 to #199 are used for each part system, and variables #500 to #999 are common for the part systems. When this function is valid, the common variables #500 to #999 also become variables for each part system.
To use part or all of the common variables #100 to #199 and #500 to #999 commonly for all part systems, set the number of parameters to be used commonly for the part systems in variables #100 to #199 and #500 to #999.
As for the variables designated to be common to part systems, the data for part system 1 is used.

(2) Tool compensation memory

When this specification is valid, the tool compensation memory used for operation in the 2nd or following part system changes to be used with the part system 1 data. Thus, the 2nd part system and following data is not used when the common data is used for the part systems.
This function is only for the lathe system and supported up to 2 part systems.

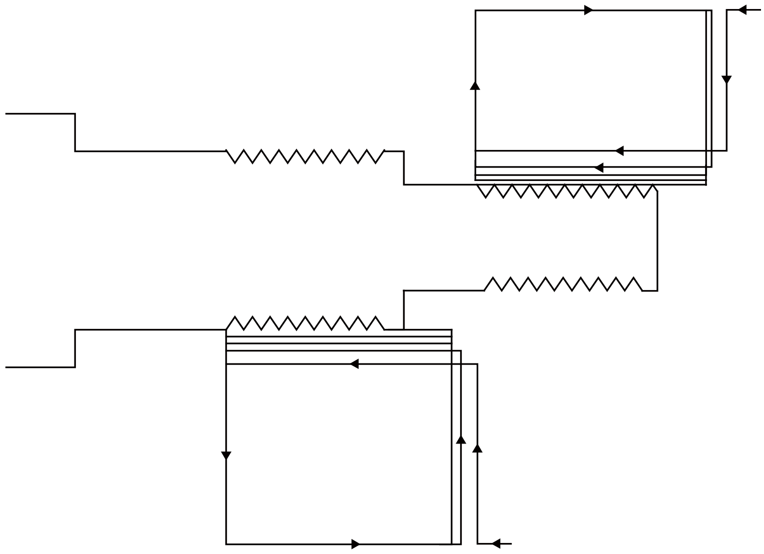
(Note) The setting (such as number of axes and axis names) for the tool compensation amount of part system 1 and part system 2 must be the same.
If the settings differ, the setting of part system 1 will be applied.

12.1.8.8 2-part System Synchronous Thread Cutting

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

The 2-part system synchronous thread cutting allows 1st part system and 2nd part system to perform thread cutting simultaneously for the same spindle.
The 2-part system synchronous thread cutting has two command; the command (G76.1) for simultaneously cutting threads in two places, which is known as the "2-part system synchronous thread cutting cycle I" and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as the "2-part system synchronous thread cutting cycle II"

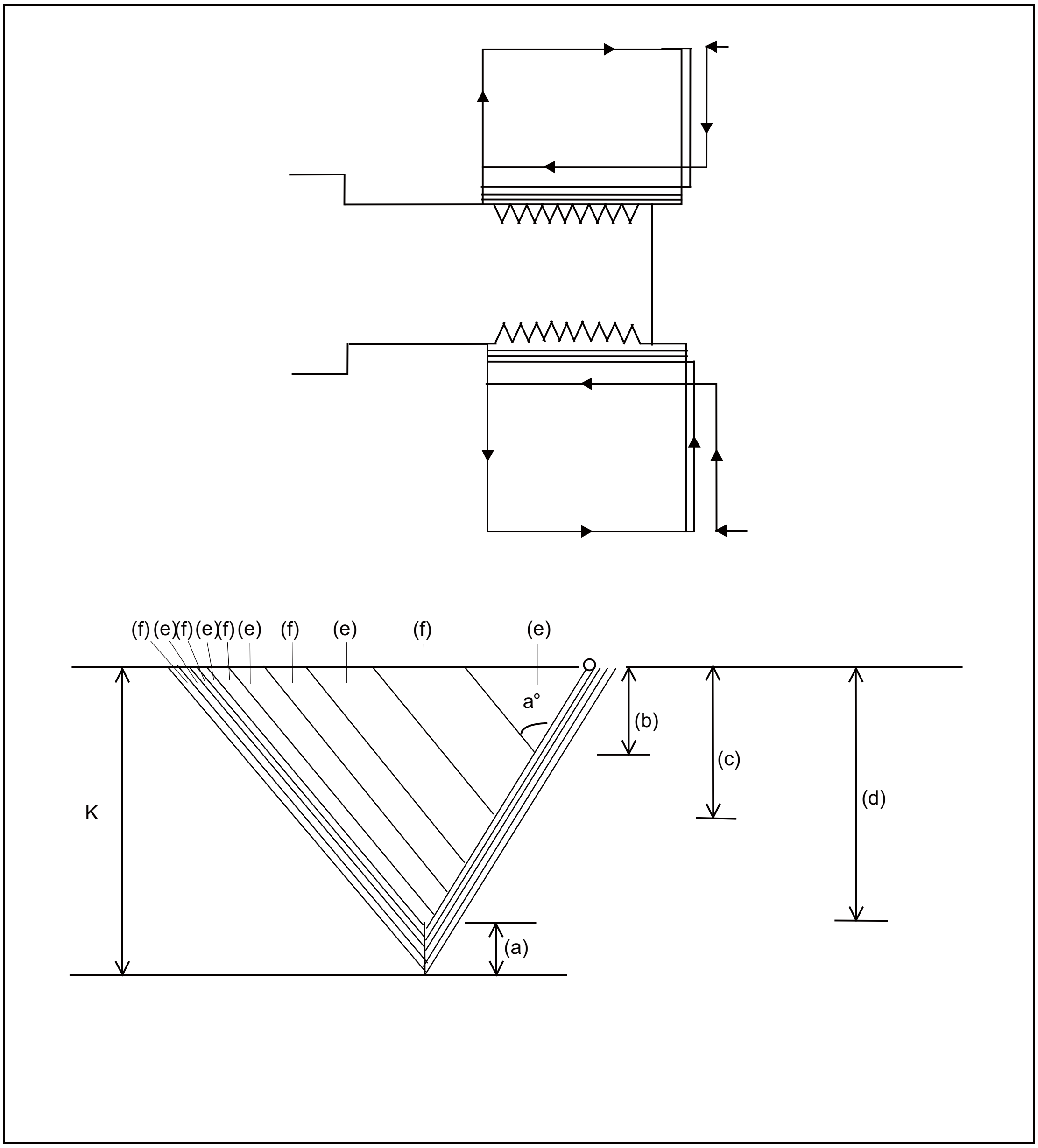
(1) 2-part System Synchronous Thread Cutting Cycle I (G76.1)
In the G76.1 cycle, G76.1 is issued simultaneously by 1st part system and 2nd part system, and the thread is cut in synchronization at the start and end of thread-cutting.



Command for 1st part system

Command for 2nd part system

(2) 2-part System Synchronous Thread Cutting Cycle II (G76.2)
G76.2 assumes the same thread cutting, and deeply cuts in with the cutting amount using 1st part system and 2nd part system alternately.



(a) Finishing allowance d
(b) △ d
(c) △ d× √‾ 2
(d) △ d× √‾ n
(e) Cutting with 1st part system
(f) Cutting with 2nd part system

12.1.8.9 Multi-part System Program Management

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Separate programs, used in each part system, can be managed under a common name in the multi-part system. This function facilitates the management of the process programs that are simultaneously executed in the multi-part systems.

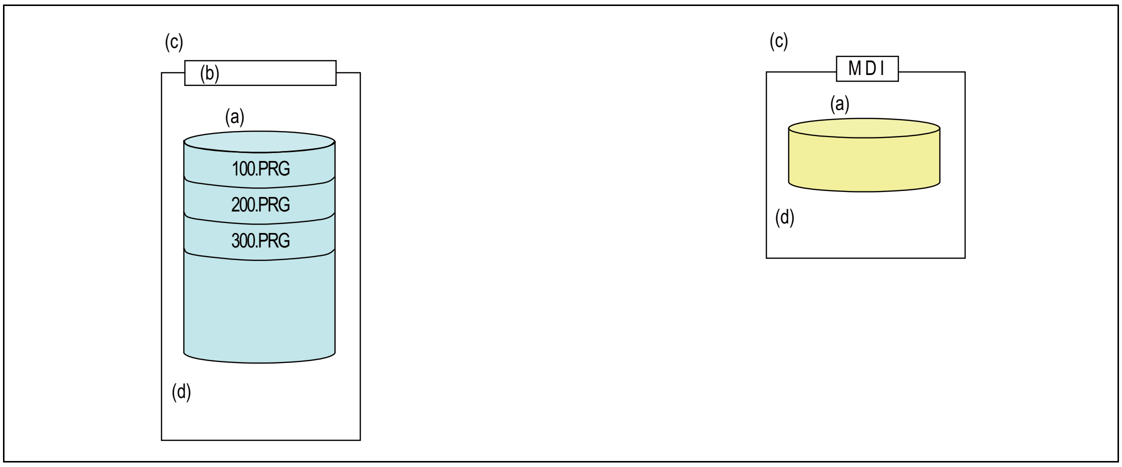
The multi-part system program management is turned valid/invalid with a parameter.

The followings are available when the multi-part system program management is valid:

- Executing the program search across all part systems in batch. (Note)
- Saving/editing the programs for each part system under one program name.
- Inputting/outputting programs of all part systems in batch.

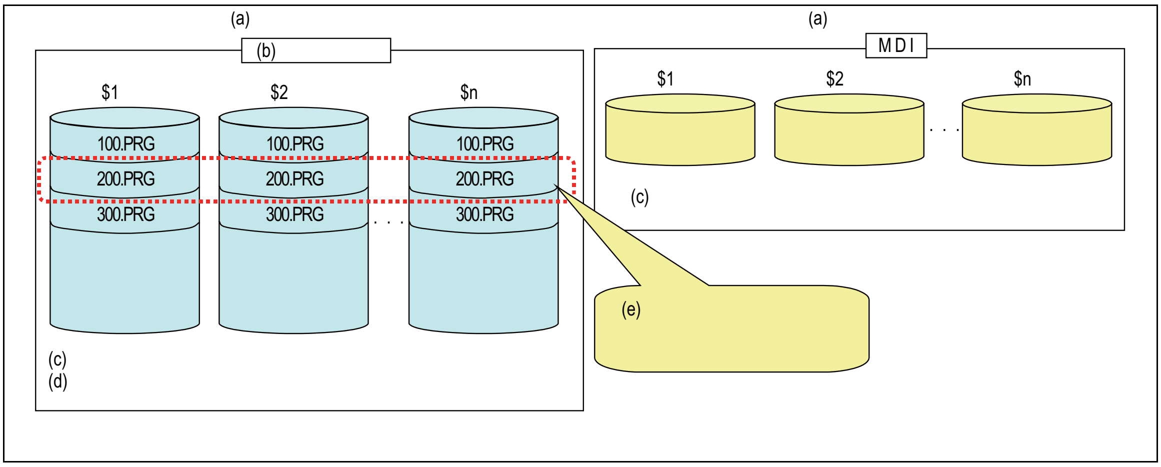
(Note) While the multi-part system program management is valid, there is an option whether the operation search, check search or restart search is executed across all part systems in batch or is executed for each part system separately.

<Multi-part system program management INVALID>



(a) Common for part systems
(b) Machining programs
(c) Program management common for part systems
(d) Programs are shared and managed across part systems.

<Multi-part system program management VALID>



(a) Program management for each part system
(b) Machining programs
(c) Programs are managed for each part system separately.
(d) Each part system can have its own program Nos.
(e) Same program Nos. can be managed across part systems in batch

12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The parameters set from the setting and display unit can be changed in the machining programs.
The data format used for the data setting is as follows.

G10 L70 ; Data setting start command
Pparameter No. Spart system No. Aaxis No. H ☐ data ; .... Bit parameter;
Pparameter No. Spart system No. Aaxis No. Ddata ; .... Numerical value parameter
Pparameter No. Spart system No. Aaxis No. <character string> ; ... Character string parameter

G11 Data setting end command

(Note 1) The sequence of addresses in a block must be as shown above.
When an address is commanded two or more times, the last command will be valid.

(Note 2) The part system No. is set in the following manner: "1" for 1st part system, "2" for 2nd part system, and so forth.
If the address S is omitted, the part system of the executing program will be applied.
As for the parameters common to part systems, the command of part system No. will be ignored.

(Note 3) The axis No. is set in the following manner: "1" for 1st axis, "2" for 2nd axis, and so forth.
If the address A is omitted, the 1st axis will be applied.
As for the parameters common to axes, the command of axis No. will be ignored.

(Note 4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation (☐) (0 to 7).

(Note 5) Only the decimal number can be commanded with the address D.
The value that is smaller than the input setting increment (#1003 iunit) will be round off to the nearest increment.

(Note 6) The character string must be put in angled brackets "<" and ">".
If these brackets are not provided, the program error (P33) will occur.
Up to 63 characters can be set.

(Note 7) Command G10L70, G11 in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.

(Note 8) 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The value of the workpiece coordinate systems selected can be set or changed by program commands. The tool offset amounts, which have been set from the setting and display unit, can be input by program commands.

(1) Workpiece coordinate system offset input

The position of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.

Gcode	Function
G10 L2 P0	External workpiece coordinate system setting
G10 L2 P1	Workpiece coordinate system 1 setting (G54)
G10 L2 P2	Workpiece coordinate system 2 setting (G55)
G10 L2 P3	Workpiece coordinate system 3 setting (G56)
G10 L2 P4	Workpiece coordinate system 4 setting (G57)
G10 L2 P5	Workpiece coordinate system 5 setting (G58)
G10 L2 P6	Workpiece coordinate system 6 setting (G59)
G10 L20 Pn (n=1 to 96)	Extended workpiece coordinate system setting (G54.1 P1 to P96)

The format for the workpiece coordinate system setting commands is shown below.

G10 L2(L20) Pp1 Xx1 Yy1 Zz1 ;	
G10 L2(L20)	: Workpiece coordinate system setting command
Pp1	: Workpiece coordinate system No.
Xx1,Yy1,Zz1	: Setting values

(Note 1) L2 can be omitted. Omitting Pp1 results in a program error. [M system]

(Note 2) L20 is used to designate the extended workpiece coordinate system.

(2) Tool compensation input

The tool compensation amounts, which have been set from the setting and display unit, can be input by program commands.

The command format differs between the [M system] and the [L system].

[M system]

Gcode	Function
G10 L10	Tool length shape compensation amount
G10 L11	Tool length wear compensation amount
G10 L12	Tool radius shape compensation amount
G10 L13	Tool radius wear compensation amount

The tool compensation input format is as follows.

G10 Ll1 Pp1 Rr1 ;	
G10 Ll1	: Command for setting compensation amount
Pp1	: Compensation No.
Rr1	: Compensation amount

(Note) When Ll1 has been omitted, the same operation will be performed as when L10 is selected. Omitting Pp1 results in a program error.

[L system]

Gcode	Function
G10 L10	Tool length compensation amount
G10 L11	Tool wear compensation amount

The tool compensation input format is as follows.

G10 L10(L11) Pp1 Xx1 Zz1 Rr1 Qq1 ;	
G10 L10(L11)	: Command for setting compensation amount
Pp1	: Compensation No.
Xx1	: X axis compensation amount
Zz1	: Z axis compensation amount
Rr1	: Nose R compensation amount
Qq1	: Hypothetical tool nose point

(Note) When Ll1 has been omitted, the same operation will be performed as when L10 is selected. Omitting Pp1 results in a program error.

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When tapping mode commands are issued, the NC system is set to the following internal control modes required for tapping.

1. Cutting override is fixed at 100%.
2. Deceleration commands at joints between blocks are invalid.
3. Feed hold is invalid.
4. Single block is invalid.
5. "In tapping mode" signal is output.

G code	Function
G63	Tapping mode ON

The tapping mode command will be canceled with the following commands:

- Exact stop check mode (G61)
- Automatic corner override (G62)
- Cutting mode (G64)
- High-accuracy control mode command (G61.1/G08P1) [M system]

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

12.1.10.2 Cutting Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When a cutting mode command is issued, the NC system is set to the cutting mode that enables smooth cutting surface to be achieved. In this mode, the next block is executed continuously without the machine having to decelerate and stop between the cutting feed blocks: this is the opposite of what happens in the exact stop check mode (G61).

G code	Function
G64	Cutting mode ON

The cutting mode command will be canceled with the following commands:

- Exact stop check mode (G61)
- Automatic corner override (G62)
- Tapping mode (G63)
- High-accuracy control mode command (G61.1) [M system]

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

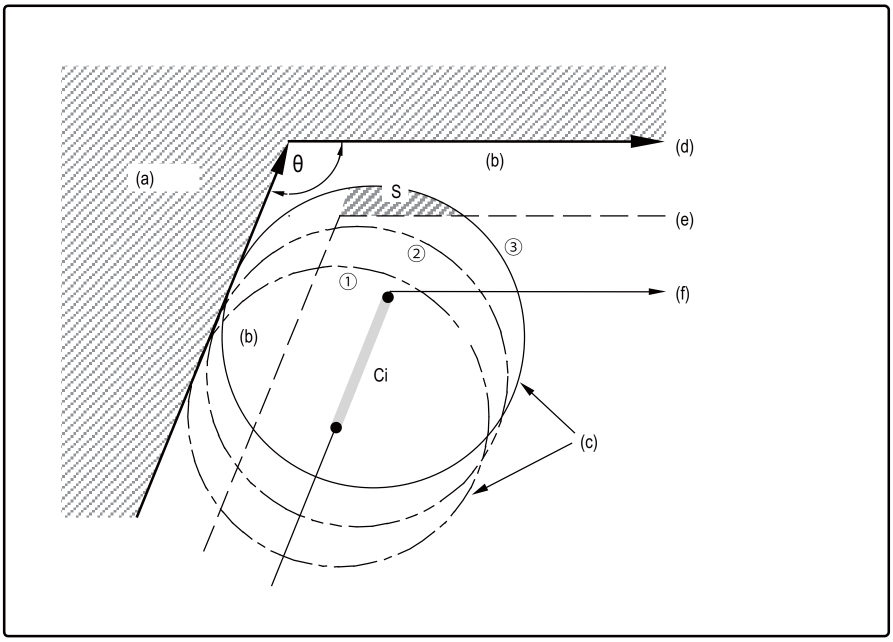
12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

To prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this function automatically applies an override on the cutting feed rate so that the cutting amount is not increased for a set time at the corner.
Automatic corner override is valid only during tool radius compensation.
The automatic corner override mode is set to ON by the G62 command and it is canceled by any of the G commands below.

G40 .... Tool radius compensation cancel
G61 .... Exact stop check mode
G63 .... Tapping mode
G64 .... Cutting mode
G61.1/G08P1 ..... High-accuracy control mode [M system]



(a) workpiece
(b) Machining allowance
(c) Tool
(d) Programmed path (finished shape)
(e) Workpiece surface shape
(f) Tool center path
θ : Max. angle at inside corner
Ci : Deceleration range (IN)

Operation

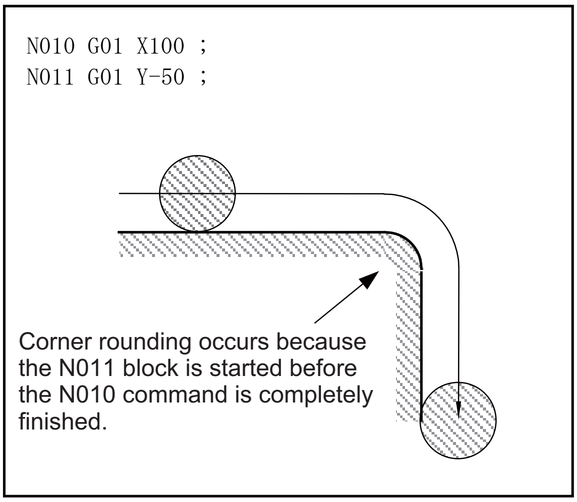
(a) When automatic corner override is not to be applied :
When the tool moves in the order of (1) -> (2) -> (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.

(b) When automatic corner override is to be applied :
When the inside corner angle θ in the figure above is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

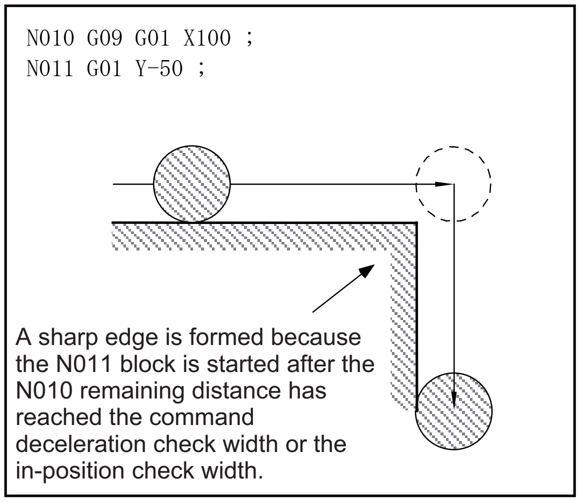
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another before executing the next block. This is effective to alleviate the machine shock and prevent the corner rounding when the feed rate of the control axis changes suddenly.

Without deceleration check



With deceleration check



The conditions for executing deceleration check are described below.

(1) Deceleration check in the rapid traverse mode

In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.

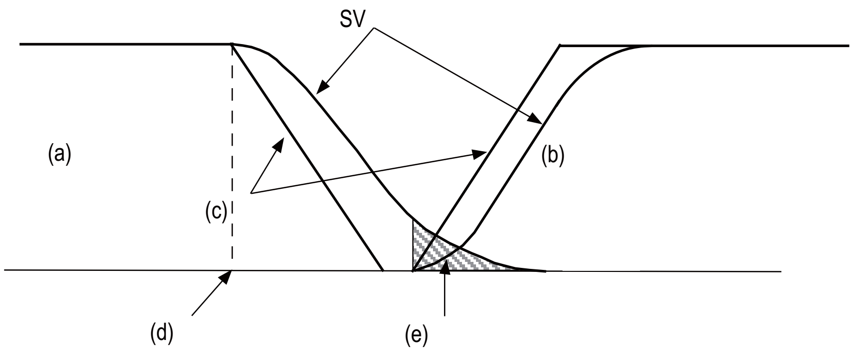
(2) Deceleration check in the cutting feed mode

In the cutting feed mode, the deceleration check is performed at the end of block when any of the conditions below is applicable before executing the next block.

(a) When G61 (exact stop check mode) is selected.
(b) When the G09 (exact stop check) is issued in the same block.
(c) When the error detect switch (external signal) is ON.

(3) Deceleration check system

Deceleration check is a system that executes the next block only after the command deceleration check is executed as shown below, and it has been confirmed that the position error amount, including the servo system, is less than the in-position check width (designated with parameter or with ",I" in same block).



SV: Servo
(a) Previous block
(b) Next block
(c) Command
(d) Block interpolation completion point
(e) In-position check width

12.2.2.1 Exact Stop Check Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

A deceleration check is performed when the G61 (exact stop check mode) command has been selected.
G61 is a modal command. The modal command is released by the following commands.

G62 .... Automatic corner override
G63 .... Tapping mode
G64 .... Cutting mode
G61.1/G08P1 .... High-accuracy control mode [M system]

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

12.2.2.2 Exact Stop Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

A deceleration check is performed when the G09 (exact stop check) command has been designated in the same block.
The G09 command should be 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 Detection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

To prevent rounding of a corner during cutting feed, the operation can be changed by turning an external signal switch ON so that the axis decelerates and stops once at the end of the block and then the next block is executed.
The deceleration stop at the end of the cutting feed block can also be commanded with a G code.
Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.4 Programmable In-position Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

",I" address is used to designate the in-position width for a linear interpolation command 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 Xx1 Zz1 Ff1 ,Ii1 ;

Xx1, Zz1	: Linear interpolation coordinate position of each axis
Ff1	: Feed rate
,Ii1	: In-position width

As with linear interpolation, ",I" address is used to designate the in-position width for a positioning command from the machining program.

G00 Xx1 Zz1 ,Ii1 ;

Xx1, Zz1	: Positioning coordinate position of each axis
,Ii1	: In-position width

In-position check operation

As for G01, after it is verified that the position error of 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.
As for G00, after it is verified that the position error of the positioning command (G00: rapid traverse) is less than the in-position width of this command, the execution of the next block is commenced.

12.3 High-speed and High-accuracy Functions [kBPM:k Block per Minute]

12.3.1 High-speed Machining Mode I (G05P1) Max.[kBPM]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 16.8	△ 16.8	△ 16.8	△ 16.8	△ 16.8	△ 16.8
L system	—	—	—	—	—	—

This function runs a machining program that approximates a free curve with fine segments at a high speed.
This is effective in increasing the speed of machining dies with a free curve.
The command format is as follows.

G05 P1	: High-speed machining mode I start
G05 P0	: High-speed machining mode I cancel

In addition to the G05P0 command, the high-speed machining mode I is canceled with the following commands.

- High-speed machining mode II (G05P2)
- High-speed high-accuracy control I (G05.1Q1)
- High-speed high-accuracy control II (G05P10000)

Fine segment capacity
G1 block fine segment capacity (for 1mm segment)

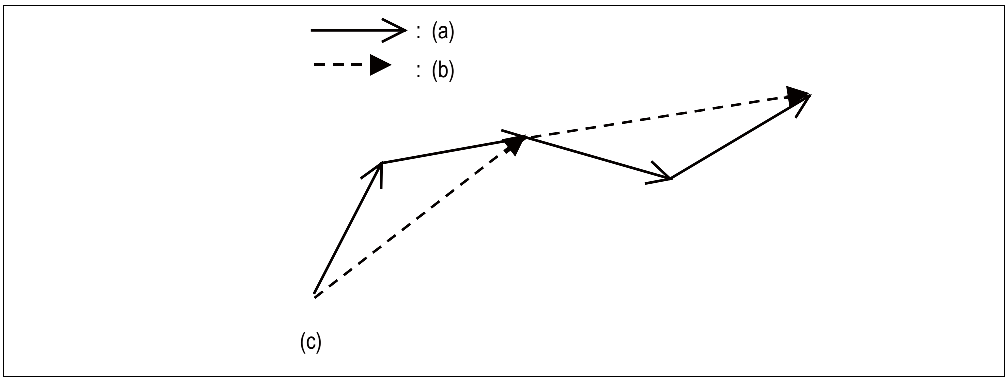
Mode	Command	Maximum feed rate when 1mm segmentG1 block is executed
High-speed machining mode Ⅰ	G05 P1	16.8 m/min

The above performance applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During G61.1 high-accuracy control mode or during cutting mode (G64)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

(1) During the high-speed machining mode I, the blocks are pre-read. If the length of the block is sufficiently short in respect to the command speed, and enough to be completed with one interpolation calculation, several blocks may be combined and converted into a block length that can be completed with one interpolation.



(a) Program command
(b) Command when program commands are combined and converted
(c) Command in high-speed machining mode I

(2) Override, maximum cutting speed clamp, single block operation, dry run, manual interruption, graphic trace and high-accuracy control mode are valid even during the high-speed machining mode I.

12.3.2 High-speed Machining Mode II (G05 P2) Max.[kBPM]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 67.5	△ 168	△ 168	△ 67.5	△ 168	△ 168
L system	—	—	—	—	—	—

This function runs a machining program that approximates a free curve with fine segments at a high speed.
This is effective in increasing the speed of machining dies with a free curve.
The command format is as follows.

G05 P2	: High-speed machining mode II start
G05 P0	: High-speed machining mode II cancel

In addition to the G05P0 command, the high-speed machining mode II is canceled with the following commands.

- High-speed machining mode I (G05P1)
- High-speed high-accuracy control I (G05.1Q1)
- High-speed high-accuracy control II (G05P10000)

Fine segment capacity
G1 block fine segment capacity (for 1mm segment)

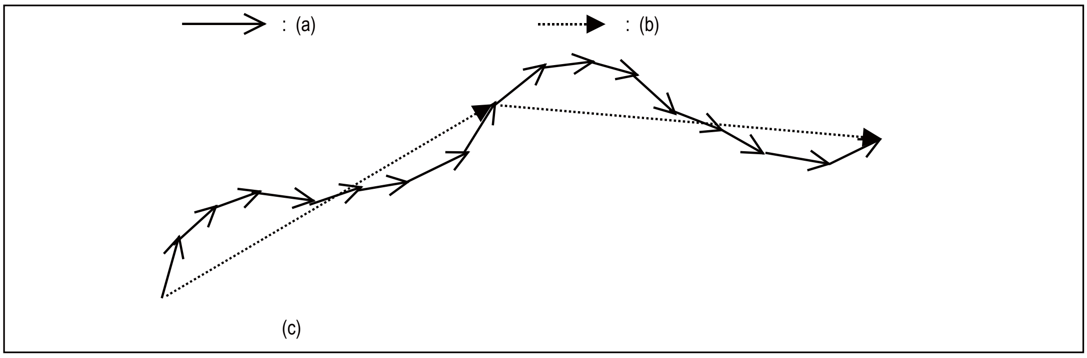
Mode	Command	Maximum feed rate when 1mm segmentG1 block is executed
Standard mode	G05 P0	16.8 m/min
High-speed machining mode Ⅱ	G05 P2	168.0 m/min(M730/M750) 67.5 m/min(M720)

The above performance applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- Block containing only axis name and movement amount (Does not contain macro or variable command)
- During G61.1 high-accuracy control mode or during cutting mode (G64)
- During tool radius compensation cancel (G40)

It may not be possible to attain the specified feed rate if deviated from the above conditions.

(1) During the high-speed machining mode II, the blocks are pre-read. If the length of the block is sufficiently short in respect to the command speed, and enough to be completed with one interpolation calculation, several blocks may be combined and converted into a block length that can be completed with one interpolation.



(a) Program command
(b) Command when program commands are combined and converted
(c) Command in high-speed machining mode II

(2) Override, maximum cutting speed clamp, single block operation, dry run, manual interrupt, graphic trace and high-accuracy control mode are valid even during the high-speed machining mode II.

(3) When using the high-speed machining mode II, adjust the parameters to eliminate the speed fluctuation at the arc and straight line and arc and arc joints.

12.3.3 High-speed High-accuracy Control 1 (G05.1Q1) Max.[kBPM] (1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 16.8	△ 33.7	△ 33.7	△ 16.8	△ 33.7	△ 33.7
L system	—	—	—	—	—	—

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed and with a high accuracy.
It is effective in increasing the speed at which dies with free-form curved surfaces are machined.

Command format

G05.1 Q1 ;	**High-speed high-accuracy control I ON**
G05.1 Q0 ;	**High-speed high-accuracy control I OFF**

The performance of simultaneous 3 axes and 1mm-length fine-segment is as follows.

High-speed high-accuracy control 1 mode	Performance of fine-segment length		Program restriction
	No radius compensation	Radius compensation	
Invalid	16.8m/min	16.8m/min	None
Valid	33.7m/min	33.7m/min	Available

Program restrictions
The machining speed may drop depending on the number of characters in one block.
During high-speed high-accuracy control I operation using tape operation, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.

Modes which can be designated by commands
The modal statuses when the G05.1 Q1 command is assigned must be as shown in the table below.
If the conditions listed are not satisfied, a program error will result.

Function	G code modal
Tool radius compensation mode	G40
Tool length compensation mode	G49
G code mirror image	G50.1
Mirror image by parameter setting	Cancel
Mirror image by external input	Cancel
Cutting mode	G64
Macro modal call mode	G67
Coordinate conversion mode	G69
Fixed cycle mode	G80
Per-revolution feed	G94
Constant surface speed control mode	G97
Interrupt-type macro mode	M97

12.3.4 High-speed High-accuracy Control 2 (G5P10000) Max.[kBPM] (Limited to 1-part System Configuration)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 67.5	△ 168	△ 168	△ 67.5	△ 168	△ 168
L system	—	—	—	—	—	—

(Note) Maximum feedrate changes according to the axis configuration and operation condition.

This function runs machining programs, in which free-form curved surfaces have been approximated by fine-segments, at high speed and with a high accuracy. It is effective in increasing the speed at which dies with free-form curved surfaces are machined.

Command format

G05 P10000 ; High-speed high-accuracy control II ON
G05 P0 ; High-speed high-accuracy control II OFF

Program restrictions
The machining speed may drop depending on the number of characters in one block.
During high-speed high-accuracy control II operation using tape operation, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.
When the fairing is valid (parameter setting), if the successive fairing is executed in the machining program, the performance of fine-segment execution may decelerate less than the value described in the above table.
In the network connection, the value described in the above table may not be guaranteed depending on the state.

Modes which can be designated by commands
The modal statuses when the G05 P10000 command is assigned must be as shown in the table below.
If the conditions listed are not satisfied, a program error will result.

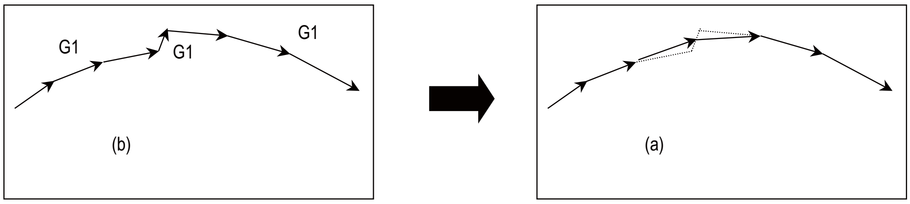
Function	G code modal
Tool radius compensation mode	G40
Mirror image by G code	G50.1
Mirror image by parameter setting	Cancel
Mirror image by external input	Cancel
Cutting mode	G64
Macro modal call mode	G67
Coordinate conversion mode	G69
Fixed cycle mode	G80
Per-revolution feed	G94
Constant surface speed control mode	G97
Interrupt-type macro mode	M97

Additional functions when high-speed high-accuracy control 2 mode is ON

(1) Fairing

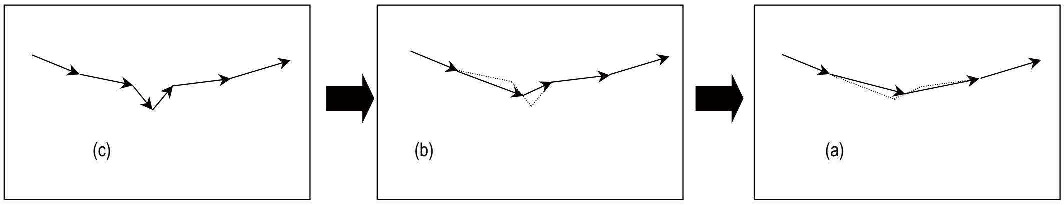
If the protruding path (zigzagging path) is shorter than the parameter setting values in the machining program generated with a CAM, etc., this function can be used to eliminate the protruding paths smaller than the setting value so that the front and back paths are smoothly connected.

This function is valid only for continuous linear commands (G1).



(a) After fairing
(b) Before fairing

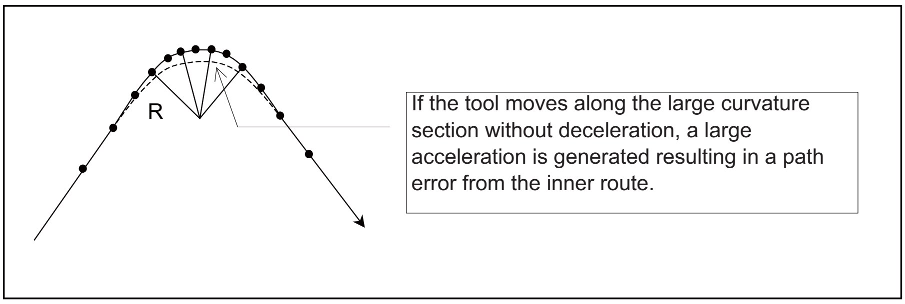
If there is any protruding path after fairing, fairing is repeated.



(a) After final faring
(b) After first fairing
(c) Before fairing

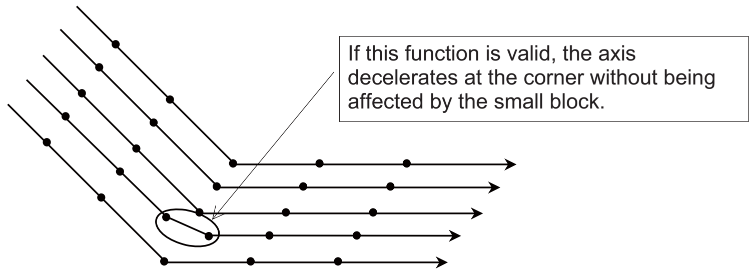
(2) Acceleration clamp speed

With the cutting feed clamp speed during the high-speed high-accuracy control 2 mode, when the following parameter is set to "1", the speed is clamped so that the acceleration generated by each block movement does not exceed the tolerable value. This function clamps the speed optimally even at a section where" angle change at each block is small but entire curvature is large" such as shown below. The tolerable acceleration value is calculated from the parameter setting values.



(3) High-speed mode corner deceleration

Conventionally during high-accuracy control, if the angle is large between the adjacent blocks in the machining program, this function automatically decelerates so that the acceleration generated when passing through the corner is within the tolerable value.
If a small block is inserted at the corner section with the machining program generated with the CAM, etc., the corner passing speed will not match the periphery. This can affect the machining surface.
If this type of small block is inserted when using high-speed mode corner deceleration, the corner will be largely judged by the parameter settings.
The small block is excluded when the angle is judged, but it not excluded from the actual movement command.



12.3.5 High-accuracy Control 1 (G61.1/G08) (1st Part System Only)

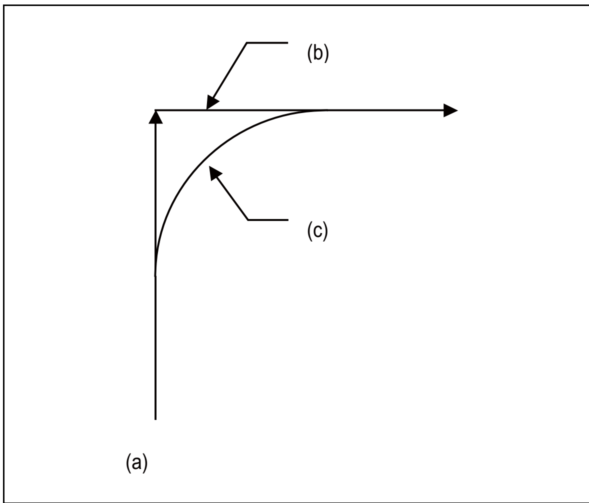
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

With this function, the error caused by the accuracy in control system during machining is to be improved. In order to achieve a high-accuracy control mode, turn initial high accuracy control ON with either parameter method or G code command method.

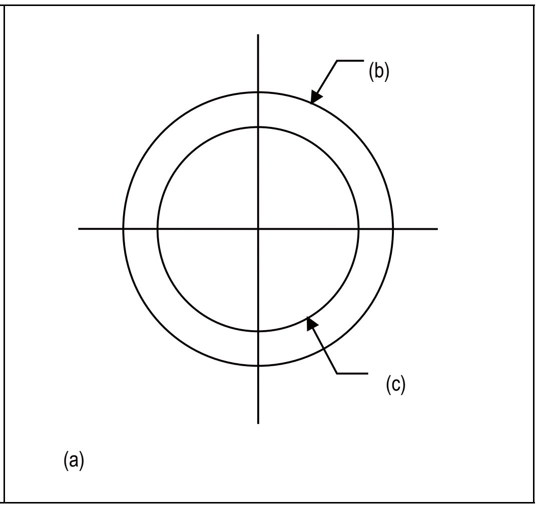
With the normal control method, there are problems as indicated below.

(1) Corner rounding occurs at the corner where two lines are connected because the next command movement starts before the previous command finishes. (Refer to Fig. 1)

(2) When cutting with the circle command, an error occurs further inside the commanded path, resulting in a smaller finish. (Refer to Fig. 2)



(a) Fig.1 Roundness at linear corner
(b) Commanded path
(c) Actual path



(a) Fig.2 Error by radius reduction with arc command
(b) Commanded path
(c) Actual path

This function uses the following six functions to reduce the shape error while minimizing the extension of machining time.

(1) Acceleration/deceleration before interpolation (linear acceleration/deceleration)
(2) Optimum speed control
(3) Vector accuracy interpolation
(4) Feed forward control
(5) Arc entrance/exit speed control
(6) S-pattern filter control

The high-accuracy control is commanded with ;
There are two types of command formats which can be changed with the parameters.

G61.1 Ff1 ;	
G61.1	: High-accuracy control mode ON
Ff1	: Feedrate command

High-accuracy control mode is validated from the block containing the G61.1 command.
"G61.1" high-accuracy control mode is canceled with one of the G code group 13's functions.

G08 P1(P0) ;	
G08	: High-accuracy control mode
P1	: High-accuracy control mode start
P0	: High-accuracy control mode end

"G08 P1" high-accuracy control mode is canceled with P0 in G08.
Command G08P_ in an independent block.
The decimal places below the decimal point are ignored for P address.
(Note) G code group in G08 is "0"; the priority is given to the function of the G code group 0 over that of the G code group 13. After "G08 P1" is commanded, G code group 13 is changed automatically to G64 (cutting) mode. Other command of "13" results in error. Even if high-accuracy control mode is canceled by "G08 P0" command, G64 (cutting) mode will not be changed. If you want to return to the function of the G code group "13" when "G08 P1" has been commanded, command again after high-accuracy control mode is canceled.

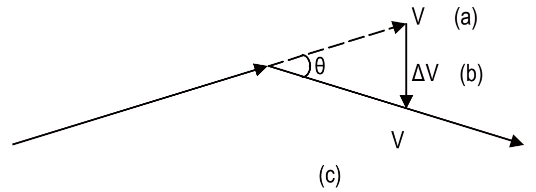
(1) Acceleration/deceleration before interpolation

By accelerating/decelerating before interpolation, the machining shape error generated by smoothing can be eliminated, 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

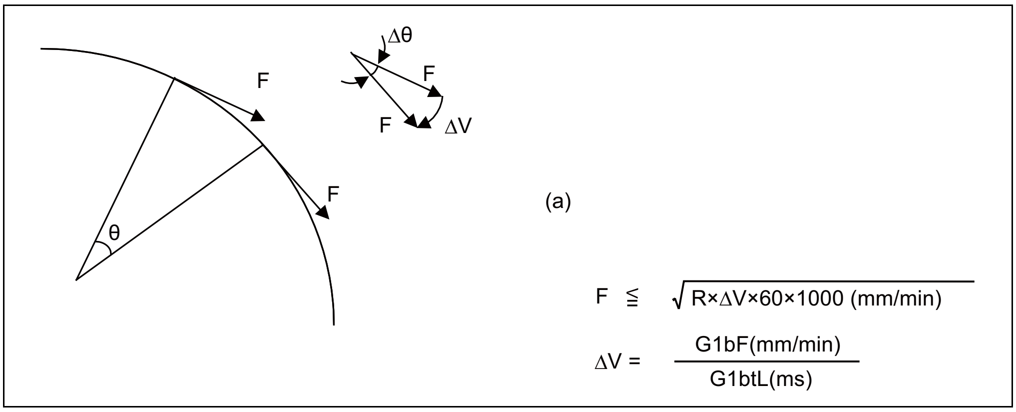
By calculating the angle of the seam between blocks, and carrying out acceleration/deceleration control in which the corner is passed at the optimum speed, highly accurate edge machining can be realized.
When the corner is entered, that corners optimum speed (optimum corner speed) is calculated from the angle with the next block. The machine decelerates to that speed in advance, and then accelerates back to the command speed after the corner is passed.



(a) Speed before entering the corner
(b) Speed change at the corner
(c) Speed after the corner is passed

(3) **Arc speed clamp**

During circular interpolation, even when moving at a constant speed, acceleration is generated as the advance direction constantly changes. When the arc radius is large enough compared to the commanded speed, control is carried out at the commanded speed. However, when the arc radius is relatively small, the speed is clamped so that the generated acceleration does not exceed the tolerable acceleration/deceleration speed before interpolation calculated with the parameters. This allows arc cutting to be carried out at an optimum speed for the arc radius.



F : Commanded speed (mm/min)

R : Commanded arc radius (mm)

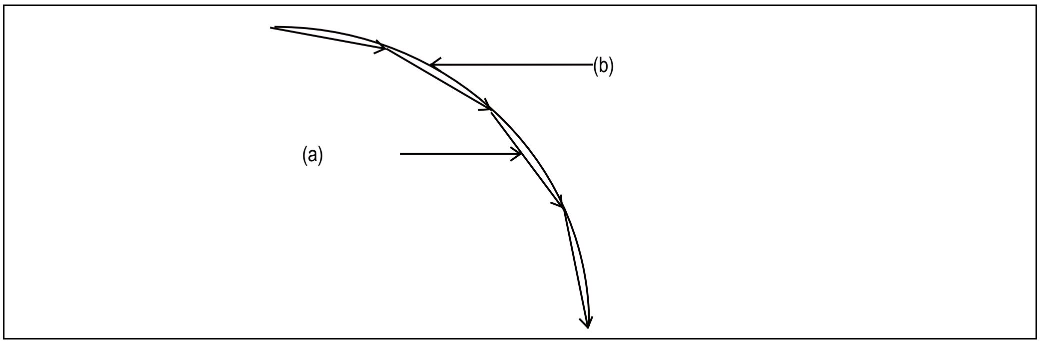
Δ θ : Angle change per interpolation unit

Δ V : Speed change per interpolation unit

(a) The tool is fed with the arc clamp speed F so that Δ V does not exceed the tolerable acceleration/deceleration speed before interpolation Δ V.

(4) **Vector accuracy interpolation**

When a fine segment is commanded and the angle between the blocks is extremely small (when not using optimum corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.

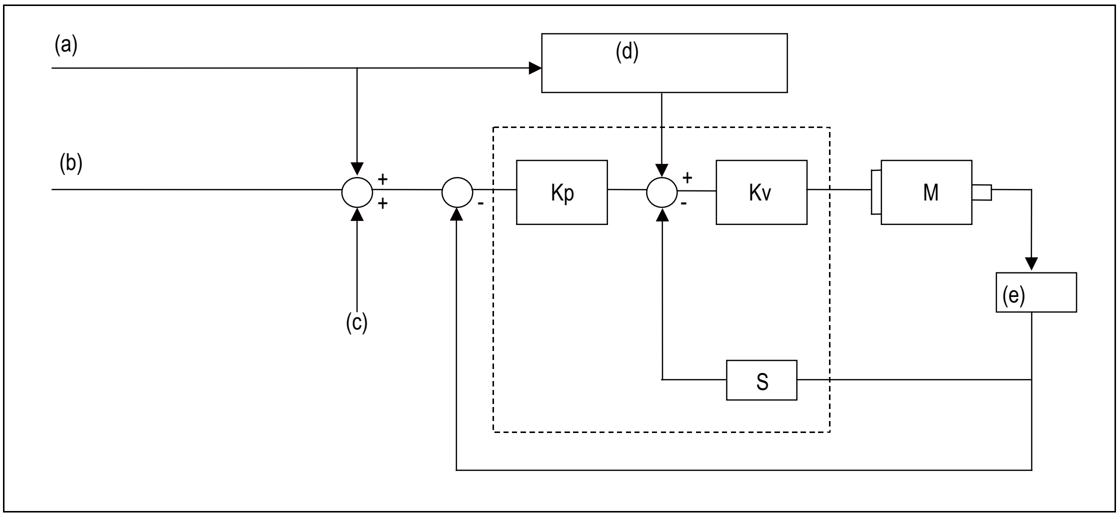


(a) Commanded path

(b) Vector accuracy interpolation

(5) Feed forward control

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



(a) Command during acceleration/deceleration before interpolation
(b) Command during acceleration/deceleration after interpolation
(c) Machine error compensation amount
(d) Feed forward control
(e) Detector
Kp : Position loop gain
Kv : Speed loop gain
M : Motor
S : Derivative

(6) Arc entrance/exit speed control

There are cases when the speed fluctuates and the machine vibrates at the joint from the straight line to arc or from the arc to straight line.

This function decelerates to the deceleration speed before entering the arc and after exiting the arc to reduce the machine vibration. If this is overlapped with corner deceleration, the function with the slower deceleration speed is valid.

(7) S-pattern filter control

This control interpolates further smoothing the changes in the segments distributed to each axis element by vector accuracy interpolation. With this, the fluctuation amplified by feed forward control is reduced and the effect onto the machine is reduced.

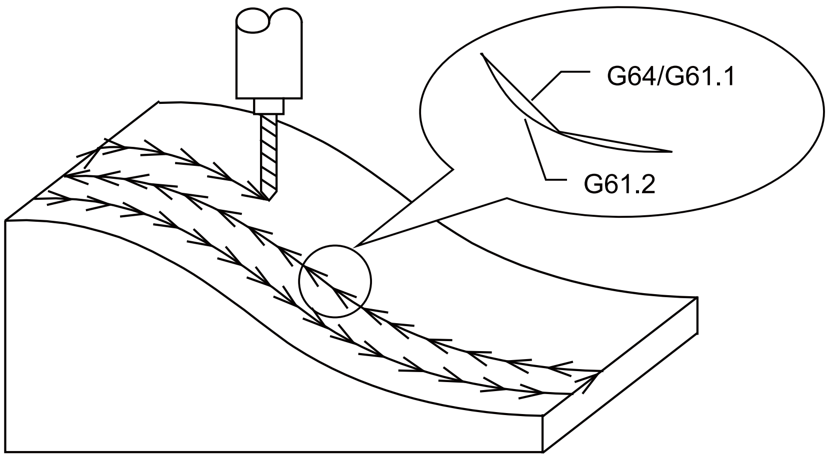
(8) Each axis arc radius reduction error compensation control

When the roundness at the machine end is, compared to the basic circle, expanded at an axis creating an ellipsis state, compensation is carried out for each axis to make a perfect circle.

12.3.6 High-accuracy Spline Interpolation1 (G61.2)(1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

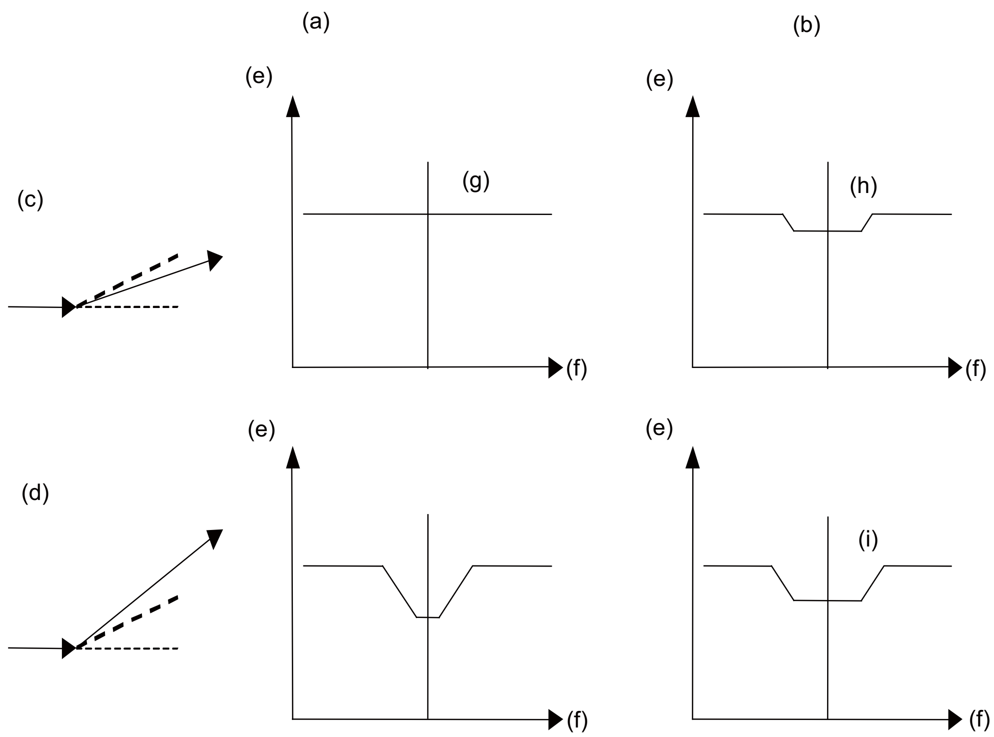
This function automatically generates spline curves that smoothly pass through rows of dots designated by a fine-segment machining program, and performs interpolation for the paths along the curves. This enables high-speed and high-accuracy machining to be achieved.
To use this function, the high-accuracy control function 1 (G08P1) is required.



12.3.8 SSS Control (1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

With normal high-accuracy control, because the angle between two blocks is compared with the corner deceleration angle to determine whether to execute corner deceleration between the blocks, there may be a case where sudden speed fluctuation occurs between blocks with an angle close to the corner deceleration angle.
With SSS (Super Smooth Surface) control, the large area path information is used instead of just the angle between the blocks. Thus, optimum speed control that is not adversely affected by minute steps or waviness is possible. This enables machining with fewer scratches and streaks on the cutting surface compared to the normal high-accuracy control function.



(a) Conventional optimum corner deceleration
(b) SSS control
(c) When smaller than corner deceleration angle
(d) When larger than corner deceleration angle
(e) Feedrate
(f) Time
(g) Do not decelerate
(h) Decelerate according to angle
(i) Gradually speed deceleration

[Features of SSS control]

(1) This is valid for machining a mold with a smooth shape using a fine segment program.
(2) This speed control is not easily affected by the error found in the path.
(3) Even if corner deceleration is not required, the speed is clamped if the predicted acceleration is high.

The length of the path direction recognized with SSS control can be adjusted with the parameter. The range is increased as the setting value increases, and the effect of the error is reduced.

(Note 1) When using this function, an option corresponding to the SSS-controlled high-accuracy control mode is required, as well.
(Note 2) The performance of fine-segment execution during SSS control differs depending on the type of high-accuracy control mode and machine model to be combined.

12.3.9 High-accuracy Acceleration/Deceleration Time Constant Extension (1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

This extends the upper limit of cutting feed time constant from 5,000[ms] to 30,000[ms] for acceleration/deceleration before interpolation.

12.3.10 Machining Condition Selection I (1st Part System Only)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	—	—	—	—	—	—

The machining condition parameter set which is consists of parameters related to the high-accuracy control can be configured in advance for each machining application (such as part machining or die machining) or machining process (such as rough or finishing), and it can be switched according to the purpose.
This can easily perform the machining which meets the conditions for the intended machining.

12.4 Programming Support Functions

12.4.1 Playback

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function enables creation of a program while proceeding with sample machining by manual (handle or job) feed or mechanical handle feed. A machining program can be edited by using the playback movement amount obtained by manual operation as the program command values.
Playback edit is disabled for the machining program in the mass-editing mode.

12.4.3 Simple Programming

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Create a part program by using NAVI MILL (for machining center system) or NAVI LATHE (for lathe system).

(1) The following machining processes can be edited.

M system	L system
- Hole drilling (Drilling, deep hole drilling, step, boring, tapping) - Face cutting (Circle, square) - Contour cutting (Circle, square, arbitrary shape) - Pocket machining (Circle, square, L pattern, U pattern) - EIA	- Turning (Outside dia., inside dia., front face) - Copying (Outside dia., inside dia., front face) - Threading (Outside dia., inside dia., front face) - Grooving (Outside dia., inside dia., front face) - Trapezoid grooving (Outside dia., inside dia., front face) - Hole drilling (Drilling, deep hole drilling, step, tapping) - EIA

(2) Cutting conditions are automatically determined from tool file and cutting condition file.
(3) The operation screen consists of LIST VIEW area (on the left) and OPERATION VIEW area (on the right). In the LIST VIEW area, the entire part program can be viewed at all time. In the OPERATION VIEW area, there are guide drawings related to each input item, which allows an easy data input.
(4) Program Checker enables the tool paths of a part program to be graphically traced. With this function, an input error can be detected at an earlier stage.
(5) Guidance function provides an operator with error recovery information.
(6) Part program is a macro-program-based NC program. Process commands can be added in the edit screen.
(7) The macro program above can be customized by the machine tool builder.

12.4.4 G code Guidance

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

G code guidance is a function to display illustration of the contents or movements of the commanded format for the G code currently under editing. This is used when creating or editing a machining program.
With this function, the G code contents under the editing process can be checked on the spot.

13

Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function compensates for the error (backlash) produced when the direction of the machine system is reversed.
The backlash compensation can be set in the cutting feed mode or rapid traverse mode.
The amount of backlash compensation can be set separately for each axis. It is set using a number of pulses in increments of one-half of the least command increments. 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

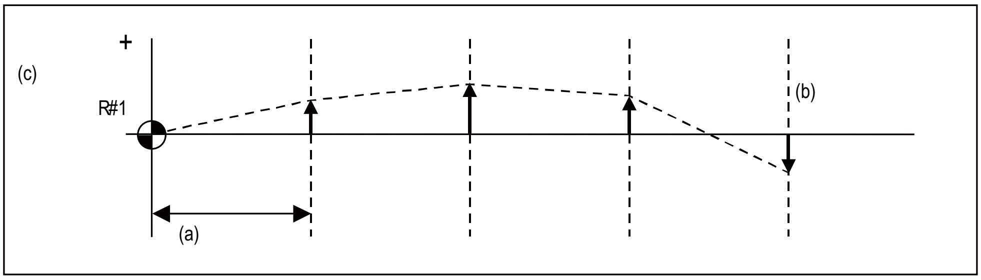
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The machine accuracy can be improved by compensating for the errors in the screw pitch intervals among the mechanical errors (production errors, wear, etc.) of the feed screws.
The compensation positions and amounts are stored in the memory by setting them beforehand for each axis, and this means that there is no need to attach dogs to the machine.
The compensation points are divided into the desired equal intervals.

1. Division intervals of compensation points : 1 to 9999999 (μ m)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 (output unit)
4. Number of compensation axes : 10 axes (including number of axes for memory-type relative position error compensation)

(1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed upon the compensation axis for which return to reference position is not made after the controller power is turned ON and the servo is turned ON.

(2) When the compensation axis is a rotary axis, select the dividing intervals so that one rotation can be divided.



(a) Division interval
(b) Compensation basic axis coordinate
(c) Compensation amount

(3) As shown in the figure above, highly individualized compensation control is exercised using the least output increment 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Machine accuracy can be improved by compensating a relative error between machine axes, such as a production error or time aging.
The compensation base axis and compensation execution axis are set by using parameters.
The compensation points are divided at any desired equal intervals.

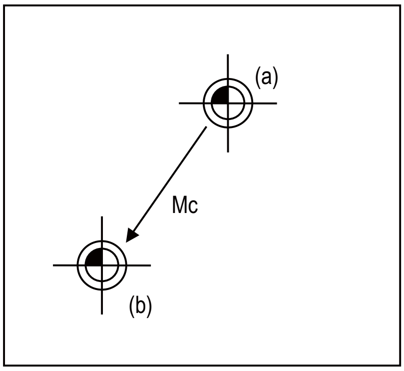
1. Compensation point dividing intervals : 1 to 9999999 (μ m)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 (output unit)
4. Number of compensation axes : 10 axes (including number of axes for pitch error compensation.)
   (1) The compensation position is set for the compensation axis whose reference position serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed upon the axis for which return to reference position is not made 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counter (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.



Mc:Compensation vector according to external machine coordinate system compensation
(a) Machine coordinate zero point when the external machine coordinate system offset amount is 0.
(b) Machine coordinate zero point

13.1.5 Circular Error Radius Compensation

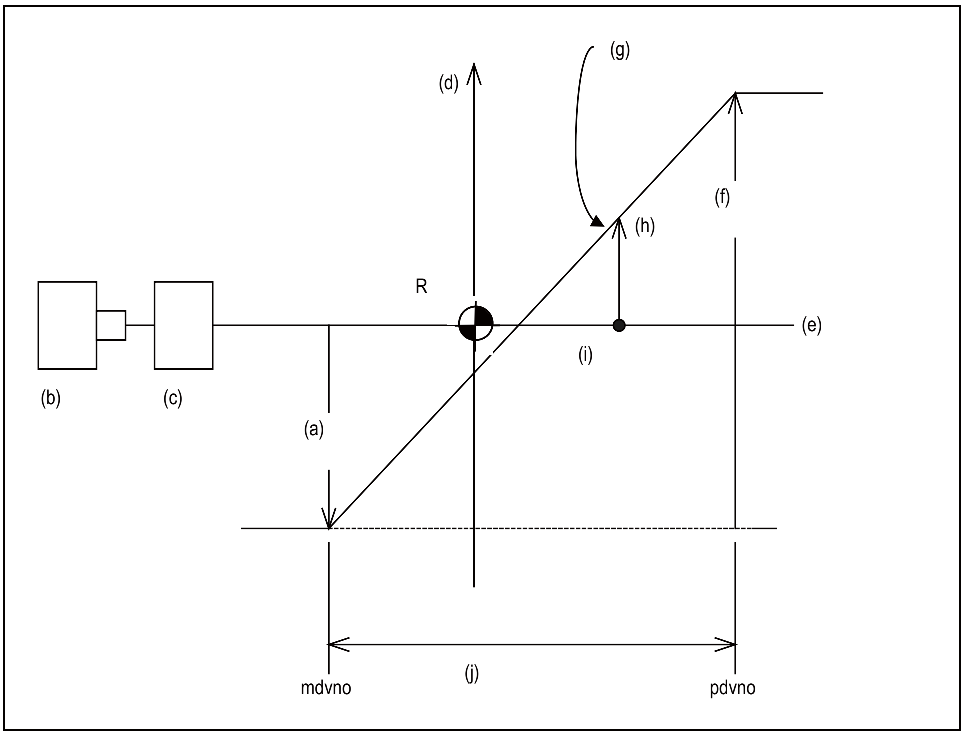
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

With commands designated during arc cutting, this function compensates for movement toward the inside of the arcs caused by a factor such as servo delay.

13.1.6 Ball Screw Thermal Expansion Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This compensates the axis feed error caused by the ball screw thermal expansion, etc. using the values set in the R register's thermal displacement compensation parameter.
The compensation amount at the end of the machine error compensation range (mdvno to pdvno) is set. The compensation line is obtained from the set compensation amount, and is linearly distributed in the machine error compensation range to attain smooth compensation. A compensation amount that changes during operation can also be compensated.



(a) offset (with sign)
(b) Motor
(c) Hold
(d) Compensation amount
(e) Ball screw
(f) maxcmp (with sign) Referring to "offset" position
(g) Compensation line
(h) Compensation amount (legcmp)
(i) Machine position
(j) Machine error compensation range (= thermal expansion compensation range)
R Reference position

(1) Conditions under which thermal displacement compensation is valid

- When function option is valid.
- When axis No. and parameter (offset, maxcmp) is set in R register.
- When pitch error compensation parameter is set correctly.

(2) Compensation operation

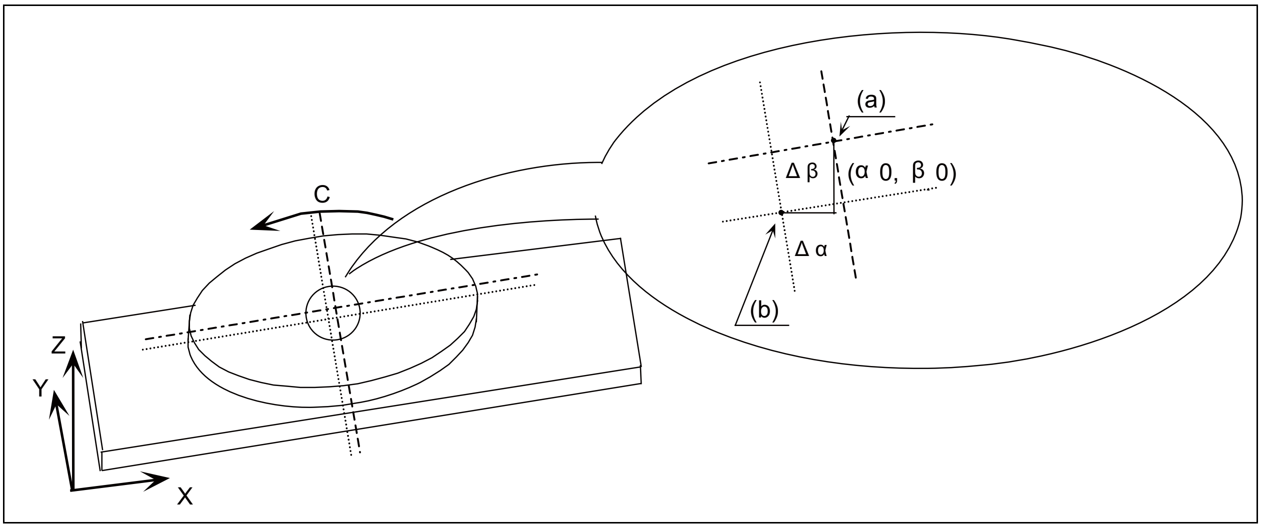
- The machine error compensation range is compensated with the thermal displacement compensation amount (absolute position compensation amount) obtained with the compensation line.
- The compensation amount is immediately calculated when thermal displacement compensation is validated.
- When the compensation amount is changed, the new compensation line is immediately obtained, and the range is compensated again with that compensation amount.
- Thermal displacement compensation can be used with machine error compensation. In this case, the sum of the machine error compensation amount and the thermal displacement compensation amount is added to the machine position and issued as the NC command.

13.1.7 Machine Rotation Center Error Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	―	△	△	―	△	△
L system	―	―	―	―	―	―

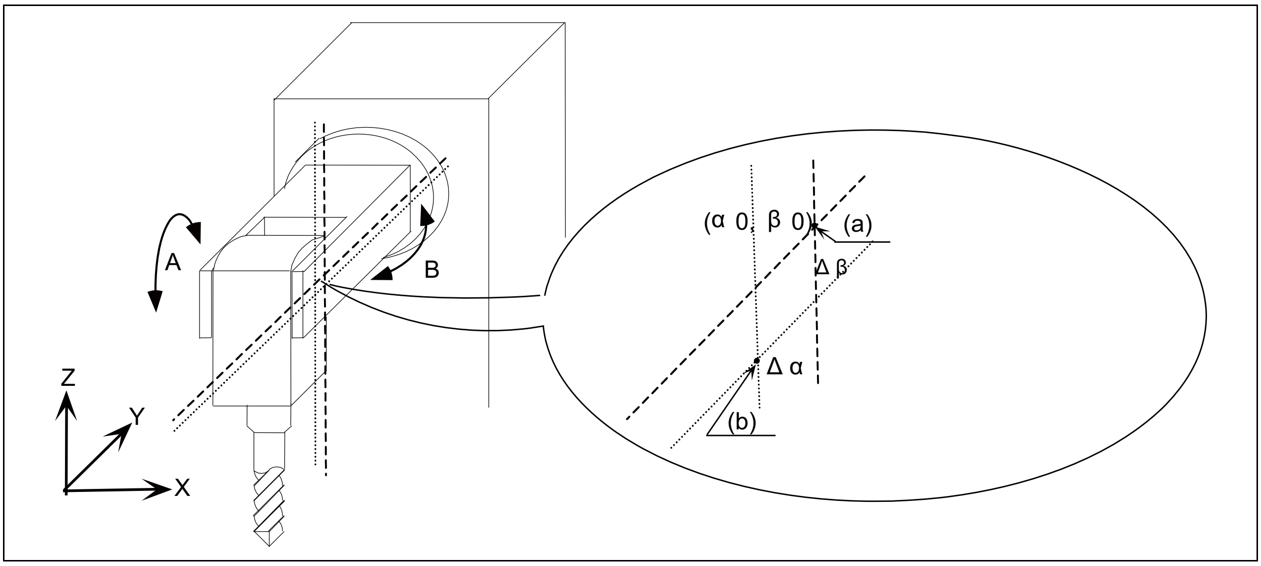
In the machine having a rotary axis, there may be a case where the actual rotation center deviates from the programmed rotation center. (In other words, "machine rotation center error" may be observed.) Higher-accuracy machining can be realized by compensating this error.
Error is compensated by dividing the error compensation amount, which depends on the rotation angle, into the orthogonal axes.

Table rotation center error



(a) Programmed rotation center
(b) Actual rotation center
$\Delta\alpha$ **Error in X-axis direction**
$\Delta\beta$ **Error in Y-axis direction**

Tool rotation center error



(a) Programmed rotation center

(b) Actual rotation center

$\Delta\ \alpha$ **Error in X-axis direction**

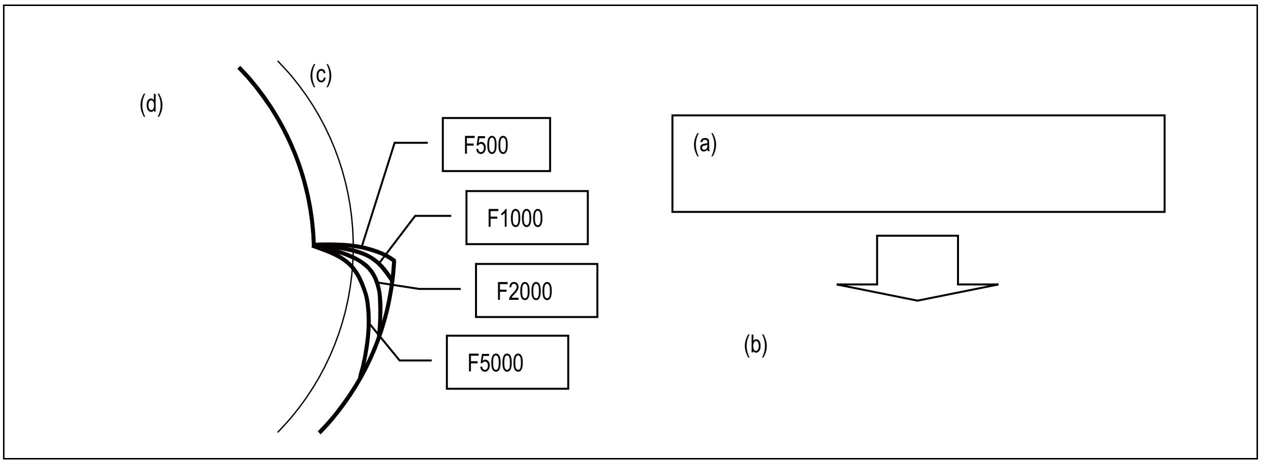
$\Delta\ \beta$ **Error in Y-axis direction**

13.1.8 Position-dependent Gradually Increasing-type Backlash Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

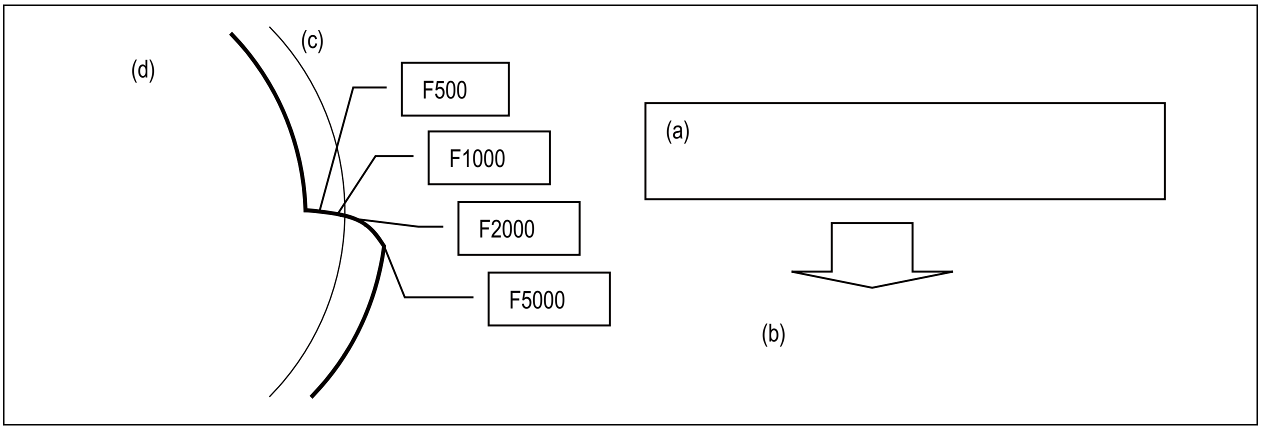
Usually, minute bumps are generated when the machine movement direction is reversed. (Lost motion)
However, due to various mechanical system factors, there may be cases where the bumps do not change in steps but gradually change after the direction reversal. (Gradually increasing-type lost motion)
Depending on the mechanical structure, the gradually increasing-type lost motion includes the type where the bump amount depends on "duration of time" from when the machine movement direction is reversed and the type where the bump amount depends on "distance". When the bump amount depends on "duration of time", compensation is possible with the OMR-II function. (13.2.4 OMR II(Backlash with Filter)).
With this function, the gradually increasing-type lost motion which depends on the distance from the point where the machine movement direction is reversed can be compensated by controlling the variation of backlash compensation amount according to the distance from the direction reversal point.

<When the bump amount changes according to speed>



(a) Shapes at rising edge of bumps are steep when the speed is low. → Bumps depend on time.
(b) Compensation with OMR-II function
(c) Basic circle
(d) Measurement result

<When the bump amount does not change according to speed>



(a) Shapes at rising edge of bumps do not change even when the speed changes. → Bumps depend on position
(b) Compensation with this function
(c) Basic circle
(d) Measurement result

13.1.9 Two-way Pitch Error Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Two-way pitch error compensation function is used to compensate the pitch error in each direction by setting the pitch error compensation amount when moving in the positive and negative direction.
This function reduces the difference of tool path between the positive and negative direction.

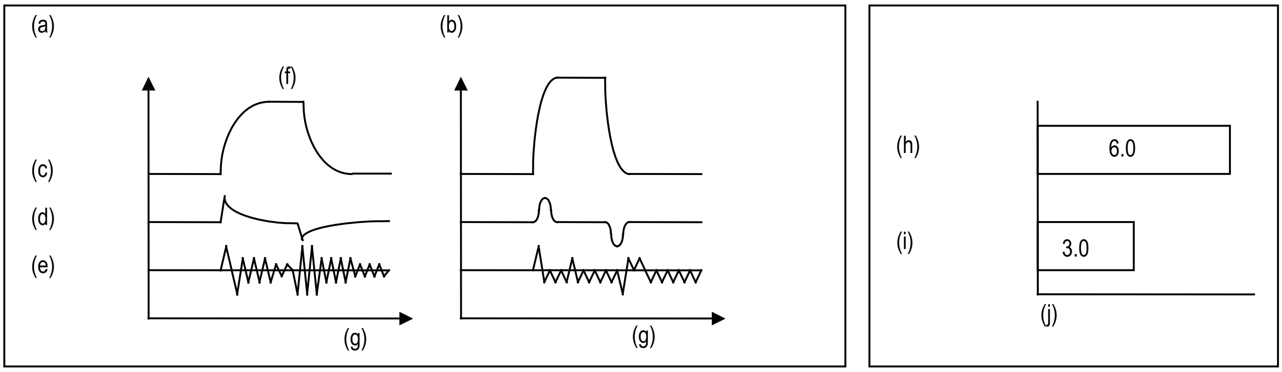
13.2 Dynamic Accuracy Compensation

13.2.1 Smooth High-gain (SHG) Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

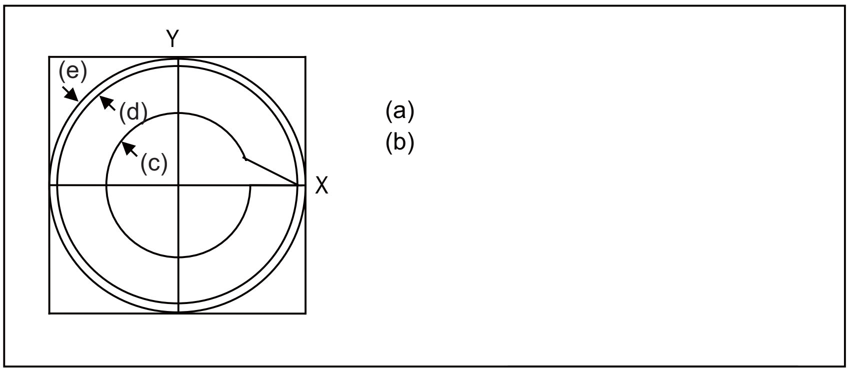
This is a high-response and stable position control method using the servo system. This SHG control achieves an approximately three-fold position loop gain equally compared to the conventional control method.
The features of the SHG control are as follows.

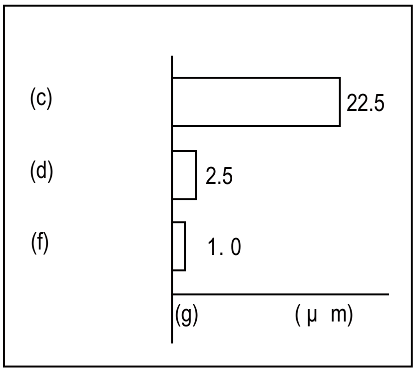
(1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/deceleration time constant can be shortened.)



(a) Conventional control (position loop gain = 33rad/S)
(b) SHG control (position loop gain = 50rad/S)
(c) Speed
(d) Current
(e) Machine vibration
(f) Step response
(g) Time
(h) Conventional control
(i) SHG control
(j) Machine vibration amount (μ m)

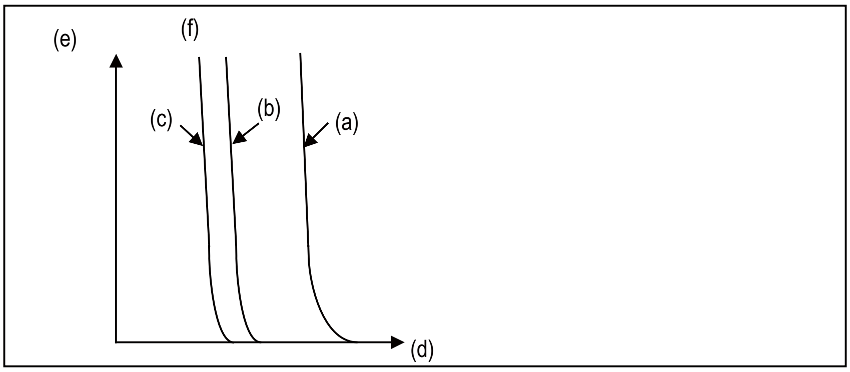
(2) The shape error is approx. 1/9 of the conventional control.

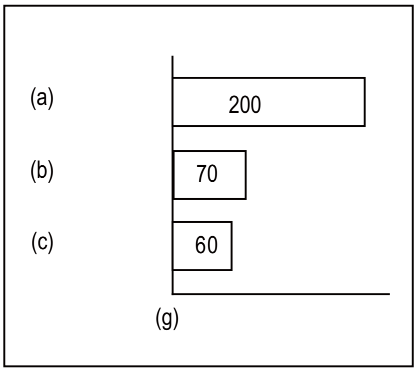




(a) Feed rate 3000mm/min.
(b) Radius 50mm
(c) Conventional control
(d) SHG control
(e) SHG control + FF (Feed forward)
(f) SHG control + FF
(g) Roundness error (μ m)

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



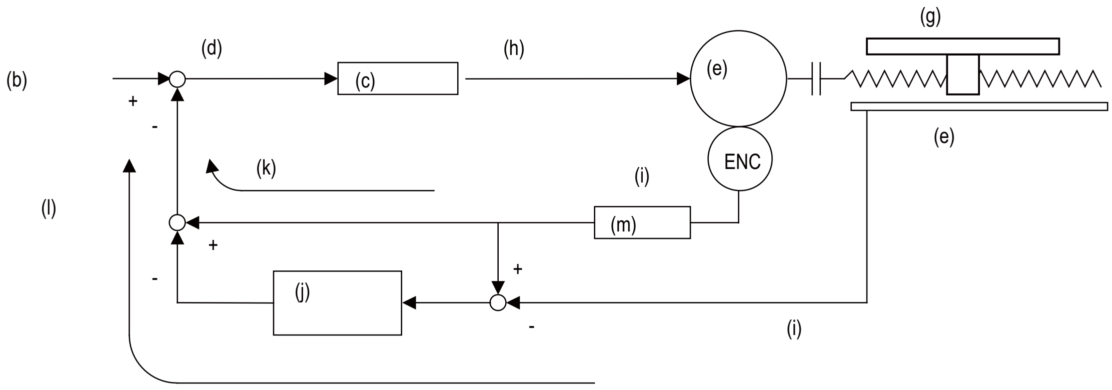


(a) Conventional control
(b) SHG control
(c) SHG control + FF (Feed forward)
(d) Time
(e) Droop
(f) Droop during rapid traverse deceleration
(g) Positioning time (ms)

13.2.2 Dual Feedback

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

If the motor and machine coupling or machine system's rigidity is low (ex. large machine, etc.) when using a closed loop system, the response during acceleration/deceleration will vibrate and cause overshooting. This can cause the position loop gain from increasing. The dual feedback function is effective in this case.
To validate the dual feedback function, use position feedback with a motor side detector in ranges with high acceleration to enable stable control. In ranges with low acceleration, use position feedback with the machine side detector (scale). This will make it possible to increase the position loop gain.
The machine side detector (scale) is required separately.



(a)

(a) Dual feedback control
(b) Position command
(c) Position control
(d) Position droop
(e) Servo motor
(f) Linear scale
(g) Table
(h) Speed command
(i) Position FB
(j) Primary delay filter
(k) High frequency FB element
(l) Low frequency FB element
(m) dead band

The state will approach the semi-closed loop system as the primary delay filter's time constant increases, so the position loop gain limit will increase. Note that the limit of the position loop gain increased with the dual feedback function is the same as the position loop gain limit for a semi-closed system that does not use a machine side detector (scale, etc.). In addition, the positioning time will increase as the primary delay filter time constant increases.

13.2.3 Lost Motion Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

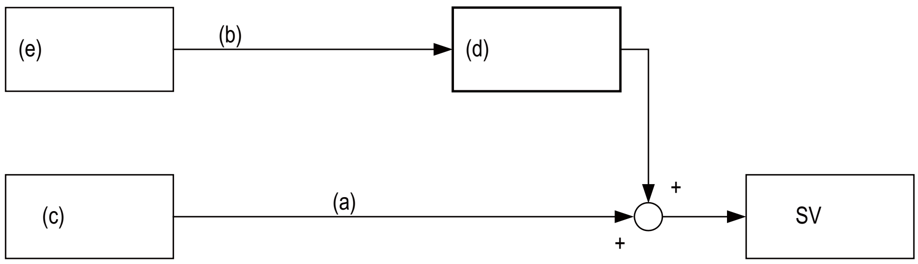
This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

13.2.4 OMR II (Backlash with Filter)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The OMR (Optimal Machine Response) control function estimates the machine or motor model (moment of inertia, clone friction, viscosity coefficient, etc.) that can cause a path error (error in actual tool path in respect to the path commanded with the program). High-accuracy machining is achieved by carrying out feed forward control based on that model. This allows error cased by quadrant protrusions during circular interpolation or quadrants on the inner side of the path to be greatly reduced.
OMR-II is a function that focuses on the quadrant protrusions, and improves the path error with this.
Quadrant path compensation is included in OMR-II.

The quadrant protrusion compensation function improves quadrant protrusions by issuing compensation to the backlash compensation amount to compensate the error when the machine system direction is reversed. By adding the compensated backlash compensation amount to the position command and sending it to the servo, the gradually increasing-type lost motion which occurs at quadrant changeover can be compensated.



(a) Position command (Fdt)
(b) Backlash compensation amount
(c) Interpolation process
(d) Quadrant protrusion compensation
(e) Machine error compensation
SV Servo

13.2.6 OMR-FF

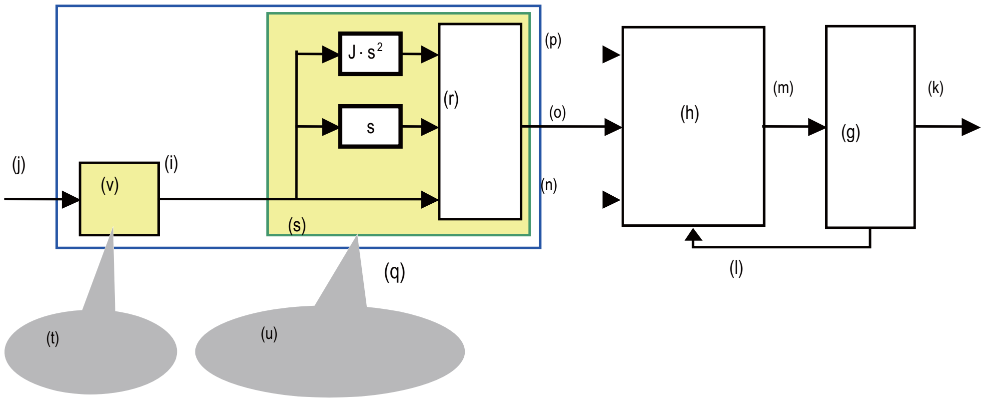
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The OMR (Optimal Machine Response) control estimates the causes (moment of inertia, clone friction, viscosity coefficient, etc.) of the path error (error in actual tool path in respect to the path commanded with the program) by making a model of the control target. Feed forward control is carried out based on that model.

With the OMR-FF control method, highly accurate machining can be achieved by using the command filter function to suppress vibration in addition to the reverse model feed forward function.
The reverse model feed forward function estimates the control target's vibration characteristics using a reverse model. The feed forward command for the appropriate position, speed and current can be obtained from this estimation. This allows the machine position to be tracked to the commanded position without causing vibration when using machine tools with low rigidity.
When the command filter function is also used, the high-range vibration elements in the command can also be suppressed.

This function can be validated under the following conditions.

- OMR-FF option is valid.
  (If the option is not valid, the conventional feed forward control will be applied.)
- The mode is the high-accuracy control mode.



(d) Set parameters:

(e) (1) Scale model time constant

(f) (2) Machine system resonance frequency ωp, anti-resonance frequency ωz, attenuation rate ζ, inertia J

(g) Machine system

(h) F/B controller

(i) Scale model position

(j) Target path

(k) Machine end position

(l) Motor position

(m) Motor torque

(n) Position command

(o) Speed FF

(p) Torque FF

(q) OMR-FF control

(r) Notch filter

(s) Machine system reverse model

(t) Determine response characteristics of machine end position

(u) Trace load position to scale model position without causing vibration

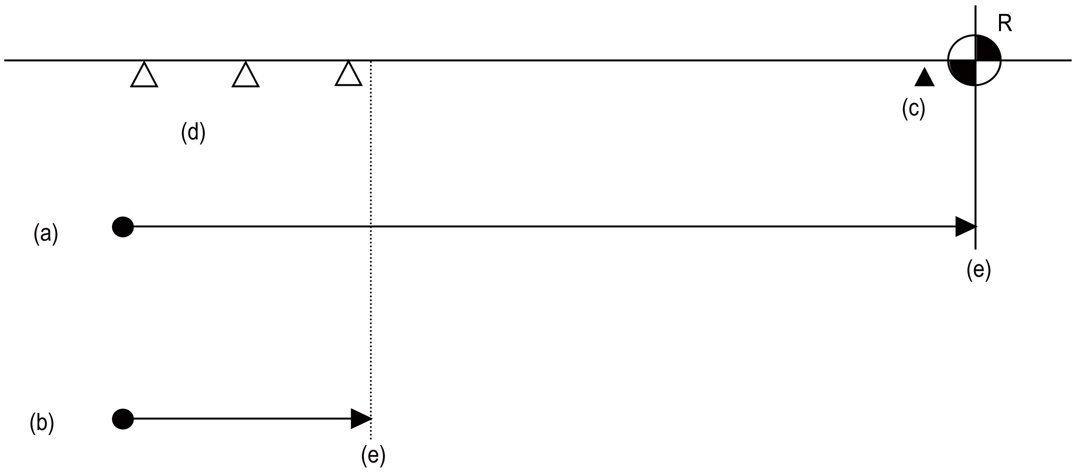
(v) Scale model

13.2.7 Distance-coded Reference Position Detection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This is the function where the scale with absolute address reference mark is used to establish the reference point in the relative position detection system.
When the reference point has not been established, it is necessary to move the axis to the reference point. On the other hand, with this function, the reference point is established by moving the axis only for several reference marks, resulting in a significant reduction of the axis movement amount.
No dog is used as the position is calculated using reference marks.

<Reference point establishment process>



(a) Ordinary method
(b) This function
(c) Dog
(d) Reference mark
(e) Reference point established

14

Automation Support Functions

14.1 Measurement

14.1.1 Skip

14.1.1.1 Skip

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

[M system]

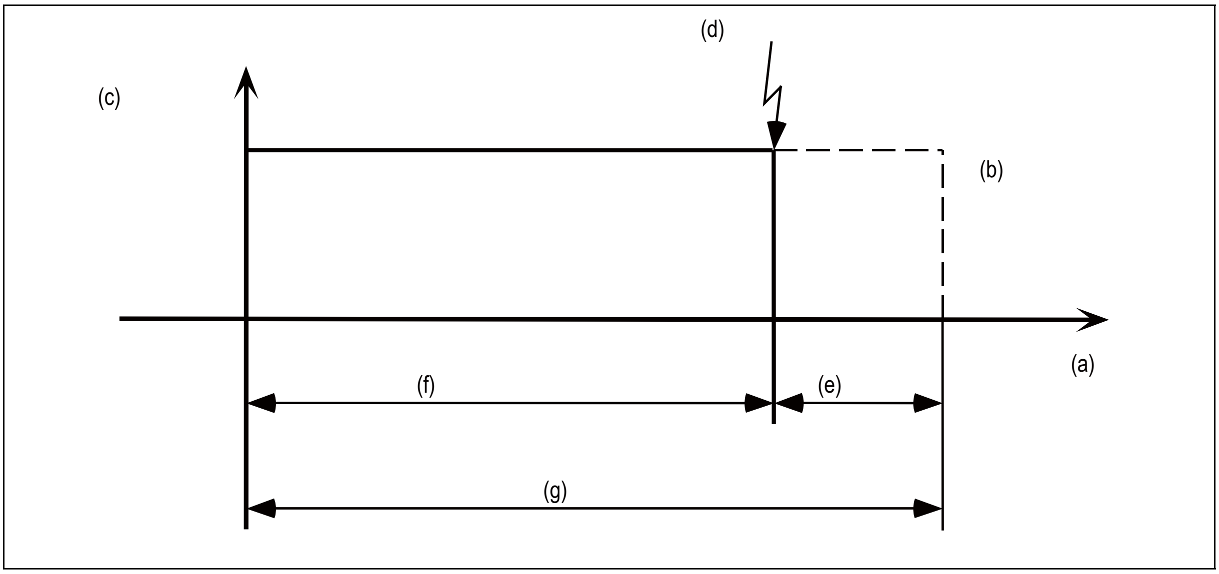
G31 Xx1 Yy1 Zz1 αα1 Rr1 Ff1 ; (α is the additional axis.)

G31	: Measurement command
Xx1,Yy1,Zz1,αα1	: Command values
Rr1	: Acceleration/deceleration command
Ff1	: Feed rate

[L system]

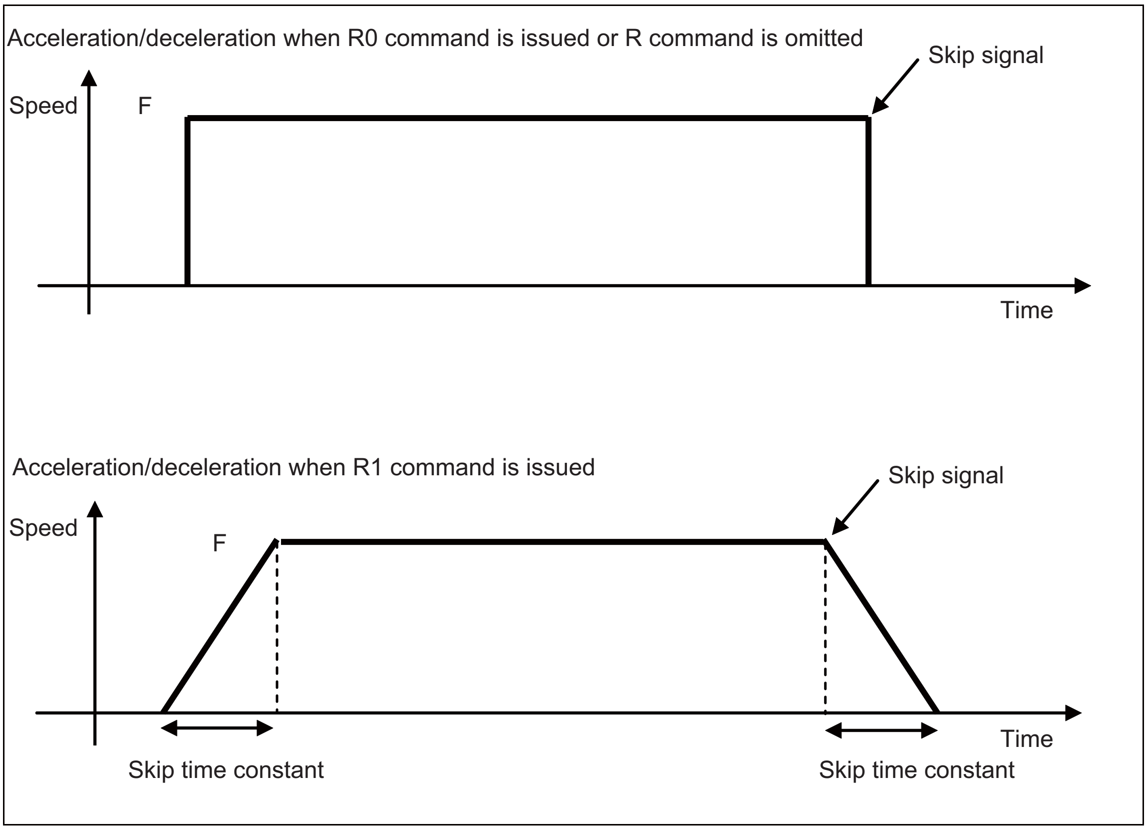
G31 Xx1/Uu1 Zz1/Ww1 Rr1 Ff1 ;

G31	: Measurement command
Xx1,Uu1,Zz1,Ww1	: Command values
Rr1	: Acceleration/deceleration command
Ff1	: Feed rate



(a) Position
(b) Programmed end point
(c) Feed rate
(d) Skip signal input
(e) Remaining distance
(f) Actual movement distance
(g) Command value

Command by G31 command basis for acceleration/deceleration command (R0/R1).
When R0 command is issued or R command is omitted, automatic acceleration/deceleration is not performed but step acceleration/deceleration is performed after interpolation in G31 block.
If R1 command is issued, automatic acceleration/deceleration is performed after interpolation even when skip signal is input.
A command is not issued or a command other than R0/R1 is issued, it will be the operation of acceleration/deceleration time constant=0(R0) and automatic acceleration/deceleration is not performed after interpolation.



Changeover of acceleration/deceleration type with R command

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.

14.1.1.2 Multiple-step Skip

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function realizes skipping by designating a combination of skip signals for each skip command.

(1) G31.n method

This function carries out skipping by designating a combination of skip signals for each skip command (G31.1, G31.2, G31.3).

The combination of the skip signals 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	: Axis address and target position
Ff1	: Feedrate (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 8. 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 during the dwell command (G04) and the next block executed under the skip conditions (to distinguish external skip signals 1 to 8) set with the parameters.

G31 Xx1 Yy1 Zz1 Pp Ff1 ;

G31	: Skip command
Xx1,Yy1,Zz1	: Axis address and target position
Pp	: Skip signal command
Ff1	: Feedrate (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 255. 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							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
:	:	:	:	:	:	:	:	:
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

14.1.1.4 PLC Skip

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function enables skip operations to be performed by signals which are input from the user PLC.

14.1.1.5 Speed Change Skip

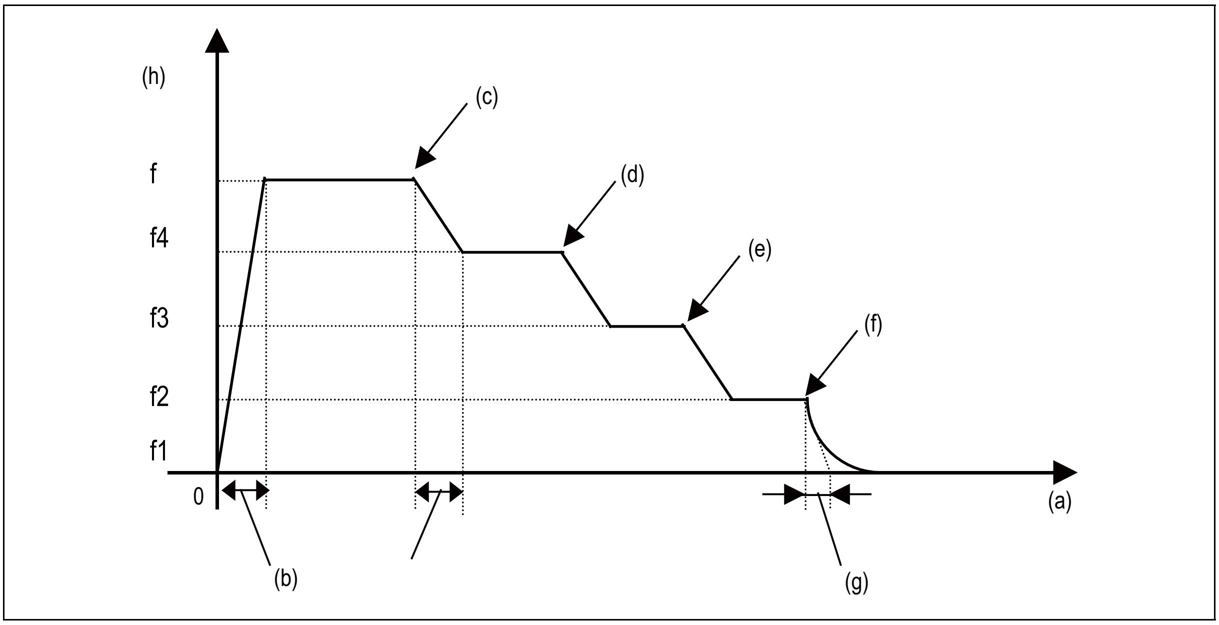
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function is used to change the feed rate or to stop the movement by inputting the skip signal during the linear interpolation.
After stopping the movement with skipping, the remaining movement command is canceled and the next block is executed.
In addition to the speed change skip option, the high-speed skip option and multiple-step skip option are required to use this function.

G31 Xx Yy Zz Ff F1 = f1 ... Fn = fn ; (n is the skip signal 1 to 8.)

G31	: Skip command
Xx, Yy, Zz	: Command position
Ff	: Feed rate when starting the cutting feed (mm/min)
fn	: Feed rate after detecting the skip signal (mm/min)
	fn = 0 : Movement stop
	fn ≠ 0 : Changing the feed rate to fn
	F1 = Feed rate after inputting the skip signal 1
	:
	F8 = Feed rate after inputting the skip signal 8

Operation example
G31 X100. Ff F1 = 0 F2 = f2 F3 = f3 F4 = f4 ;



(a) Time
(b) Skip time constant
(c) Skip signal 4
(d) Skip signal 3
(e) Skip signal 2
(f) Skip signal 1
(g) Position loop time constant (position loop gain^{-1})
(h) Speed

14.1.2 Automatic Tool Length Measurement

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function moves the tool in the direction of the tool measurement position by assigning a distance between the measurement start position to the measurement position. The machine stops when the tool reaches the sensor and automatically calculates the difference between the coordinate position where the tool stopped and measurement position. It registers this difference as the tool length compensation amount for that tool.

(1) Automatic Tool Length Measurement (M system)

This function moves the tool in the direction of the tool measurement position by commanding a distance 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 coordinate position when the tool has stopped and commanded coordinate position. It registers this difference as the tool length compensation amount for that tool.

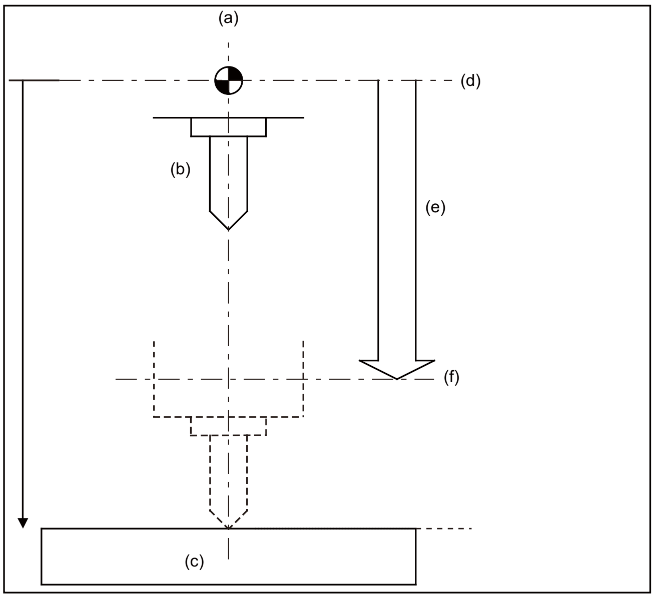
If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current compensation amount is further 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 Zz1 Rr1 Dd1 Ff1 ;

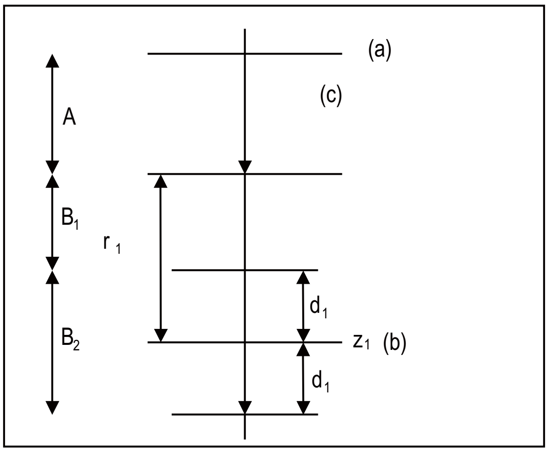
G37 ;	: Measurement command
Zz1	: Measurement axis address and measurement position X, Y, Z, α (α = optional axis)
Rr1	: The distance between the point at which tool movement is to start at the measurement speed and the measurement position
Dd1	: The range in which the tool is to stop
Ff1	: The measurement rate

When Rr1_, Dd1_ and Ff1_ have been omitted, the values set in the parameters are used.



At this time, the tool length compensation 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 compensation amount which is then registered in the tool compensation table.

(a) Tool change position
(b) Tool
(c) Sensor
(d) Reference position (In case of machine coordinate system zero point.)
(e) Amount of movement based on tool length measurement
(f) Tool length measurement position (Za1)



Area A : Moves with rapid traverse feed rate.
Areas B_1, B_2 : Moves with the measurement speed (f1 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.

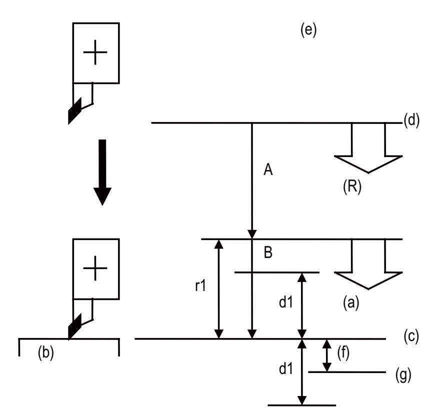
(a) Start point
(b) Measurement position
(c) r1, d1, and f1 can also be set in parameters.

(2) Automatic tool length measurement (L series)

This function moves the tool in the direction of the tool measurement position by commanding a distance 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 coordinate position when the tool has stopped and commanded coordinate position. It registers this difference as the tool length compensation 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 αα1 Rr1 Dd1 Ff1 ;	
G37	: Measurement command
αα1	: Measurement axis address and measurement position coordinate ... X,Z
Rr1	: The distance between the point at which tool movement is to start at the measurement speed and the measurement position (Radial value fixed / incremental position)
Dd1	: The range in which the tool is to stop (Radial value fixed: incremental position)
Ff1	: The measurement rate
When Rr1_, Dd1_ and Ff1_ have been omitted, the values set in the parameters are used.	



(a) F feed
(b) Measuring instrument
(c) Measurement position
(d) Start position
(e) r1, d1, and f1 can also be set in parameters.
(f) Compensation amount
(g) Sensor ON
(R) Rapid traverse feed

When the tool moves from the measurement 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 (sensor signal) turns ON during the tool is moving in the B area, an error occurs. If the measurement position arrival signal (sensor signal) does not turn ON although the tool passes through the measurement position x1 (z1) and moves by d1, an error occurs.

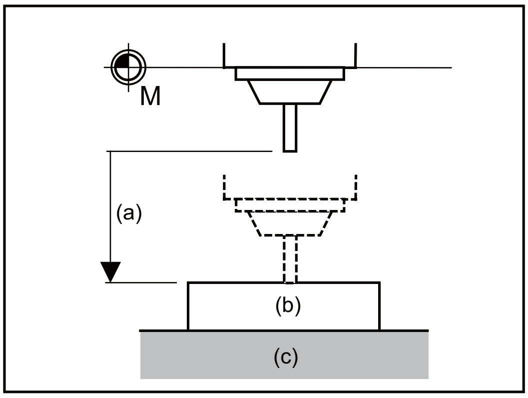
14.1.3 Manual Tool Length Measurement 1

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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

(1) Manual tool length measurement I [M system]

When the tool is at the reference position, this function enables the distance from the tool tip to the measurement position (top of workpiece) to be measured and registered as the tool length compensation amount.



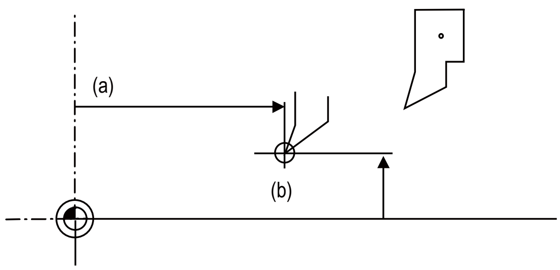
(a) Manual movement amount (tool length compensation amount)
(b) Workpiece
(c) Table

(2) Manual tool length measurement I [L system]

This is the function to calculate the tool length compensation amount automatically by moving the tool to the measurement point with manual feed. There are two types of measurement methods in manual tool length measurement I: the basic point method and the measurement value input method. The method is selected by setting parameter.

(a) Basic point method

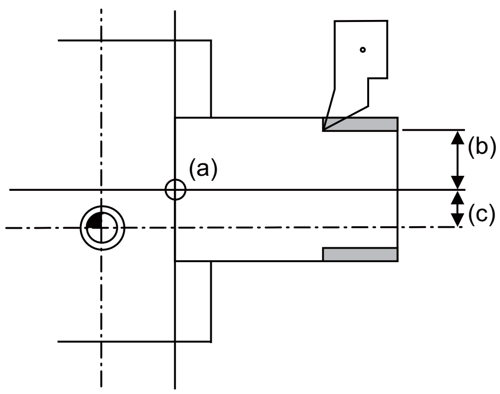
Obtain the tool length with the tool nose placed on the measurement point.
To carry out the basic point method, a point to place the tool nose on (measurement point) is required.
Set the measurement point in parameter beforehand.



(a) Z-axis + parameter value
(b) Measurement point

(b) Measurement value input method

Actually cut the workpiece. Measure its dimensions, and obtain the tool length from the measured values.
The measurement basic point is characteristic for each machine (the center of the chuck face, etc.)
To carry out the measurement value input method, a workpiece for measuring is required.
To measure the workpiece, set the basic point (measurement basic point) in parameter beforehand.



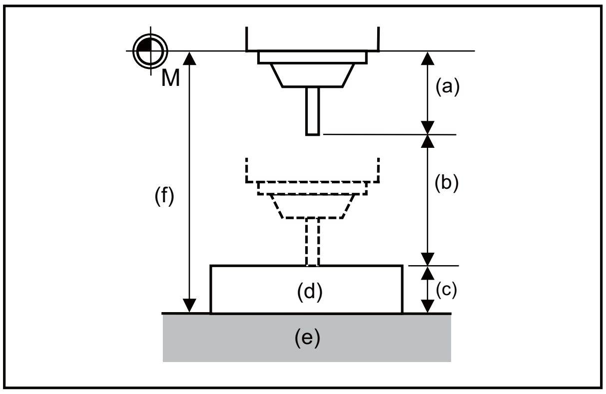
(a) Measurement basic point
(b) Measurement value (For X-axis)
(c) X-axis + parameter value

14.1.4 Manual Tool Length Measurement 2

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

(1) Manual tool length measurement II [M system]

When the tool is positioned at the reference position, this function enables the distance from the reference position to the tool tip to be measured and registered as the tool length compensation amount. In this case, the position of the gauge block used as a reference must be set as the basic height.

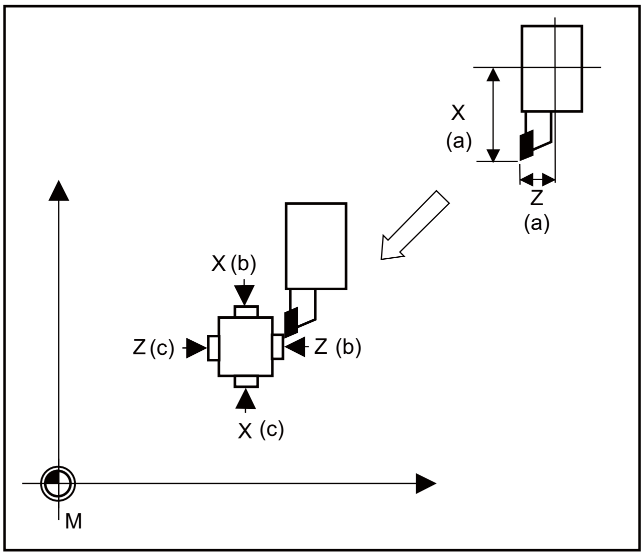


(a) Tool length compensation amount
(b) Manual movement amount
(c) Basic height
(d) Gauge block
(e) Table
(f) TLM basic length (setup parameter)

If the height axis designation parameter is ON, the axis designated for plane selection basic axis K is the axis targeted for measurement as the height axis.
Furthermore, if the tool length measurement check parameter is ON, an input OK/cancel confirmation message appears after input key has been pressed.

(2) Manual tool length measurement II [L system]

A device in which a touch sensor is built is used. Simply by causing the tool nose to touch the touch sensor in manual feed, the tool compensation amount can be calculated and stored in tool compensation amount memory.
Preset the machine coordinate position of the touch sensor touch face in parameter as the measurement basic position.



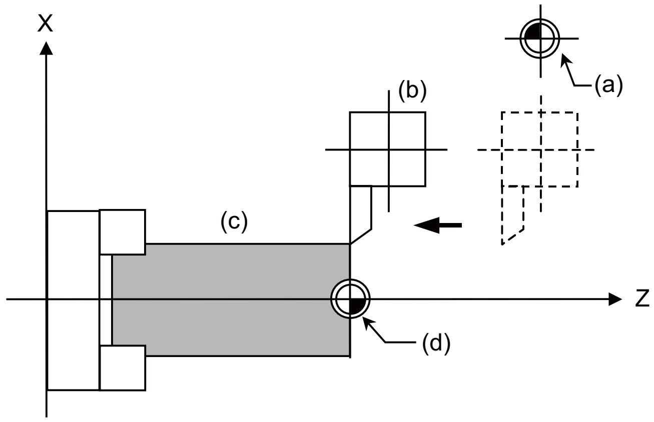
(a) Tool length
(b) - touch face
(c) + touch face

14.1.5 Workpiece Coordinate Offset Measurement

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

The external workpiece coordinate offset data for the Z axis can be set by cutting the workpiece face by means of manual operations and inputting the workpiece measurement signal.
By pressing the menu key, data can be set in the Z axis of an arbitrary coordinate offset.
Coordinate offset setting for the Z axis (2nd axis) of the part system 1 to 4 is possible.
Note that the workpiece coordinate offsets for multiple part systems cannot be measured at the same time.
(Note) Measurement is disabled for the part system with one-axis structure.

Example of measurement of external workpiece coordinate offset data for Z axis



(a) Machine zero point
(b) Tool post
(c) Workpiece
(d) Workpiece coordinate zero point

(1) Measurement method with workpiece measurement signal
  (a) Select the tool, and cut the workpiece face.
  (b) When the workpiece measurement signal is input, the external workpiece coordinate offset data for the Z axis is calculated from the machine coordinate position, length of the tool used and tool nose wear compensation amount, and stored in the memory.

(2) Measurement method with "MeasVal take in" menu
  (a) Select a tool, and cut the workpiece face.
  (b) When the "MeasVal take in" menu is pressed, workpiece coordinate offset data is calculated from the machine coordinate position, tool length and tool nose wear compensation amount of a used tool, and external workpiece coordinate offset. The workpiece coordinate offset data is stored in the Z-axis of the selected coordinate system offset.

14.1.6 Workpiece Position Measurement

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

The workpiece position measurement function is used to measure each axis' coordinate point by installing a sensor on the spindle and the sensor contacting the workpiece with the manual feed or handle feed.
The surface, hole center and width center coordinates are calculated from the measured coordinates, and those calculated results are set in the workpiece coordinate offset.
The axis to be measured is designated with parameter.
The workpiece position measurement is available for all the part systems.

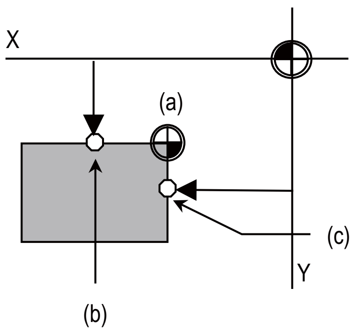
Here the measurement axes are explained as "X", "Y" and "Z".

(1) Surface workpiece offset measurement

The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.
Measurement position coordinate X = X axis' skip machine position + sensor diameter/2 ± center compensation amount (horizontal) ± skip flow amount (horizontal)
Measurement position coordinate Y = Y axis' skip machine position + sensor diameter/2 ± center compensation amount (vertical) ± skip flow amount (vertical)
Measurement position coordinate Z = Z axis' skip machine position - sensor length

The sensor diameter/2 and skip flow amount change between +/- with the tool movement direction during the measurement.
The X and Y axes use the sensor diameter, center compensation amount (horizontal/vertical) and skip flow amount (horizontal/vertical).
The Z axis uses the sensor length.

The measurement position coordinate of the X axis, Y axis or Z axis is set in the specified workpiece coordinate offset.



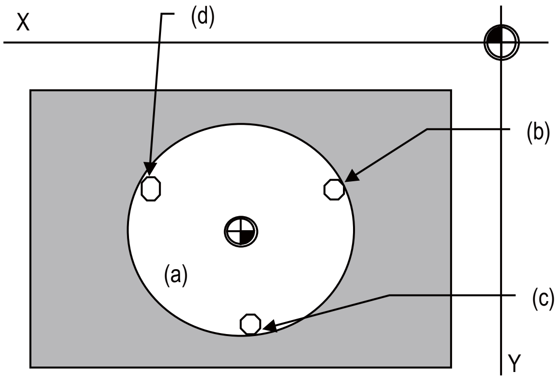
To set the workpiece coordinate offset, the X axis is measured and the X axis' offset coordinate is set. Then, the Y axis' offset is measured and set. Finally, the Z axis' offset is measured and set.

(a) Workpiece coordinate zero point
(b) Y axis measurement constant position
(c) X axis measurement constant position

(2) Hole center workpiece offset measurement

The measurement position coordinates of two axes (X, Y) are measured at three points, and the hole center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X and Y axes.
Measurement position coordinate X = X axis' skip machine position + center compensation amount (horizontal) ± skip flow amount (horizontal
Measurement position coordinate Y = Y axis' skip machine position + center compensation amount (vertical) ± skip flow amount (vertical)

The +/- of the skip flow amount changes according to the tool movement amount during measurement.



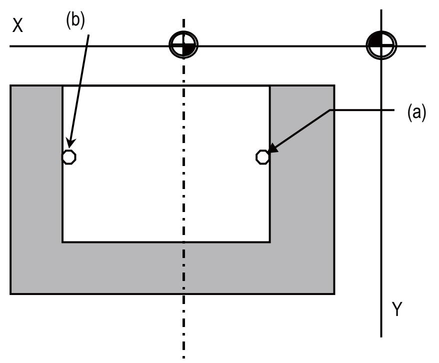
To set the workpiece coordinate offset, the position X and Y of the measurement A point are measured, and the measured values are set in the measurement A point. In the same manner as the measurement A point, the measurement B point and then C point are measured and set. The hole center coordinate is calculated by setting the workpiece coordinate system after setting three points, and the calculated result is set in the workpiece coordinate offset.

(a) Workpiece coordinate zero point
(b) Measurement A point
(c) Measurement B point
(d) Measurement C point

(3) Width center workpiece offset measurement

The two points among the measurement position coordinate of the X axis, Y axis or Z axis are measured, and each axis' groove center is calculated. The calculated result is set in the specified workpiece coordinate offset.
The workpiece position measurement coordinates are calculated from the skip machine position of the X, Y and Z axes.
Measurement position coordinate X = X axis' skip machine position + center compensation amount (horizontal) ± skip flow amount (horizontal)
Measurement position coordinate Y = Y axis' skip machine position + center compensation amount (vertical) ± skip flow amount (vertical)
Measurement position coordinate Z = Z axis' skip machine position - Sensor length

The +/- of the skip flow amount changes according to the tool movement amount during measurement



To set the workpiece coordinate offset, the position X (position Y, position Z) of the measurement A point is measured, and the measured value is set in the measurement A point. In the same manner as the measurement A point, the measurement B point is measured and set.
The groove width center coordinate of the X axis (Y axis, Z axis) is calculated by setting the workpiece coordinate system after setting two points, and the calculated result is set in the workpiece coordinate offset

(a) Measurement A point
(b) Measurement B point

14.1.7 Rotation Measurement

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

The offset (rotation center and rotation angle) of the rotary coordinate system is measured, and the results are set to the workpiece coordinate system offset (rotation center) and the parameters.

<Measurement using touch sensor>
Measurement counter X = X axis skip position (Machine position)
Measurement counter Y = Y axis skip position (Machine position)

<Simple measurement (measurement without using touch sensor)>
Measurement counter X = X axis machine position + center compensation (H) + skip past amount (Horizontal axis) (Note)
Measurement counter Y = Y axis machine position + center compensation (V) + skip past amount (Vertical axis) (Note)

(Note) The skip past amount is added for only the axis that moved last.
The sign (+ or -) of the skip past amount depends on the movement direction of the axis.

14.2 Tool Life Management

14.2.1 Tool Life Management

14.2.1.1 Tool Life Management I

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

(1) M system

For the user PLC designated tool, that tool's usage time (0 to 4000 hours) or frequency of use (0 to 65000 times) is accumulated, and the tool usage state is monitored.The life of up to 1000 tools can be managed. The maximum number of tools to be registered varies depending on options.

(2) L system

Tool life management is performed using the time and frequency of use of a tool.
The life for up to 80 tools (tool numbers 1 to 80) can be managed.

(a) Management by the time of use
The cutting time after specification of a tool selection (T) command (G01, G02, and G33) is added to the tool use time for the specified tool.
If the use time reaches the life time when a tool selection command is specified, an alarm is given.

(b) Management by the frequency of use
The tool use counter corresponding to the specified tool No. is incremented each time a tool selection (T) command is specified for the tool.
If the counter reaches the limit number when a tool selection command is specified, an alarm is given.

14.2.1.2 Tool Life Management II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

(1) 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.

- Number of groups : 1000 sets for all part systems. (For 1 part system : 999 sets)

(Round off the last digit of the compensation sets for the independent and multi-part system so that they will be multiples of 10. For example, when the maximum compensation set is 40 and the number of part systems is 3, the compensation sets par part system is 10.)

- Number of tools in group : Max. 1000 tools

(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.

- Number of groups: Max. 40 sets (each part system)/ For 1 part system: 80 sets
- Number of tools in group: Max. 16 tools

14.2.1.3 Tool life management III

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

For the user PLC designated tool, that tool's usage time (0 to 4000 hours) or frequency of use (0 to 65000 times) is accumulated, and the tool usage state is monitored. The life for up to 1000 tools can be managed.

This function is not controlled by the group No.

14.2.2 Number of Tool Life Management Sets

The number of tools that can be managed for their lives are shown below.

[M system]

Function name	Common for part systems or 1-part system	Independent for systems and multi-part system
Number of tool life management tools (200 sets)	200 sets	100 sets/part system
Number of tool life management tools (400 sets)	400 sets	200 sets/part system
Number of tool life management tools (600 sets)	600 sets	300 sets/part system
Number of tool life management tools (800 sets)	800 sets	400 sets/part system
Number of tool life management tools (1000 sets)	1000 sets	500 sets/part system

[L system]

Function name	Common for part systems or 1-part system	Independent for systems and multi-part system
Number of tool life management tools (80 sets)	80 sets	40 sets/part system

14.2.2.1 80 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

14.2.2.2 200 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

14.2.2.3 400 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

14.2.2.4 600 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

14.2.2.5 800 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

14.2.2.6 1000 Sets

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

14.3 Others

14.3.1 Programmable Current Limitation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function allows the current limit value of the NC 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 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 ;	
G10	: Current limit input command
L14	: Current limit value setting (+ side/- side)
X	: Axis address
dn	: Current limit value 1% to 999%

(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 999%. Commands that exceed this range will cause a program error.
(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".

14.3.2 Auto Power OFF

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Auto power OFF function notifies that the control unit's power can be turned OFF after shutting the display unit down by entering "automatic power OFF request" signal from user PLC to NC.

15

Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

All operations are stopped by the emergency stop signal input and, at the same time, the drive section is stopped 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

With the input from the user PLC, it is possible to prohibit the parameter setting or deletion, and the program edit on 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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 alarm	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 on the screen with the register at the time when the error occurred if the system stops due to a system error.
Absolute position detection system alarm	An alarm in the absolute position detection system is displayed.
Auxiliary axis alarm	The auxiliary axis alarm is displayed.
Computer link error	The computer link alarm is displayed.
User PLC alarm	The user PLC alarm is displayed.
Program error	This alarm occur during automatic operation, and the cause of this alarm is mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.
Network service error	The network service alarm is displayed.

15.2.3 Operation Stop Cause

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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

15.2.4 Emergency Stop Cause

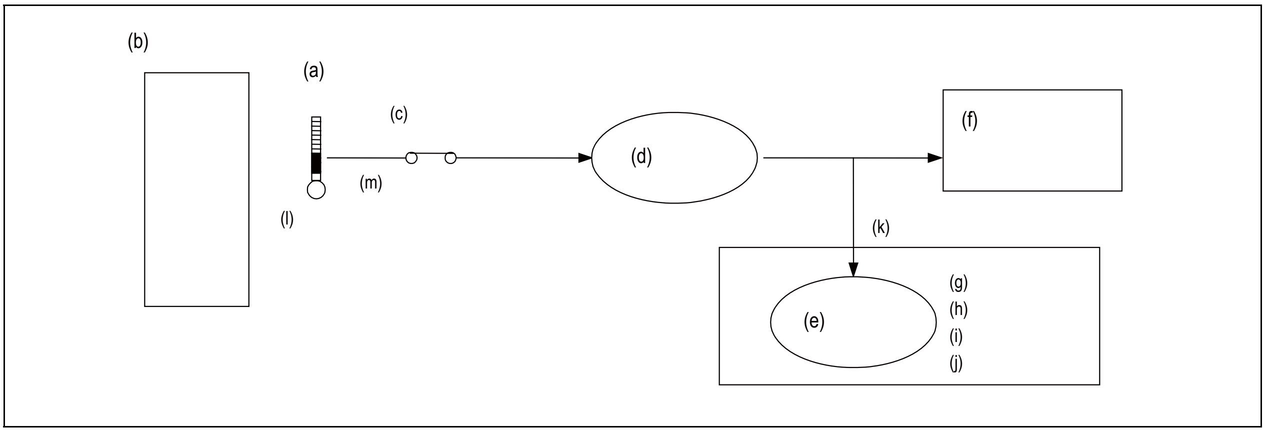
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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 Thermal Detection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	O	O	O	O	O	O
L system	O	O	O	O	O	O

When overheating is detected in the control unit, an overheat signal is output at the same time as the alarm is displayed. If the system is in automatic run at the time, run is continued, but it cannot be started after reset or completion by M02/M30. (It can be started after block stop or feed hold.)
When the temperature falls below the specified temperature, the alarm is released and the overheat signal is turned OFF.
The overheat alarm occurs at 80 C or more in the control unit.



(a) Overheat detection
(b) Control unit
(c) Parameter
(d) Temperature alarm
(e) User PLC
(f) Message display
(g) Cooling fan rotation
(h) Lamp alarm
(i) Emergency stop
(j) Others
(k) Bit device
(l) (a temperature of 80 C)
(m) (Default: valid)

(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.

15.2.6 Battery Alarm/Warning

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When it is time for changing batteries, alarm or warning is displayed.
When a warning is displayed, immediately backup all the necessary data and change batteries.
When an alarm is displayed, there is a possibility that memory has been lost.

15.3 Protection

15.3.1 Stroke End (Over Travel)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When limit switches and dogs have been attached to the machine and a limit switch has kicked a dog, the movement of the machine is stopped by the signal input from the limit switch.
At the same time, the alarm output is sent to the machine.
The stroke end state is maintained and the alarm state is released by feeding the machine in the reverse direction in the manual mode to disengage the dog.

15.3.2 Stored Stroke Limit

The areas where tool entry is prohibited can be set.
The stored stroke limits I, II, IIB, IB and IC are handled as follows.

<table>
<tr><th>Type</th><th>Prohibited range</th><th>Explanation</th></tr>
<tr><td>Ⅰ</td><td>Outside</td><td>- Set by the machine tool builder.
- When used with II, the narrow range designated by the two types becomes the movement valid range.</td></tr>
<tr><td>Ⅱ</td><td>Outside</td><td rowspan="2">- Set by the user.
- By using the program command, the parameter can be changed and the function can be switched to ON/OFF.
- Select II or IIB with the parameters.</td></tr>
<tr><td>Ⅱ B</td><td>Inside</td></tr>
<tr><td>Ⅰ B</td><td>Inside</td><td>- Set by the machine tool builder.</td></tr>
<tr><td>Ⅰ C</td><td>Outside</td><td>- Set by the machine tool builder.</td></tr>
</table>

15.3.2.1 Stored Stroke Limit I/II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(1) Stored Stroke Limit I

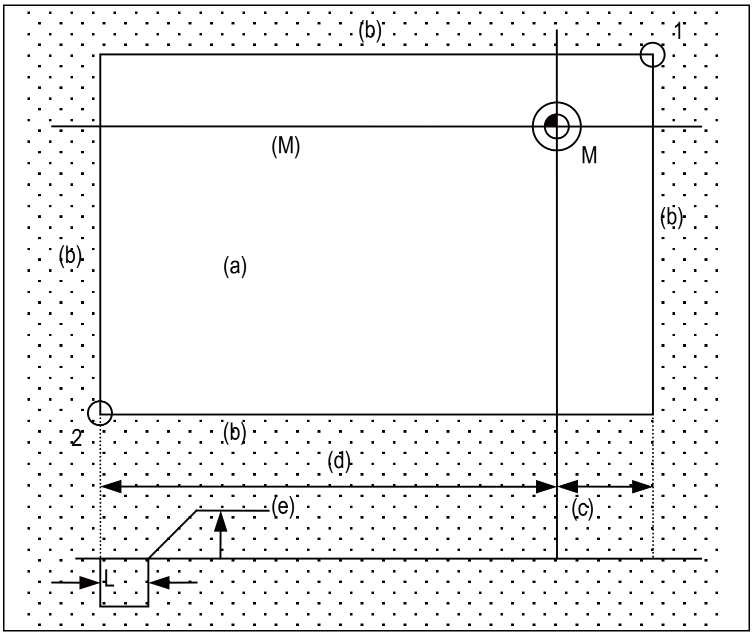
This is the stroke limit function used by the machine tool builder, and the area outside the set limits is the entrance prohibited area.

The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit II function described in the following section, and the tolerable area of both functions is the movement valid range.

The setting range is -99999.999 to +99999.999mm.

The stored stroke limit I function is made valid not immediately after the controller power is turned ON but after reference position return.

The stored stroke limit I function will be invalidated if the maximum and minimum values are set to the same data.



The position of points 1 and 2 are set using the machine coordinate. : Prohibited area

(a) Machine movement valid range
(b) Prohibited area
(c) "+" setting
(d) "-" setting
(e) Feedrate
(M) Machine coordinate system
1: Point 1
2: Point 2

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.

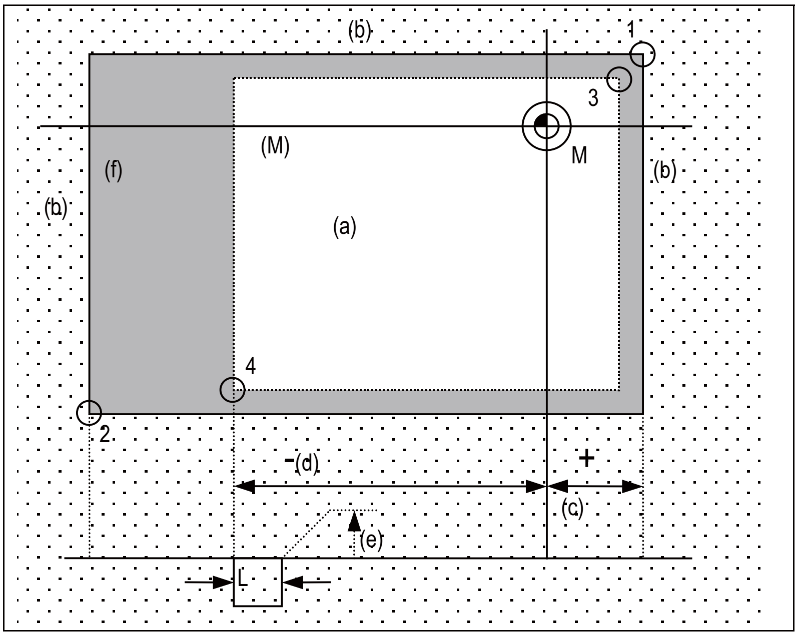
(2) Stored Stroke Limit II

This is the stroke limit function which can be set by the user, and the area outside the set limits is the prohibited area.

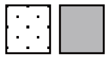
The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit I function described in the foregoing section, and the tolerable area of both functions is the movement valid range.

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 position of points 3 and 4 are set with the machine coordinate.
The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

 :

Prohibited area

(a) Machine movement valid range
(b) Prohibited area
(c) "+" setting
(d) "-" setting
(e) Feed rate
(f) Area prohibited by stored stroke limit function II
(M) Machine coordinate system
1: Point 1
2: Point 2
3: Point 3
4: Point 4

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.

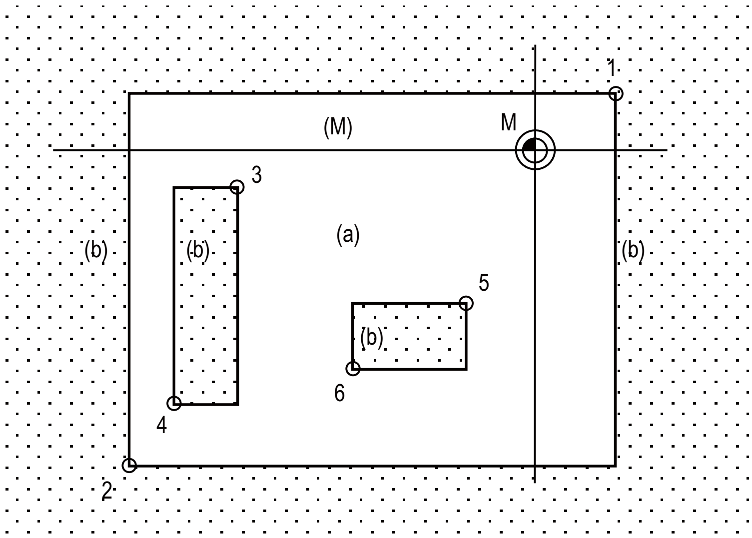
The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

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

15.3.2.2 Stored Stroke Limit IB

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Three areas where tool entry is prohibited can be set using the stored stroke limit I, stored stroke limit II, IIB and stored stroke limit IB functions.



The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.
The area determined by points 3 and 4 is the prohibited area set with stored stroke limit IIB.
The area determined by points 5 and 6 is the prohibited area set with stored stroke limit IB.

(a) Machine movement valid range
(b) Prohibited area
(M) Machine coordinate system
1: Point 1
2: Point 2
3: Point 3
4: Point 4
5: Point 5
6: Point 6

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.

(Note 1) Bear in mind that the following will occur if the same data is set for the maximum and minimum value of the tool entry prohibited area:

(1) When zero has been set for the maximum and minimum values, tool entry will be prohibited in the whole area.

(2) If a value other than zero has been set for both the maximum and minimum values, it will be possible for the tool to move in the whole area.

15.3.2.3 Stored Stroke Limit IIB

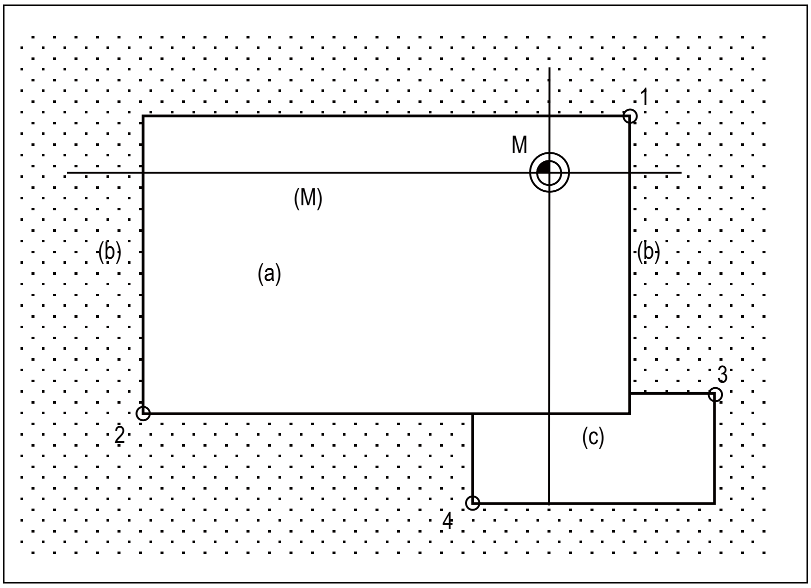
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

A parameter is used to switch between this function and stored stroke limit II. With stored stroke limit IIB, the range inside the boundaries which have been set serves as the tool entry prohibited area.

15.3.2.4 Stored Stroke Limit IC

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The boundary is set for each axis with the parameters. The inside of the set boundary is the additional movement range.
This cannot be used with stored stroke limit IB.



The position of points 3 and 4 are set with the machine coordinate.
The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I. : Prohibited area

(a) Machine movement valid range
(b) Prohibited area
(c) Additional movement range
(M) Machine coordinate system
1: Point 1
2: Point 2
3: Point 3

15.3.3 Stroke Check before Travel

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

By commanding, from the program, the boundary for prohibiting machine entry as a coordinate position on the machine coordinate system, entry into the inner side of that boundary can be prohibited. Only the three basic axes can be set.
As compared to the normal stored stroke limit with which the movement stops just before the set prohibited range, with this function, if a command exceeding the valid movement range is issued, a program error will occur before moving in that block.

G code	Function
G22	Stroke check before movement function ON
G23	Stroke check before movement function OFF

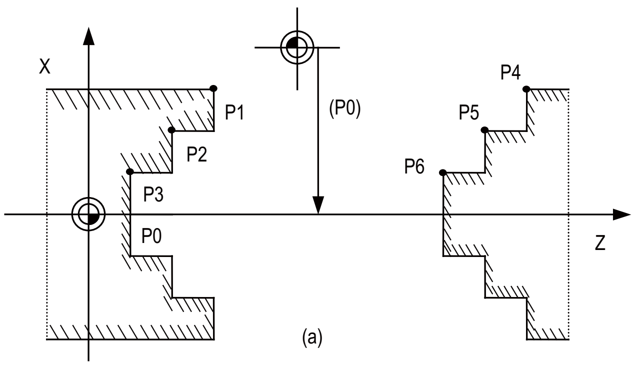
15.3.4 Chuck/Tailstock Barrier Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	○	○	○	○	○	○

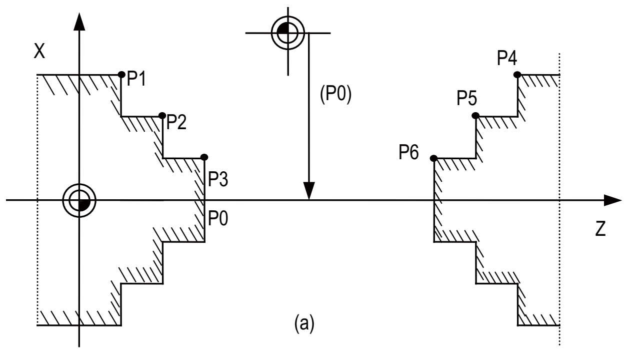
By limiting the tool nose point move range, this function prevents the tool from colliding with the chuck or tailstock because of a programming error. When a move command exceeding the area set in a given parameter is programmed, the tool is stopped at the barrier boundaries.
Program format

G22 ; ..... Barrier ON
G23 ; ..... Barrier OFF (cancel)

(1) When the machine is about to exceed the area, the machine is stopped and an alarm is displayed. To cancel the alarm, execute reset.
(2) The function is also effective when the machine is locked.
(3) This function is valid when all axes for which a barrier has been set have completed reference position return.
(4) The chuck barrier/tailstock barrier can be set per each part system.
(5) Chuck barrier/tailstock barrier setting.



(a) (Form 1)



(a) (Form 2)

The chuck barrier and tailstock 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 tailstock barrier. The X axis is set with the coordinate position (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 tailstock 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

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The machine movement will decelerate and stop as soon as the interlock signal, serving as the external input, is turned ON.
When the interlock signal is turned OFF, the machine starts moving again.
(1) In the manual mode, only that axis for which the interlock signal is input will stop.
(2) In the automatic mode, all axes will stop when the interlock signal is input to even one axis which coincides with the moving axis.
(3) Block start interlock
While the block start interlock signal (*BSL) is OFF (valid), the execution of the next block during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.
(Note 1) This signal is valid for all blocks including internal operation blocks such as fixed cycles.
(Note 2) This signal (*BSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.
(4) Cutting start interlock
While the cutting start interlock signal (*CSL) is OFF (valid), the execution of all movement command blocks except positioning during automatic operation will not be started. The block whose execution has already commenced is executed until its end. Automatic operation is not suspended. The commands in the next block are placed on standby, and their execution is started as soon as the signal is turned ON.
(Note 1) The signal is valid for all blocks including internal operation block such as fixed cycles.
(Note 2) This signal (*CSL) is set ON (invalid) when the power is turned ON. If it is not used, there is no need to make a program with the PLC.

15.3.6 External Deceleration

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function reduces the feed rate to the deceleration speed set by the parameter when the external deceleration input signal, which is the external input from the user PLC, has been set to ON. External deceleration input signals are provided for each axis and for each movement direction ("+" and "-"), and a signal is valid when the signal in the direction coinciding with the direction of the current movement has been input. When an axis is to be returned in the opposite direction, its speed is returned immediately to the regular speed assigned by the command.
When non-interpolation positioning is performed during manual operation or automatic operation, only the axis for which the signal that coincides with the direction of the current movement has been input will decelerate.
However, with interpolation during automatic operation, the feed rate of the axis will be reduced to the deceleration rate if there is even one axis for which the signal that coincides with the direction of current movement has been input.

The external deceleration input signal can be canceled using a parameter for the cutting feed only.

15.3.8 3D Machine Interference Check

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	☆	☆	—	—	—
L system	—	—	—	—	—	—

Under development.

15.3.9 Door Interlock

15.3.9.1 Door Interlock I

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(Outline of function)
Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.
When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo drive unit 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.
(a) 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.
(b) 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 NC 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.

15.3.9.2 Door Interlock II

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Outline of function

Under the CE marking scheme of the European safety standards (machine directive), the opening of any protection doors while a machine is actually moving is prohibited.

When the door open signal is input from the PLC, this function first decelerates and stops all the control axes, establishes the ready OFF status, and then shuts off the drive power inside the servo amplifiers so that the motors are no longer driven.

With the door interlock function established by the door open II signal, automatic start can be enabled even when the door open signal has been input. However, the axes will be set to the interlock status.

Description of operation

When a door is open

The NC system operates as follows when the door open II signal is input:

(1) It stops operations.
All the axes decelerate and stop.
The spindle also stops.

(2) The complete standby status is established.

(3) After all the NC 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.

15.3.10 Parameter Lock

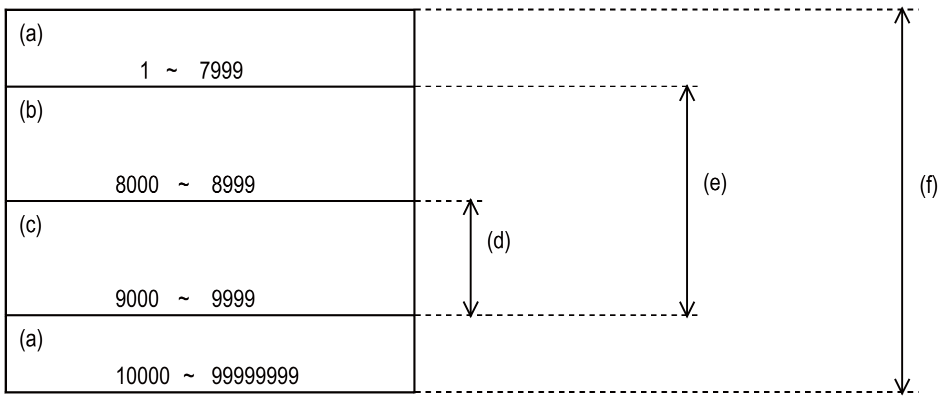
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function is used to prohibit changing the machine parameter.

15.3.11 Program Protection (Edit Lock B, C)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L 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.



(a) Machining program A
(b) Machining program B (User-prepared standard subprogram)
(c) Machining program C (Machine maker customized program)
(d) Editing is inhibited by edit lock C.
(e) Editing is inhibited by edit lock B.
(f) Editing is inhibited by data protect.

15.3.12 Program Display Lock

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function allows the display of only a target program (label address 9000) to be invalidated for the program display in the monitor screen, etc.
The operation search of a target program can also be invalidated.
The validity of the display is selected with the parameters. The setting will be handled as follows according to the value.

0: Display and search are possible.
1: Display of the program details is prohibited.
2: Display and operation search of the program details are prohibited.

15.3.13 Safety Observation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

With the safety observation function, safety access to the working part of the machine without shutting the power is possible, which means that adjustment and preparation can be done during the machine working, and so the period of time until restarting the machine can be shortened.
The safety of this function is based on Machinery safety EN954-1:1996 of European Safety Standards.
The safety functions for the access to the working part are consolidated in Safety Observing Unit to provide the higher safety of the machine.

Function	Description
Speed observation function	By observing the axis speed doubly, the safety door of the machine can be opened so that setup operations, etc. can be done without shutting the power. Safety Observing Unit observes the speed set for each axis, and if the speed exceeds the setting speed, the unit commands an emergency stop immediately. Special external detector is not necessary.
Input observation function	Safety Observing Unit checks the consistency of two conditions. If the consistency is not verified, it is output by two paths. The observation result is output to outside by single path. This is for the double observation of the signals for safety, operation preparation, etc.
Emergency stop function	When Emergency stop is input or an error occurs, shutdown output will be executed doubly. This is based on the safety circuit structure. Emergency stop cannot be canceled unless the safety of shutdown parts (contactor, etc.) is assured. In emergency, shutdown output is performed immediately. If that does not stop the NC, the power supply will be shutdown.

15.3.14 Vertical Axis Pull-up

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When emergency stop or power interruption occurs, motor brake and vertical axis drop prevention function is supposed to work to prevent vertical axis (such as Z axis of vertical machining center) from dropping.
However, there might be a case that the vertical axis drops by several μm because of decrepitude of motor brake.
Therefore, emergency stop or the power interruption during finish machining at low speed might cause damage to the cutting tool in operation.
Vertical axis pull-up function prevents the tool from breakage, through pulling up the cutting tool during emergency stop or instantaneous power interruption at low cutting speed.

15.4 Maintenance and Troubleshooting

15.4.1 Operation History

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This is a maintenance function which is useful for tracing down the history and NC operation information and analyzing trouble, etc. This information is saved in the history data file, and can be displayed on the screen and output to a file.
The machine tool builder password is required to use his function.

The following two types of history data files are used.
- All history: The key history, alarm/warning history, PLC signal history, and AC input power error history is recorded in date/time order.
- Key history: Only the key history is recorded in date/time order.

(Note)If the date and time are the same, the history will be displayed in the following order for each part system.
(1) Alarm/warning history
(2) PLC input/output signal history
(3) Key history
(4) AC input power error history

15.4.2 Data Sampling

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The NC data sampling function samples the NC internal data (speed output from NC to drive unit, and feedback data from the drive unit, etc.). This data can be output as text data.

Item	Specifications
Sampling cycle	1.7ms × setting value
Number of sampled axes	NC axis : 1 to 16axes Spindle : 1 to 4 axes
Number of sampled channels	1 to 8 points
Sampling data size	Maximum 1,310,720 points (Note 1) This is the entire data size. The data size per channel will decrease when the number of sampled channels increases. (Note 2) If the open DRAM memory is insufficient, the maximum data size will decrease.

- The data set with this function is not output to the parameters.
- The state returns to "Sampling stop" when the power is turned ON.

15.4.3 NC Data Backup

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The NC data back up function backs up the NC parameters, etc., on an HD or IC card.
The data can also be restored.

15.4.4 MELDASNET

15.4.4.1 Machine Tool Builder Network System

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

With the machine builder network system, using the software for personal computer "MELDAS remote diagnosis tool kit" which Mitsubishi Electric Corporation sells separately to machine builders, NC is connected with the personal computer of the machine builder by the communication circuit, and the diagnosis data file required for the service is transmitted and received.

15.4.4.2 Anshin-net Service

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

By means of a network that connects the CNC systems of end users all over Japan with the Ryoden Kouki Engineering (RKE) Call Center, detailed information services and speedier maintenance support can be provided.
A separate agreement must be signed with RKE in order to receive this service.
(Note) This service is applied only in Japan.

Service menu

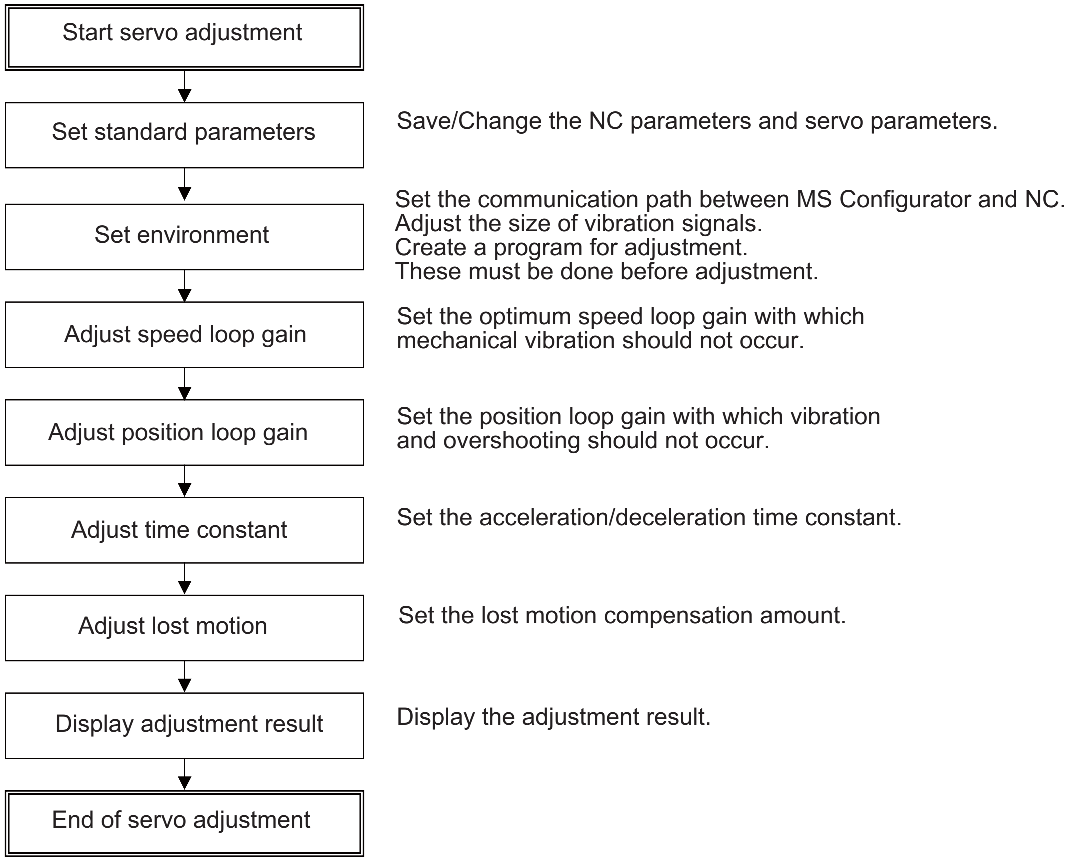
(1) Peace-of-mind menu		
Automatic notification of alarms	Standard	The Call Center is aware of the circumstances under which alarms have occurred during automatic operation. Speedy support can be provided even during unmanned operation.
Single-action call	Standard	Simply by pressing the single-action button, the machine status information can be transmitted so that technical support can be received from expert engineers.
Periodical monitoring	Standard	Malfunctioning is prevented by monitoring and analyzing the warning messages on a periodical basis.
Backup	Standard	Speedy action is taken to deal with even the loss of data in the memory by backing up the parameter data on a periodical basis.
(2) Handy menu		
Operator notification	Standard	The operator is automatically notified of the completion of machining, error occurrence such as operation stop, or other trouble by voice messages to the operator's cell phone, etc.
Periodical monitoring report	Standard	Transmission of written reports complied by totalizing the monitored data comes in handy for managing and formulating production plans.
Remote monitoring	Optional	The operating conditions of the machine can be ascertained at any time by simply accessing the Call Center from the user's telephone.
Machining data sharing	Optional	The Call Center's servers are released to users who can then utilize them as media for storing programs.

15.4.5 Servo Automatic Tuning

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

With this function, the servo parameters can be automatically adjusted by connecting CNC and MS Configurator, which is an application that runs on a regular PC.
MS Configurator measures and analyzes the machine characteristics to automatically adjust the servo parameters while having the motor run by test NC programs or vibration signals.
(Note) The separate software is required.

The servo is adjusted with the MS Configurator according to the following flow.



MS Configurator supports the following servo parameter automatic adjustment function and data measurement function.

<Waveform measurement function>

Frequency response measurement	:	Measures the frequency response (speed command - speed FB) of speed loop for the designated axis. The result will be presented as Bode diagram.
Frequency response measurement of machine	:	Measures the frequency response (torque command - speed FB) of machine system for the designated axis. The result will be presented as Bode diagram.
Measurement function (with program creation function)	:	Measures the Chronological data measurement, Arc shape error measurement, Synchronous tapping error measurement, Measuring arbitrary path.

<Automatic adjustment function>

Initial notch filter setup	:	Automatically adjusts the notch filter when the initial resonance is large.
Velocity loop gain adjustment	:	Automatically adjusts the notch filter and the speed loop gain.
Time constant adjustment	:	Automatically adjusts the acceleration/deceleration time constant.
Position loop gain adjustment	:	Automatically adjusts the position loop gain.
Lostmotion adjustment	:	Automatically adjusts the quadrant protrusion amount of the designated axis.
Lostmotion 3 adjustment	:	Automatically adjusts the lost motion type 3 for the quadrant protrusion amount of the designated axis.

<Environment setup>

Communication path setup	:	Sets the path to communicate with NC. The model of connected NC is selected.
Program creation	:	Creates machining programs for adjustment.

<Assistance setup function>

Parameter setup	:	Saves/changes the servo parameters.

15.4.6 Automatic Backup

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

With this function, system data, ladder program and custom software can be automatically backed up in case of system failure. Up to the three most recent backup data can be stored.
The backup execution timing can be set with parameters.

15.4.7 System Setup

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

System setup function enables automatic settings for the NC's initial startup just by inputting the minimally required items.
With this function, settings required for preliminary operation of the motors (servo motor, spindle motor and auxiliary axis motor) can be provided easily, resulting in shortening of the time required for the NC initial startup.

The required setting items for this function are as follows.

(1) Number of spindle connections, number of auxiliary axis connections
(2) Number of axis and command type in each part system
(3) Setting of the servo interface connection channel and rotary switch for each axis drive. Also, converter /motor type connected with each spindle drive.
(4) Setting of the servo interface connection channel and rotary switch for each NC axis. Also, the motor type, encoder type, and the converter type that is connected with the servo drive.

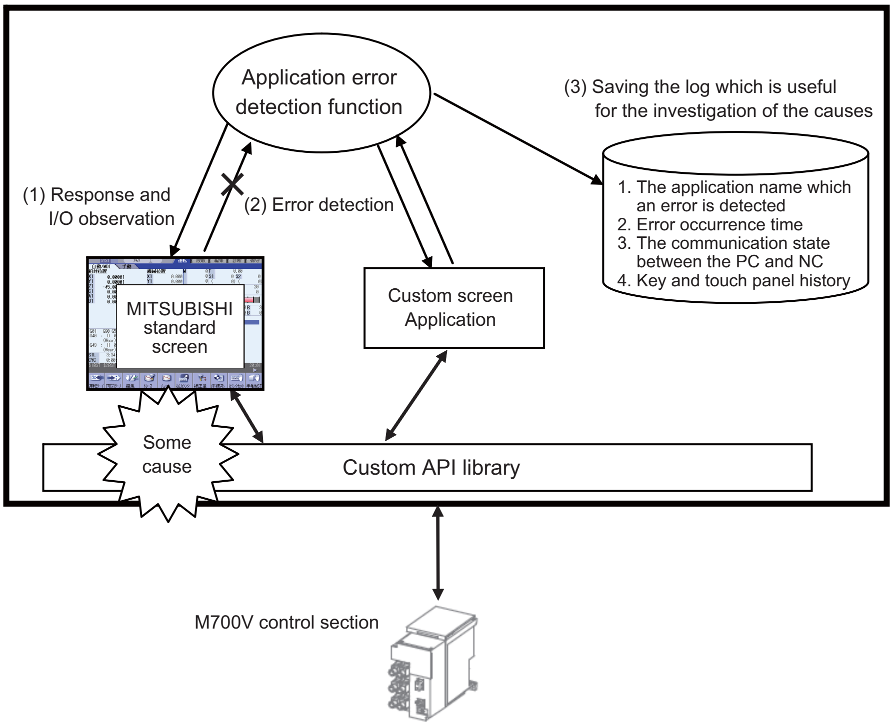
By setting above items, the following setting required for the NC's initial startup can be executed.

(1) SRAM clear and file format
(2) Various parameter settings (machine parameter and user parameter)
Note that there are two methods of setting spindle parameters: using spindle parameter files provided by Mitsubishi, and setting in system setup screen.
(3) PLC ladder writing settings required for the manual operation (JOG mode only)
Note that a menu has to be selected again.

15.4.11 Application Error Detection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	—	—	—
L system	○	○	○	—	—	—

Application error detection function observes applications such as MITSUBISHI standard screen or custom screen. When an error such as screen lock is detected, this function saves information and data in the log to investigate the causes easily.



(1) Response from the application and a change of I/O is observed on a periodical basis.
(2) When an application with no response and no change of I/O for a certain period is detected, it is determined that there is an error.
(3) When an error is detected, the log which is useful for the investigation of the causes will be saved.

16

Drive System

16.1 Servo/Spindle

Refer to the following manuals for details on the servo and spindle system.
DRIVE SYSTEM DATA BOOK (IB-1500273(ENG))
MDS-D/DH Series Specifications Manual (IB-1500875(ENG))
MDS-D-SVJ3/SPJ3 Series Specifications Manual (IB-1500158(ENG))
MDS-DM Series Specifications Manual (IB-1500891(ENG))

16.1.1 Feed Axis

16.1.1.1 MDS-D-V1/D-V2 (200V)

16.1.1.1.1 Servo motor: HF**-A48 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.1.2 Servo motor: HF**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.1.3 Servo motor: HF**-A74 (16000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.1.4 Servo motor: HP**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.1.5 Servo motor: HP**-A74 (16000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.1.6 Servo motor: HF-KP**JW04 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.2 MDS-DH-V1/DH-V2 (400V)

16.1.1.2.1 Servo motor: HF**-A48 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.2.2 Servo motor: HF-H**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.2.3 Servo motor: HF-H**-A74 (16000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.2.4 Servo motor: HP-H**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.2.5 Servo motor: HP-H**-A74 (16000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.3 MDS-D-SVJ3 (200V)

16.1.1.3.1 Servo motor: HF**-A48 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.3.2 Servo motor: HF**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.3.3 Servo motor: HF-KP**JW04 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.4 MDS-DM-V3/SPV2/SPV3 (200V)

16.1.1.4.1 Servo motor: HF**-A48 (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.4.2 Servo motor: HF**-A51 (1000kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.1.4.3 Servo motor: HF-KP**J* (260kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.2 Spindle

16.1.2.1 MDS-D-SP (200V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.2.2 MDS-DH-SP (400V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.2.3 MDS-D-SPJ3 (200V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.2.4 MDS-D-SP2 (200V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.2.5 MDS-DM-SPV2/SPV3 (200V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.3 Auxiliary Axis

16.1.3.1 Index/Positioning Servo : MR-J2-CT

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

16.1.3.1.1 Servo motor : HC-SF/HC-RF (16 kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

16.1.3.1.2 Servo motor : HA-FF/HC-MF (8 kp/rev)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	—	—	—
L system	□	□	□	—	—	—

16.1.4 Power Supply

16.1.4.1 Power Supply : MDS-D-CV (200V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.4.2 Power Supply : MDS-DH-CV (400V)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.4.3 AC Reactor for Power Supply

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

16.1.4.4 Ground Plate

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

17

Machine Support Functions

17.1 PLC

17.1.1 Built-in PLC Processing Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

An exclusive sequence program that controls the various signals between the controller and machine to realize operation applicable to each machine can be created and built in.
The sequence execution modes include high-speed processing and main processing.

(1) High-speed processing
This mode provides repeated execution at 3.5ms cycles. It is used to process signals requiring high speeds.
The max. number of program steps for high-speed processing (1 period) is 1000 steps when using basic commands.

(2) Main processing
This mode provides normal sequence processing. The processing cycle depends on the number of sequence steps.

17.1.2 PLC Functions

17.1.2.1 Built-in PLC Basic Function

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.
L system	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.	○ *Index qualifi-cation is available.

(1) Ladder commands
Basic commands (bit processing commands)
43 commands including LD, LDI, OR, ORI, AND, ANI, OUT, PLS, etc.
Function commands
188 commands including data transfer, 4 basic arithmetic operations, logic arithmetic operations, large/small identification, binary/BCD conversion, branching, conditional branching, decoding, encoding, etc.

17.1.2.2 PLC Exclusive Instruction

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

PLC dedicated instruction is provided for certain limited applications, enabling a complex machining process, which is difficult to carry out only by the basic instructions and function instructions.

PLC dedicated instructions include:

(1) ATC dedicated instruction (ATC)

This is an instruction to function ATC, or magazine index control, tool exchange with arm, etc.
ATC dedicated instructions are as follows.

- Tool No. search
- Tool change
- Tool table forward/reverse run
- Pointer (which indicates magazine index position) forward/reverse run
- Tool data read/write

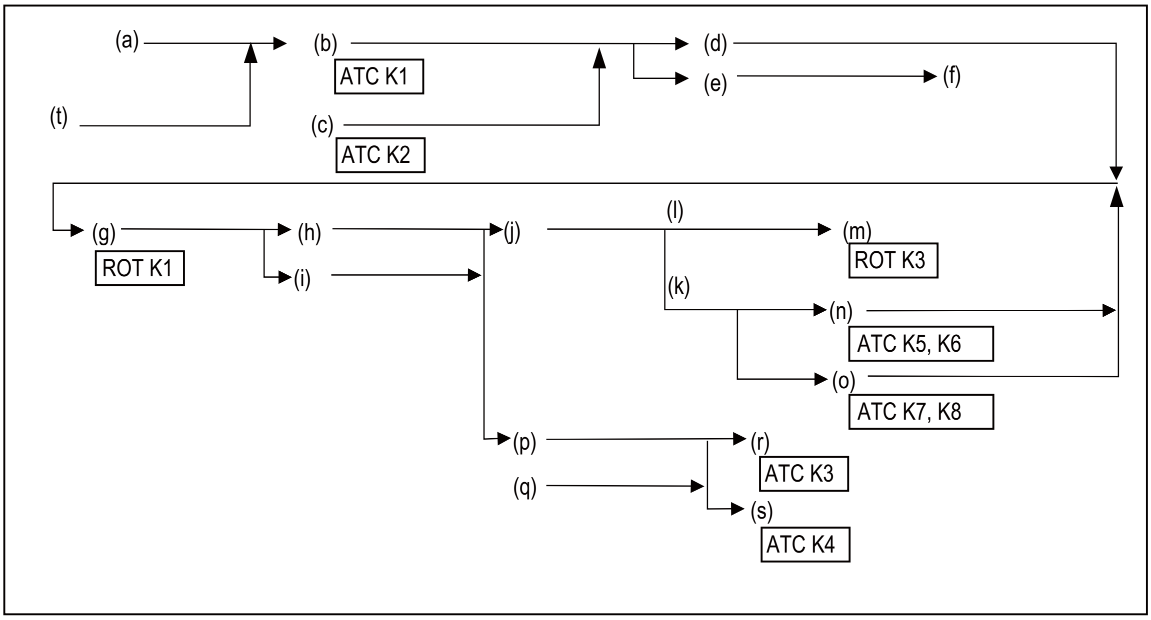
(2) Rotary body control instruction (ROT)

This is an instruction to determine the rotary body's target position or rotation direction, or to function as a ring counter.

This is used when calculating the rotation direction or number of index steps of the magazine or turret based on the output data figured from ATC dedicated instruction tool No. search processing, or when controlling the rotary body position.

Using the ATC and ROT instructions

The order for using the ATC and ROT instructions when T is commanded or tool exchange is commanded is shown below.



(a) T command
(b) Tool No. search
(c) Tool No. logical search
(d) Matching place No.
(e) Number of matches
(f) Error process
(g) Rotary body index
(h) Turning direction
(i) Number of steps, etc.
(j) Magazine turn
(k) Variable pointer method
(l) Fixed pointer method
(m) Ring counter control
(n) Pointer forward run/reverse run
(o) Tool table forward run/reverse run
(p) Magazine stop
(q) Tool change command
(r) Tool change
(s) Arbitrary position tool change
(t) Pointer or ring counter value

(1) Devices

The table below lists the devices which can be used by the PLC. (PLC4B)

Device	Device No.	Unit	Details
X*	X0 to X1FFF (8192 points)	1 bit	Input signal to PLC: Machine input, etc.
Y*	X0 to Y1FFF (8192 points)	1 bit	Output signal from PLC: Machine output, etc.
M	M0 to M10239 (10240 points)	1 bit	Temporary memory
F	F0 to F1023 (1024 points)	1 bit	Temporary memory. Alarm message interface
L	L0 to L511 (512 points)	1 bit	Latch relay (back up memory)
SB	SB0 to SB1FF (512 points)	1 bit	Link special relay
B	B0 to B1FFF (8192 points)	1 bit	Link relay
SM*	SM0 to SM1023 (1024 points)	1 bit	Special relay
V	V0 to V255 (256 points)	1 bit	Edge relay
SW	SW0 to SW1FF (512 points)	16 bit	Link special relay
SD	SD0 to SD1023 (1024 points)	16 bit	Special register
T	T0 to T703 (704 points)	1 bit/16 bit	Timer (The variable/fixed boundary is set with a parameter.) (Note 2)
ST	ST0 to ST63 (64 points)	1 bit/16 bit	Integrated timer (100ms unit)
C	C0 to C255 (256 points)	1 bit/16 bit	Counter (The variable/fixed boundary is set with a parameter.)
D	D0 to D2047 (2048 points)	16 bit/32 bit	Data register. Register for calculation
R*	R0 to R13311 (13312 points)	16 bit/32 bit	File register. CNC word interface
W	W0 to W1FFF (8192 points)	16 bit/32 bit	Link register
Z	Z0 to Z1 (2 points)	16 bit	Address index
N	N0 to N7 (8 points)		Master controller nesting level
P*	P0 to P2047 P4000 to P4005 (2048 points) (Note3)		Label for conditional jump, subroutine call command
K	K-32768 to K32767		Decimal constant for 16-bit command
	K-2147483648 to K2147483647		Decimal constant for 32-bit command
H	HO to HFFFF		Hexadecimal constant for 16-bit command
	HO to HFFFFFFFF		Hexadecimal constant for 32-bit command

(Note 1) Devices marked with * in the device column have designated applications. Do not use devices other than those corresponding to the input/output signals with the machine side (input/output signals of the remote I/O unit), even if it is an undefined vacant device.

(Note 2) Distinction of 10ms timer and 100ms timer is performed by command. (10ms timer is performed by OUTH command, 100ms timer is performed by OUT command.)

(Note 3) There are two types of P-device: common pointer and local pointer. The points above is the sum of the two.

(2) 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.

(3) 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.

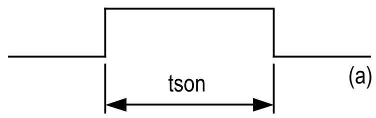
(4) Real spindle speed output

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.

(5) High speed input/output signal

There are signals that can be input and output at a 3.5ms cycle for high-speed processing.

(a) Input signal ON time



(a) tson ≧ 3.5ms

(b) After the signal output is set in the interface, it can be output to the machine side with a max. 3.5ms delay. The input also appears on the interface with a 3.5ms delay.

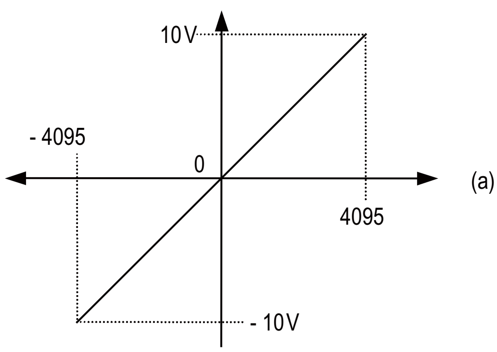
(c) The signals used for high-speed processing are assigned with the parameters. Assignment is possible in a continuous 16-point unit.

(6) 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>



(a) Contents of file register

Output voltage	0 to ± 10V(± 5%)
Resolution	Full scale (10V)/4095
Load condition	10kΩ resistance load (standard)
Output impedance	220Ω

(Note) The remote I/O unit DX120/DX121 is required for analog output.

17.1.3 PLC Support Functions

17.1.3.1 Alarm Message Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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 46 characters.

17.1.3.2 Operator Message Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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.

17.1.3.3 Memory Switch (PLC Switch)

17.1.3.3.1 PLC Switch 32 Points

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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.

17.1.3.3.2 PLC Switch 64 Points

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

64 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.

17.1.3.4 Load Meter Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

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.

17.1.3.5 User PLC Version Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The user PLC version can be displayed in the software list on the Software Configuration screen.

17.1.3.6 Multi-ladder Program Register and Execution

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Up to 20 sequence programs can be registered. Only one execution type can be set in one program. The following five execution types are available

- "Initial" (Initialization process)	: Started only once when power is turned ON.
- "Scan" (High-speed process)	: Started at the standard interrupt cycle.
- "Scan" (Main process)	: Constantly started in process other than high-speed process.
- "Standby" (Standby process)	: Called from the high-speed process or main process.
- "Low-speed"	: This execution type is not used.

17.1.3.7 Ladder Program Writing during RUN

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Ladder program can be edited while PLC is running. This function is available, either by GX Developer or PLC onboard edit.

17.1.3.8 PLC Protection

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

A keyword can be set to protect the sequence programs stored in CNC.
The following two types of conditions can be selected at the keyword registration.
- Write protection: Writing and editing are restricted.
- Read/Write/Display protection: Writing, reading, verification and list display are restricted.

For details, refer to the PLC Programming Manual.

17.1.4 Built-in PLC Capacity [Steps]

17.1.4.1 Standard PLC Capacity

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ 64000	○ 64000	○ 64000	○ 64000	○ 64000	○ 64000
L system	○ 64000	○ 64000	○ 64000	○ 64000	○ 64000	○ 64000

17.1.4.2 Large PLC Capacity

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△ 128000	△ 128000	△ 128000	△ 128000	△ 128000	△ 128000
L system	△ 128000	△ 128000	△ 128000	△ 128000	△ 128000	△ 128000

17.1.5 Machine Contact Input/Output I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

⚠ CAUTION

❗ Follow the remote type machine contact input/output interface described in this manual.
(Connect a diode in parallel with the inductive load or connect a protective resistor in serial with the capacitive load, etc.)

Refer to the "Hardware Specifications" in the "General Specifications" for details.

With the Mitsubishi CNC M700V Series, if an operation panel I/O unit is mounted, up to three channels can be used for the remote type machine contact input, and up to three channels can be used for the handled.
If the operation panel I/O unit is not mounted, up to two channels can be used for the remote type machine contact input, and up to two channels can be used for the handled.
The handle connection method differs for when the operation panel I/O unit is mounted or not.
When using the operation panel I/O unit, the expansion DIO card (HN397) can be mounted to increase the DIO by three channels.
Remote I/O unit list

17.1.6 Ladder Monitor

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function enables the operating status of the sequence circuit to be checked on the controller's setting and display unit.
The monitor functions include the following.

(1) Circuit monitoring
(2) Screen stop by monitor stop trigger point
(3) Entry monitoring
(4) Decimal-hexadecimal conversion present value monitoring

17.1.7 PLC Development

17.1.7.1 On-board Development

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

On-board refers generically to the PLC related operations carried out with the CNC unit.
The Mitsubishi CNC M700V Series on-board realizes functions and operations similar to the MELSEC Series ladder development tool (GX Developer).
This improves the user's ladder editing operations, and provides an easy-to-use onboard with powerful ladder monitoring functions.

17.1.7.2 MELSEC Development Tool (GX Developer)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function enables the data of the PLC contained inside the NC system to be developed and debugged using the GX Developer installed in a personal computer (OS:Windows).
Many and varied functions of the GX Developer make it possible to reduce the PLC data development and debugging time.

17.1.8 PLC Parameter

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.

17.1.8.1 PLC Constant (150 Points)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

17.1.8.2 PLC Constant Extension (Up to 755 Points)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

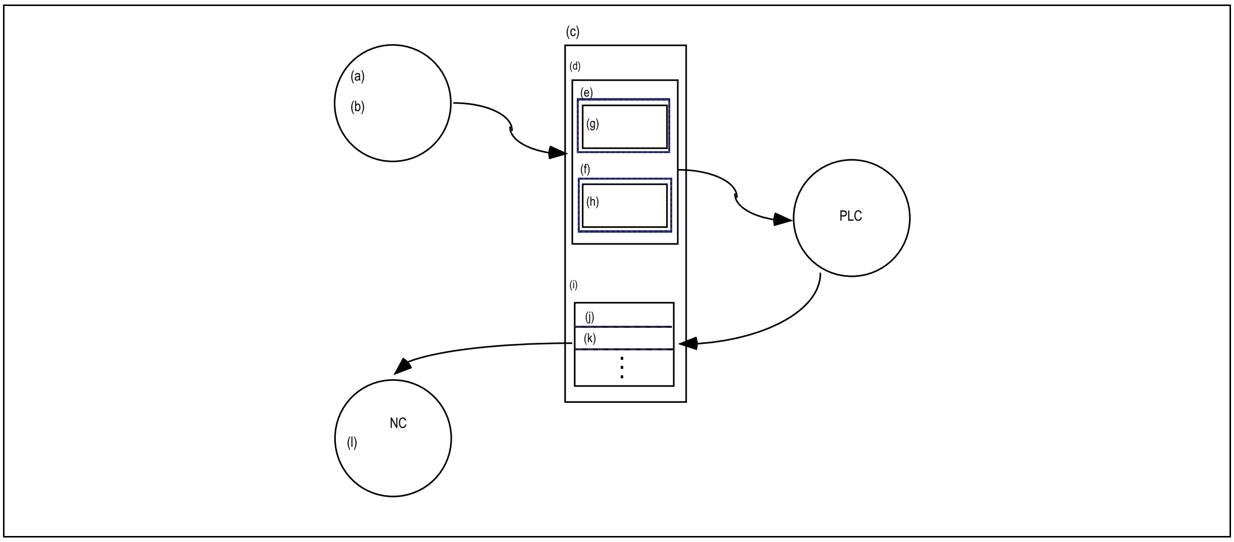
There are PLC constants that can be set with data types as parameters used by the built-in PLC. The set data is set in the R register of the PLC and backed up. If data is set in the R register corresponding to the PLC constant with sequence program MOV commands, etc., the data will be backed up in the PLC constant parameter. However, the display will not change, so enter another screen, and then select this screen again.
The number of PLC constants is 150 points in the basic area and 755 points at the maximum in the extended area.

17.1.10 Pallet Program Registration

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

Pallet program function allows machining program to be registered for each pallet of the auto pallet changer (hereinafter referred to as APC) and assists in the machining setups.
Operation search for the registered program will be executed by PLC using the external search function.

Machining program for each pallet is registered at "Pallet program registration" screen in the setup screen group.
Registered program is output to PLC device.
If necessary, execute an external search on PLC ladder by referring to the program No. registered for each pallet.



(a) HMI system
(b) Pallet program registration screen
(c) PLC device
(d) Pallet program registration interface
(e) Pallet 1
(f) Pallet 2
(g) Machining program No.
(h) Operation program No.
(i) External search interface
(j) Device 1
(k) Program No.
(l) main system

17.2 Machine Construction

17.2.1 Servo OFF

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

When the servo OFF signal (per axis) is input, the corresponding axis is set in the servo OFF state.
When the moving axis is mechanically clamped, this function is designed to prevent the servomotor from being overloaded by the clamping force.
Even if the motor shaft should move because of some reason 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.

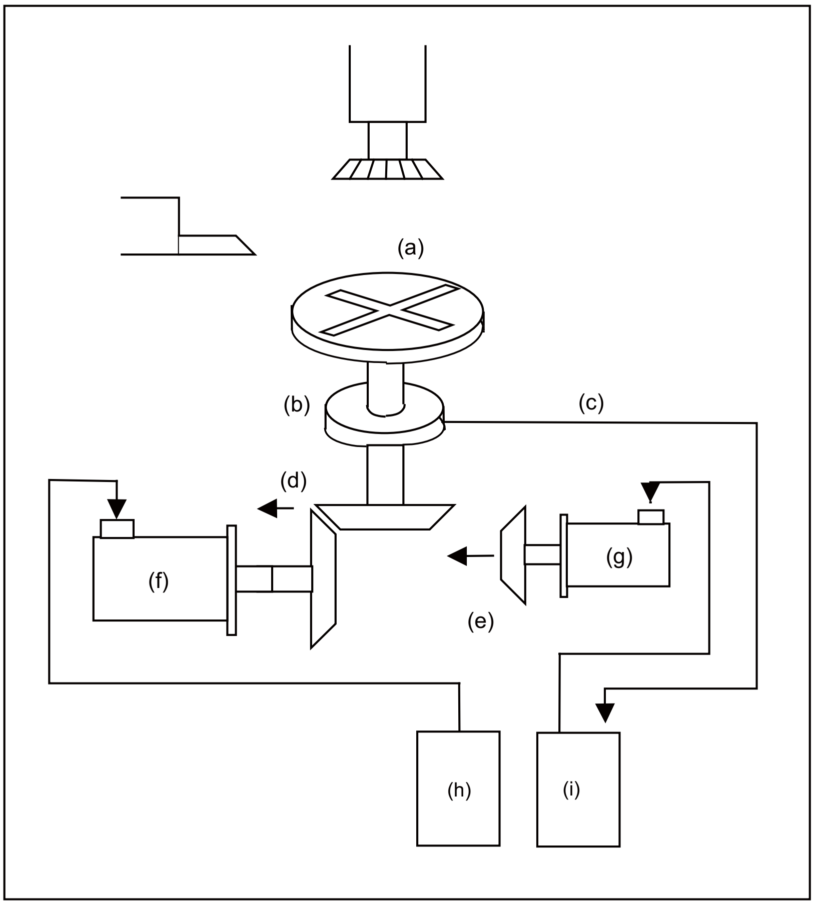
When follow up function is designated, the movement amount will be compensated even in the emergency stop state.
The axis is simultaneously set to servo OFF state and the interlock state.

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 position display will not deviate.

17.2.2 Axis Detachment

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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 position return.



This shows the configuration of a machine for which switching between the C axis and turning table is performed. When the spindle motor is connected, the C axis is placed in the detached status.
As a result, the position feedback of the detector is ignored.

(a) C-axis/turning table
(b) Rotary magnetic scale
(c) (Position feedback)
(d) (OFF with C-axis control)
(e) (Coupled with C-axis control)
(f) Spindle motor
(g) C-axis motor
(h) Spindle drive unit
(i) C-axis drive unit

POSITION X 123.456 Z 0.000#1 C 345.678><	The detached status > < is indicated on the right of the POSITION display on the POSITION screen and at the same time the servo ready for the controller output signal is set to OFF. The 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.

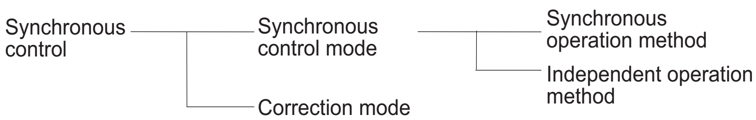
17.2.3 Synchronous Control

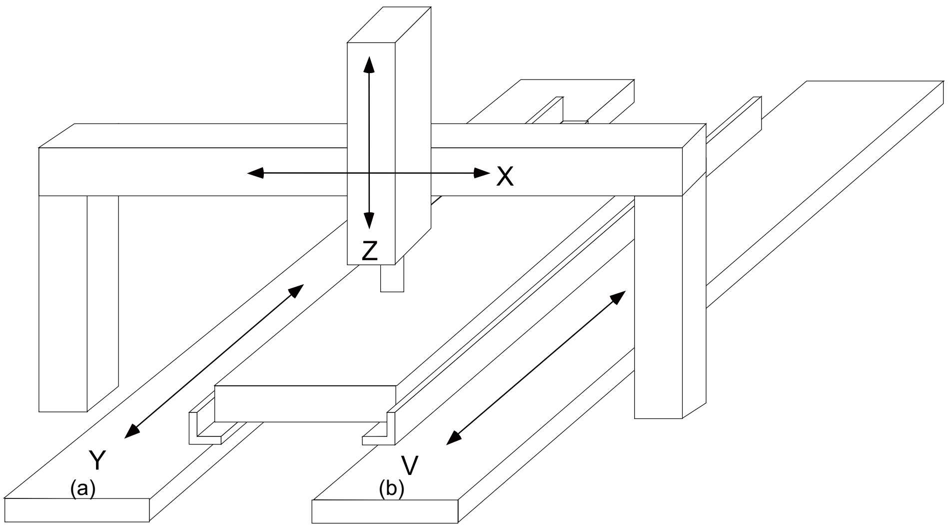
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

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 control axis synchronization across part systems and the synchronous control cannot be used simultaneously. (L system)

- The slave axis is controlled with the movement command for the master axis.
- One slave axis can be set to one master axis.
- Up to 3 sets of master axis / slave axis can be set in total for all the part systems.

The structure of the synchronous control is as follows.





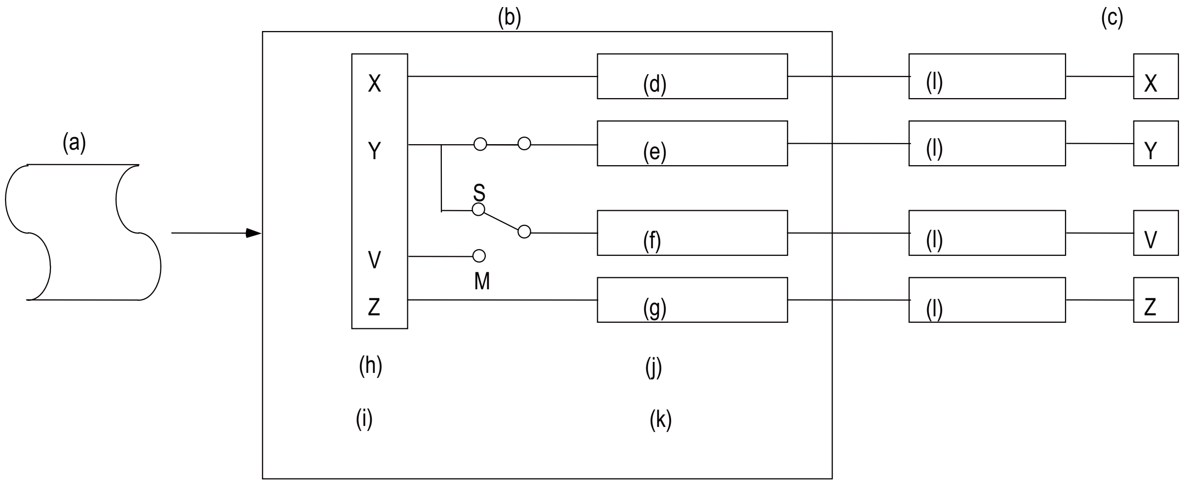
(a) (Master axis)
(b) (Slave axis)

(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.



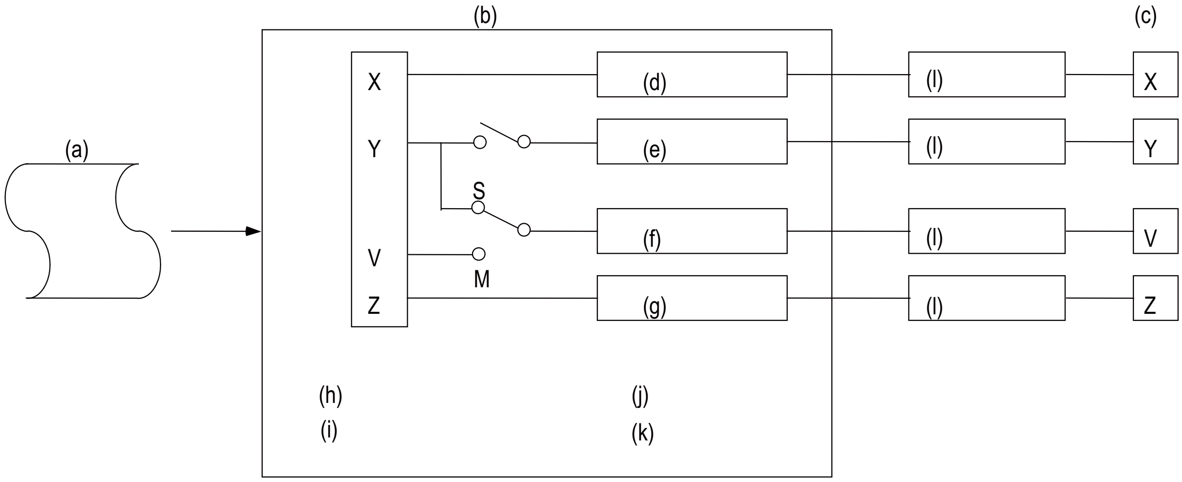
(a) Machining program
(b) CNC system
(c) Axis motor
(d) X axis control
(e) Y axis control
(f) V axis control
(g) Z axis control
(h) NC control section
(i) Calculation of movement directions, movement amount Calculation of feed rate
(j) Position control section
(k) Reference position return Backlash compensation
(l) Servo control

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 error between the positions exceeds the allowable synchronization error value set in the parameter. However, when the zero point is not established, the synchronous error is not checked.

Even during synchronous operation, pitch error compensation, backlash compensation and external machine coordinate compensation are performed independently for each master axis and slave axis.

Designation/cancellation of synchronous operation is executed at "all axes in-position".

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



(a) Machining program
(b) CNC system
(c) Axis motor
(d) X axis control
(e) Y axis control
(f) V axis control
(g) Z axis control
(h) NC control section
(i) Calculation of movement directions, movement amount Calculation of feed rate
(j) Position control section
(k) Reference position return Backlash compensation
(l) Servo control

Even during independent operation, pitch error compensation, backlash compensation and external machine coordinate compensation are performed independently for each master axis and slave axis.
Designation/cancellation of independent operation is executed at "all axes in-position".

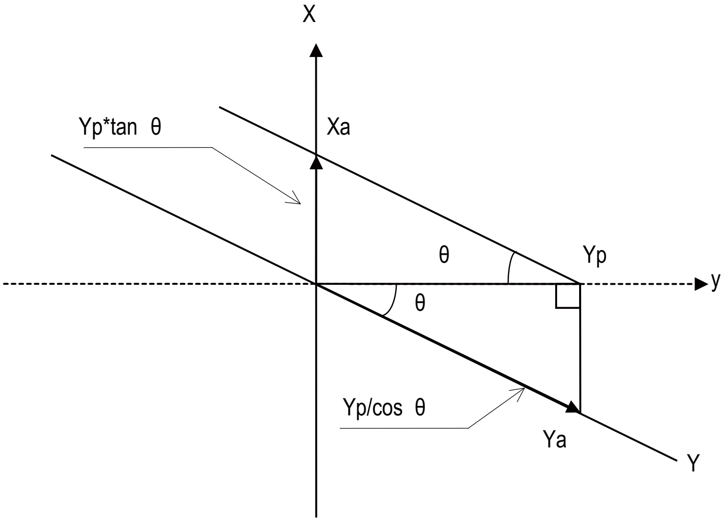
(2) Correction mode
The synchronization is temporary canceled to adjust the balance of the master and slave axes during the synchronous control mode in the machine adjustment. Each axis can be moved separately with the manual handle feed or the arbitrary feed in manual mode. If the operation mode other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

17.2.4 Inclined Axis Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	—	—	—
L system	△	△	△	△	△	△

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables it to be controlled by the same program as that for an orthogonal axis.
The inclination angle is set using a parameter, and axes are controlled using the movement amounts of the axes which are obtained through conversion and compensation using this angle.

<Example of use> When the X axis serves as the basic axis and the Y axis serves as the inclined axis



X: Actual X axis
Y: Actual Y axis
y: Programmed Y axis
θ : Inclination angle

Yp, the Y-axis position on the programmed coordinates (on the orthogonal coordinates), is the position of Xa and Ya which are produced by synthesis of X axis and Y axis.
Therefore, the Y-axis (inclined axis) movement amount is expressed by the following formula:
Ya=Yp/cos θ …… (1)
The X-axis (basic axis) movement amount is compensated by the inclined movement of the Y axis, and it is expressed as follows:
Xa=Xp-Yp*tan θ … (2)
The Y-axis (inclined axis) speed is as follows:
Fa=Fp/cos θ

Xa, Ya and Fa are the actual movement amounts and speed. Xp, Yp and Fp are the movement amounts and speed on the program coordinates.

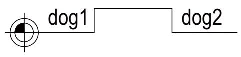
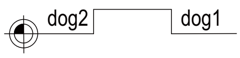
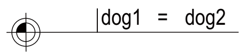
17.2.5 Position Switch

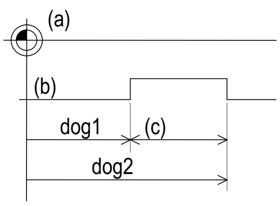
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24
L system	○ 24	○ 24	○ 24	○ 24	○ 24	○ 24

The position switch (PSW) function provides hypothetical dog switches in place of the dog switches provided on the machine axes by setting the axis names and coordinate position 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 coordinate position indicating the hypothetical dog positions (dog1, dog2) on the coordinate axes whose names were set by parameters ahead of time in place of the dog switches provided on the machine axes are set using position switches. When the machine has reached the hypothetical dog positions, a signal is output to the device supported by the PLC interface.

Example of dog1, dog2 settings and execution

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



(a) Basic machine coordinate system zero point
(b) Hypothetical dog
(c) PSW width

17.2.7 Index Table Indexing

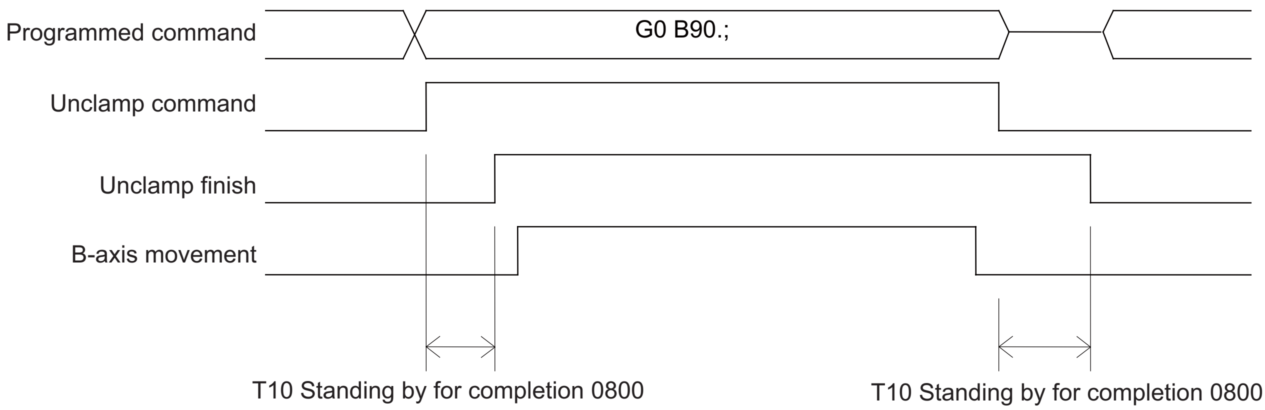
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The indexing of the index table can be performed by setting the index axes.
Programming is facilitated because, in terms of the index commands, only the indexing angle need to be designated using the address of the programmed axis serving as the index setting axis, and there is no need to designate special M codes for clamping and unclamping the table.

The following operations are performed for the index table indexing function.

(1) Set "1" to the "index axis selection" parameter for the axis along which the indexing table is to be indexed.
(2) Designate the movement commands (absolute or incremental) for the selected axis using a program.
(3) The unclamp command signal is now output prior to the axis movement.
(4) When the axes are unclamped, the unclamp finish signal is set (ladder used for processing).
(5) After checking the unclamp finish signal, the designated axis starts moving.
(6) Upon completion of the movement, the unclamp command signal is set to OFF.
(7) Clamp the axes and set the unclamp finish signal to OFF (ladder used for processing).
(8) After checking that the unclamp finish signal is OFF, processing of the next block is initiated.

<Operation timing chart>



17.2.8 Auxiliary Axis Control (J2-CT)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	—	—	—
L system	△	△	△	—	—	—

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) The constant inclination acceleration/deceleration is automatically controlled. 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 No. by the start signal.

(c) Manual mode
In this mode, the axis is rotated at a constant speed in the designated direction while the start signal is ON. When the start signal is set to OFF, the axis is positioned at the nearest station position.

(d) Arbitrary coordinate mode
In this mode, the axis is positioned at the arbitrary coordinates designated with the PLC by the start signal. When the start signal is set to OFF prior to the completion of the positioning, the axis immediately decelerates and stops.

(e) Manual handle mode
In this mode, axis travel is carried out by the pulse command (manual handle command) sent from the PLC.

(f) Reference position return mode
In this mode, the axis is positioned at the coordinate reference position. Two methods are used: one method is based on a dog switch and the other method is to carry out positioning to the reference position 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.

17.2.9 Tool Length Compensation along the Tool Axis

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

(1) Changing the tool length compensation in tool axis direction and compensation amount

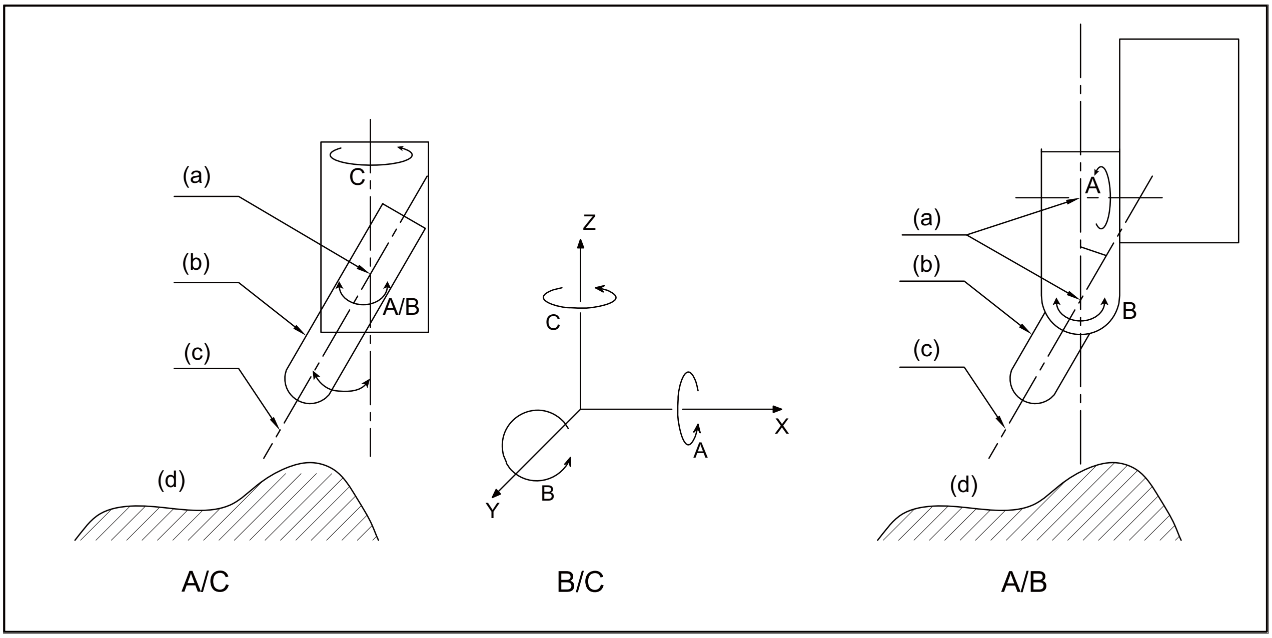
Even if the tool axis direction is not the Z axis direction because the rotary axis is rotated, the tool can be compensated in the tool axis direction. By setting the deviation of the tool length value set when the program was created and the actual tool length as a compensation amount, the program can be made more flexible. This is especially effective for programs with many rotary axis movement commands.

The tool length compensation amount in the tool axis direction can be changed by rotating the manual pulse generator when the tool length compensation along tool axis mode and tool length compensation amount along the tool axis change mode are valid.

(2) Machine configuration

The tool length compensation in tool axis direction is carried out in respect to the direction of the tool nose axis (rotary axis).

The axis which determines the compensation direction is designated with the parameters as a combination of the Z-axis rotation C axis (spindle) and X-axis rotation A axis or Y-axis rotation B axis.



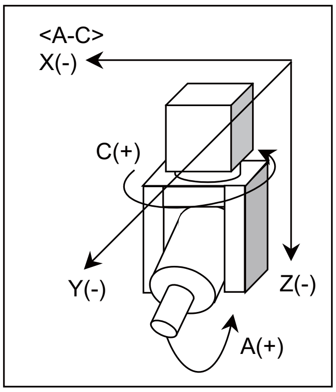
(a) Rotation center
(b) Tool
(c) Axis direction (compensation direction)
(d) Workpiece

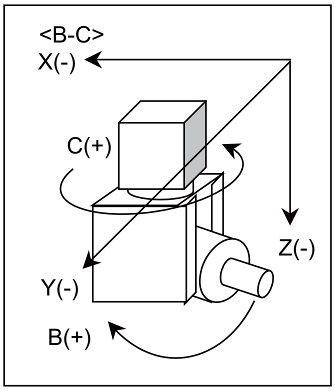
17.2.10 Tool Handle Feed & Interruption

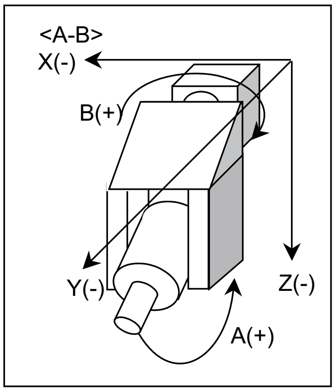
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	△	△	—	△	△
L system	—	—	—	—	—	—

The tool handle feed & interrupt function makes it possible to move the axis with the manual pulse generator in the tool axis direction, tool diameter direction X and tool diameter direction Y within the hypothetical coordinate system over the tool axis.

Normal handle feed and interrupt is applied when this function is not provided.







This is also effective for a machine configuration having the tool tilt 1 axis and table tilt 1 axis.

17.2.11 Tool Center Coordinate Display

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	△	—	—	△
L system	—	—	—	—	—	—

The tool center coordinates, handle interrupt amount (tool axis movement) and tool center point speed are displayed during the tool center point control function, tool length compensation along tool axis function, and tool handle feed & interrupt function (tool axis direction handle feed, tool handle interrupt, tool diameter direction handle feed, nose center rotation handle feed).
The path using the tool center coordinate position is drawn on the Graphic Trace screen.
The 5-axis related function option is required to validate this function.
The tool center point will not be displayed if the 5-axis related function option is not provided.

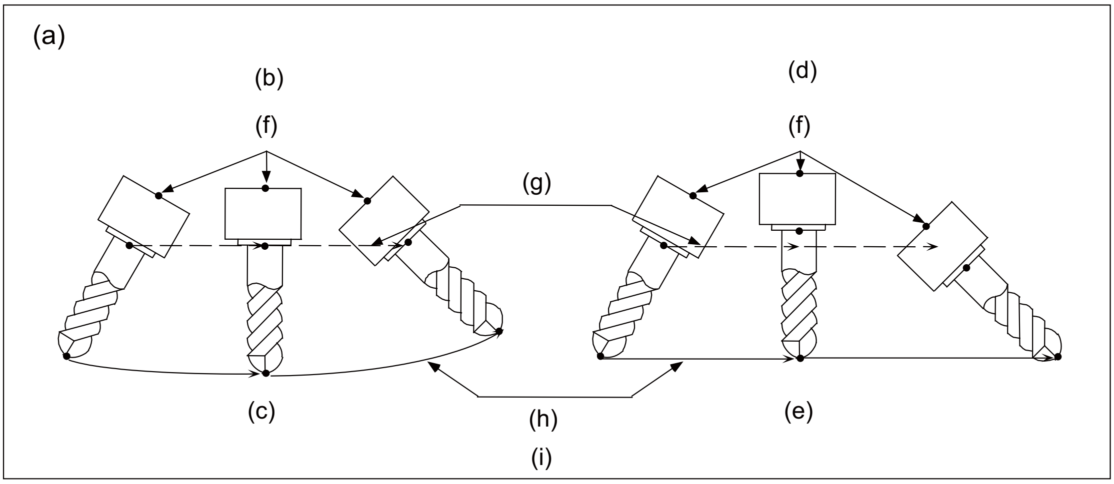
<5-axis related function>

- Tool length compensation along tool axis function
- Tool handle feed & interrupt function
- Tool center point control function

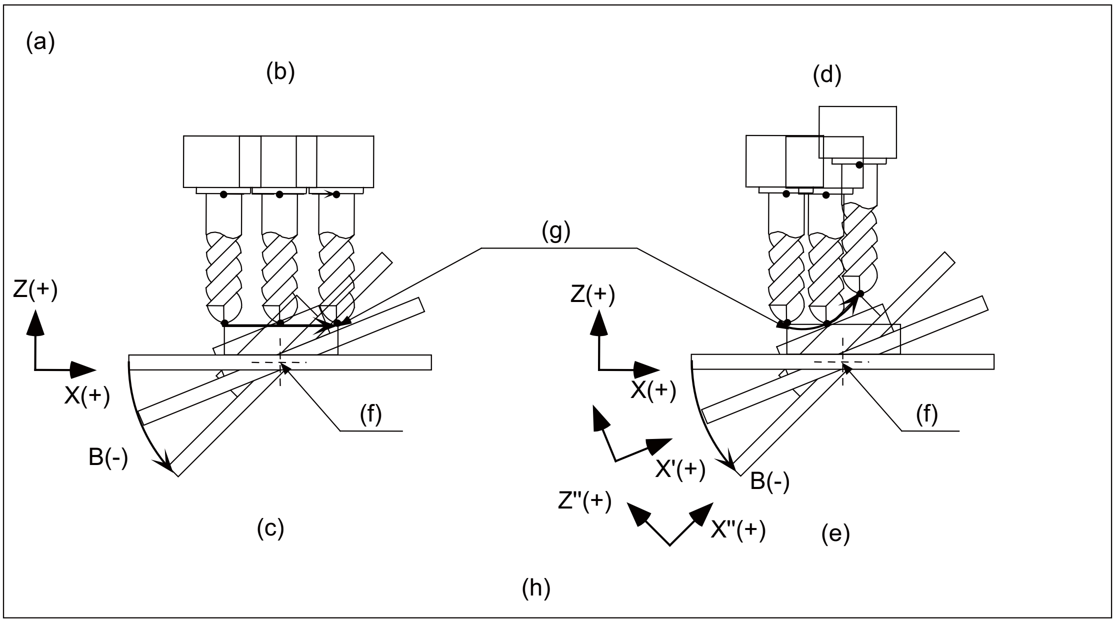
17.2.12 Tool Center Point Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	△	—	—	△
L system	—	—	—	—	—	—

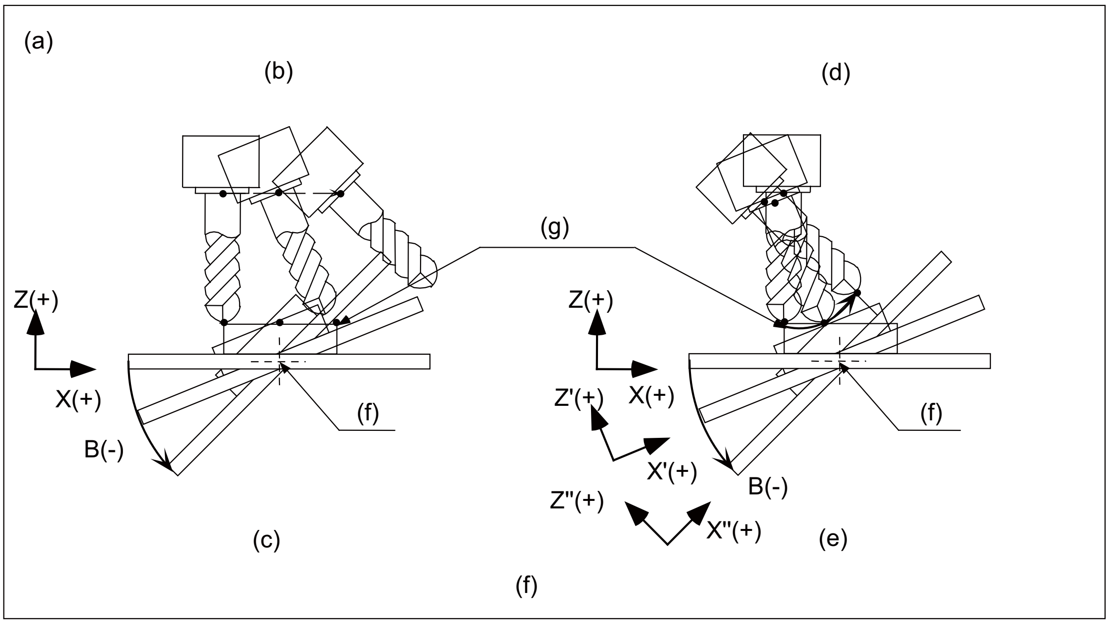
The tool center point control function controls so that the position command in the machining program is at the tool center point on the coordinate system (table coordinate system) which rotates together with the workpiece. This function can be applied with the 5-axis machine including the tool tilt type with two rotary axes on the head (Fig. 1 (a)), the table title type with two rotary axes on the table (Fig. 1 (b)), and the compound type with rotary axes on the tool and table (Fig. 1 (c)).



(a) <Tool tilt type>
(b) When tool center point control is OFF and tool length compensation along tool axis is ON
(c) Controls so that the holder center point moves straight.
(d) When tool center point control is ON
(e) Controls so that the tool center point moves straight.
(f) Rotation center
(g) Program path
(h) Tool center point path
(i) Fig. 1 (a)



(a) <Table tilt type>
(b) When tool center point control is OFF and tool length compensation along tool axis is ON
(c) Controls so that the holder center point is at the workpiece coordinate system's position.
(d) When tool center point control is ON
(e) Controls so that the tool center point is at the table coordinate system's position.
(f) Rotation center
(g) Tool center point path
(h) Fig. 1 (b)



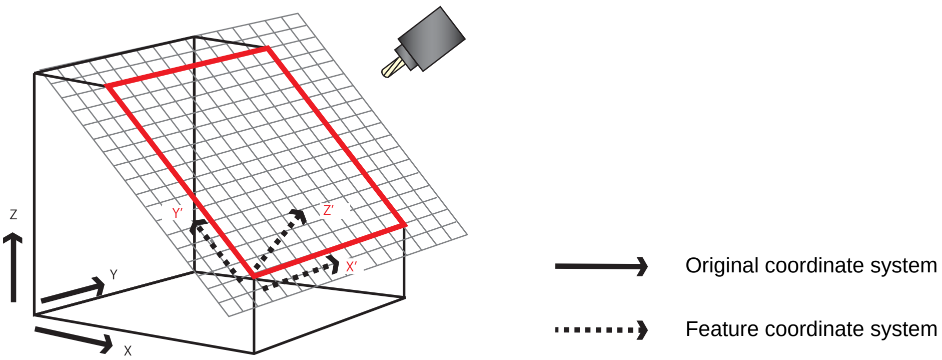
(a) <Compound type>
(b) When tool center point control is OFF and tool length compensation along tool axis is ON
(c) Controls so that the holder center point is at the workpiece coordinate system's position.
(d) When tool center point control is ON
(e) Controls so that the tool center point is at the table coordinate system's position.
(f) Rotation center
(g) Tool center point path
(h) Fig. 1 (c)

An option is required to validate this function. If tool center point control is commanded when the option is not provided, a program error will occur.

17.2.13 Inclined Surface Machining Command

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	—	—	—	—	—	—

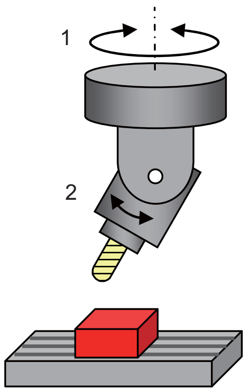
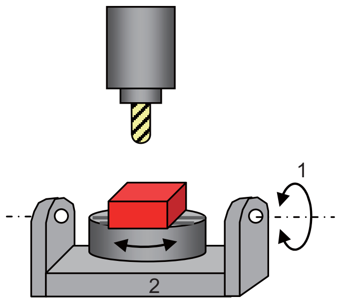
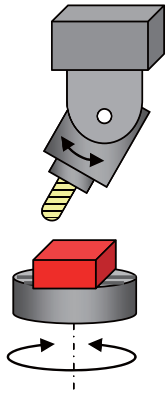
This function is to define the new coordinate system (called the "feature coordinate system") which was created by rotation and reference position translation of the X, Y, Z axes of the original one (before the inclined surface machining command was applied). By using this function, an arbitrary spatial plane can be defined, and machining on that plane can be carried out with normal program commands.
Also, the tool axis can automatically be controlled to the + Z direction of the newly defined feature coordinate system. Since the feature coordinate system will be re-created in accordance with the tool axis direction, machining programs can be developed without the need to consider the direction of the feature coordinate system and rotation of the tool axis.
This function is available only for the 5-axis machining.



The feature coordinate system is defined in the following method.

- Designation with Euler angle.
- Designation with the roll angle, pitch and yaw angle.
- Designation with the three points on the plane.
- Designation with two vectors.
- Designation with a projection angle.
- Designation with the tool axis direction.

This function is compatible with the following types of machine.

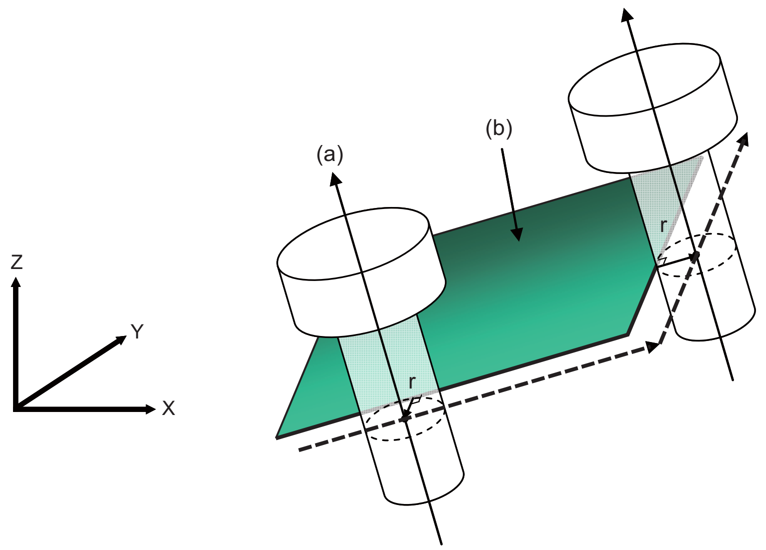
Type	Tool tilt type	Table tilt type	Compound type
Description	Two rotary axes on the tool side	Two rotary axes on the table side	One rotary axis on each of the tool side and table side
Example of machine			
Primary rotary axis	The 2nd rotary axis on the tool side	The 2nd rotary axis on the table side	Rotary axis on the tool side

17.2.14 Tool Radius Compensation for 5-axis Machining

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	△	—	—	△
L system	—	—	—	—	—	—

This function is to compensate the tool radius of the 5-axis machine with two rotary axes, in accordance with the change of the workpiece direction and inclination of the tool due to the movement of the rotary axis. The 3-dimensional tool radius compensation was realized by searching the tool path on the workpiece by the program command and calculating the compensation vector on the plane (compensation plane) which is vertical to the path in the tool direction

This function is available only for 5-axis machine and requires an option.



r : Compensation amount

- - - - - - : Tool center path

———— : Program path

(a) : Tool direction

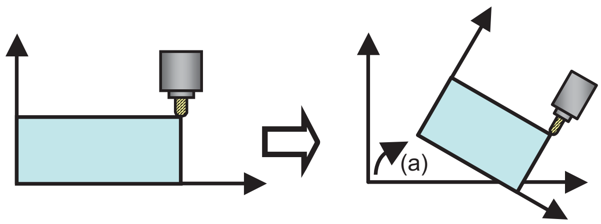
(b) : Compensation plane

This function is compatible with the same types of machine as listed in the inclined surface machining command.

17.2.15 Workpiece Installation Error Compensation

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	△	—	—	△
L system	—	—	—	—	—	—

This function is for the 5-axis machine. This compensates the error when a workpiece is placed off the workpiece coordinate system to enable machining according to the program. In this function, a new coordinate system with the workpiece as its reference position will be defined (called "workpiece installation coordinate system) and the program will be executed in this new coordinate system.



(a) Error at the workpiece installation

This function is compatible with the same types of machine as listed in the inclined surface machining command.

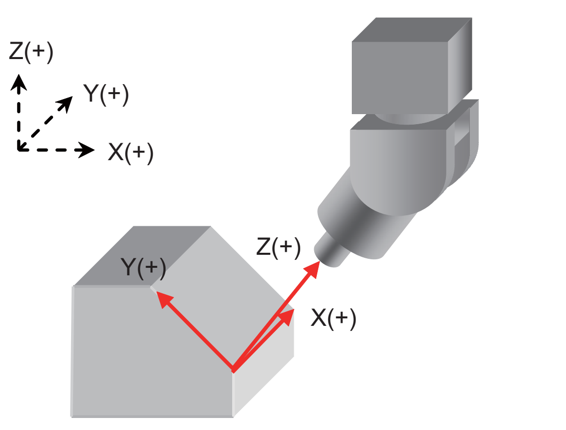
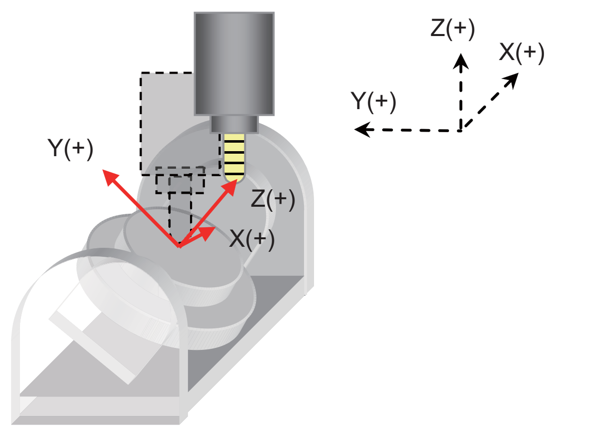
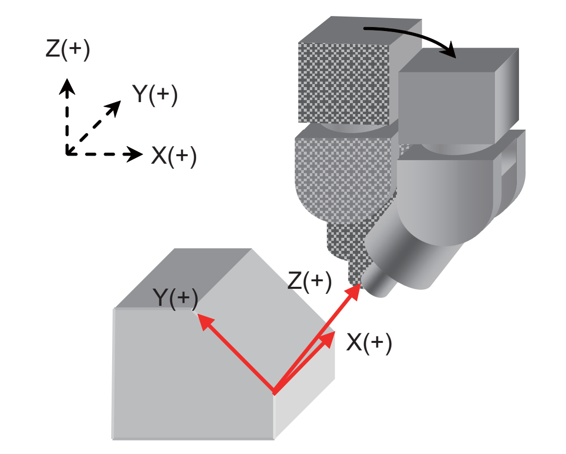
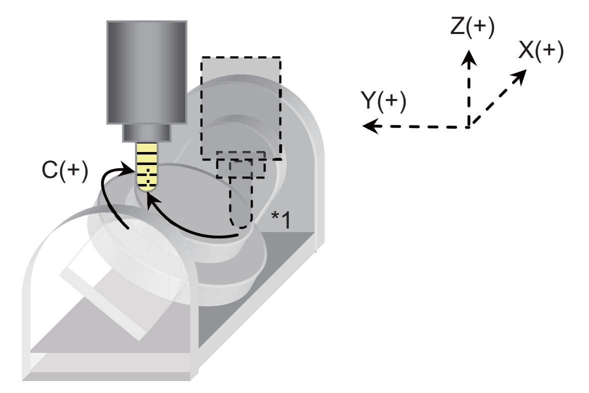
17.2.16 Manual Feed for 5-axis Machining

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	△	—	—	△
L system	—	—	—	—	—	—

By selecting the hypothetical coordinate system to be machined, axis can be moved with manual feed (JOG, HANDLE or INCREMENTAL) in the coordinate system with this function.
If a movement command on the hypothetical coordinate is issued, it can be easy to setup because multiple axes is moved by NC according to the tool angle or the inclination of the table.

The manual feed for 5-axis machining function consists of the hypothetical coordinate feed and tool center point rotation.
Movement command for the linear axis operates as the hypothetical coordinate feed and manual feed will be carried out on the hypothetical coordinate.
Movement command for the rotary axis operates as the tool center point rotation and manual feed will be carried out with the rotary axis and 3 linear axes which are issued a movement command to keep position relations of the workpiece and tool tip.

Standard manual feed is applied when this function is not provided.

	<Tool tilt B-C axis>	<Table tilt A-C axis>
Linear axis: hypothetical coordinate feed		
Rotary axis: tool center point rotation		 *1 Keep the position of the workpiece and tool tip along C axis rotation.

Original coordinate system

Hypothetical coordinate system

17.3 PLC Operation

17.3.1 Arbitrary Feed in Manual Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

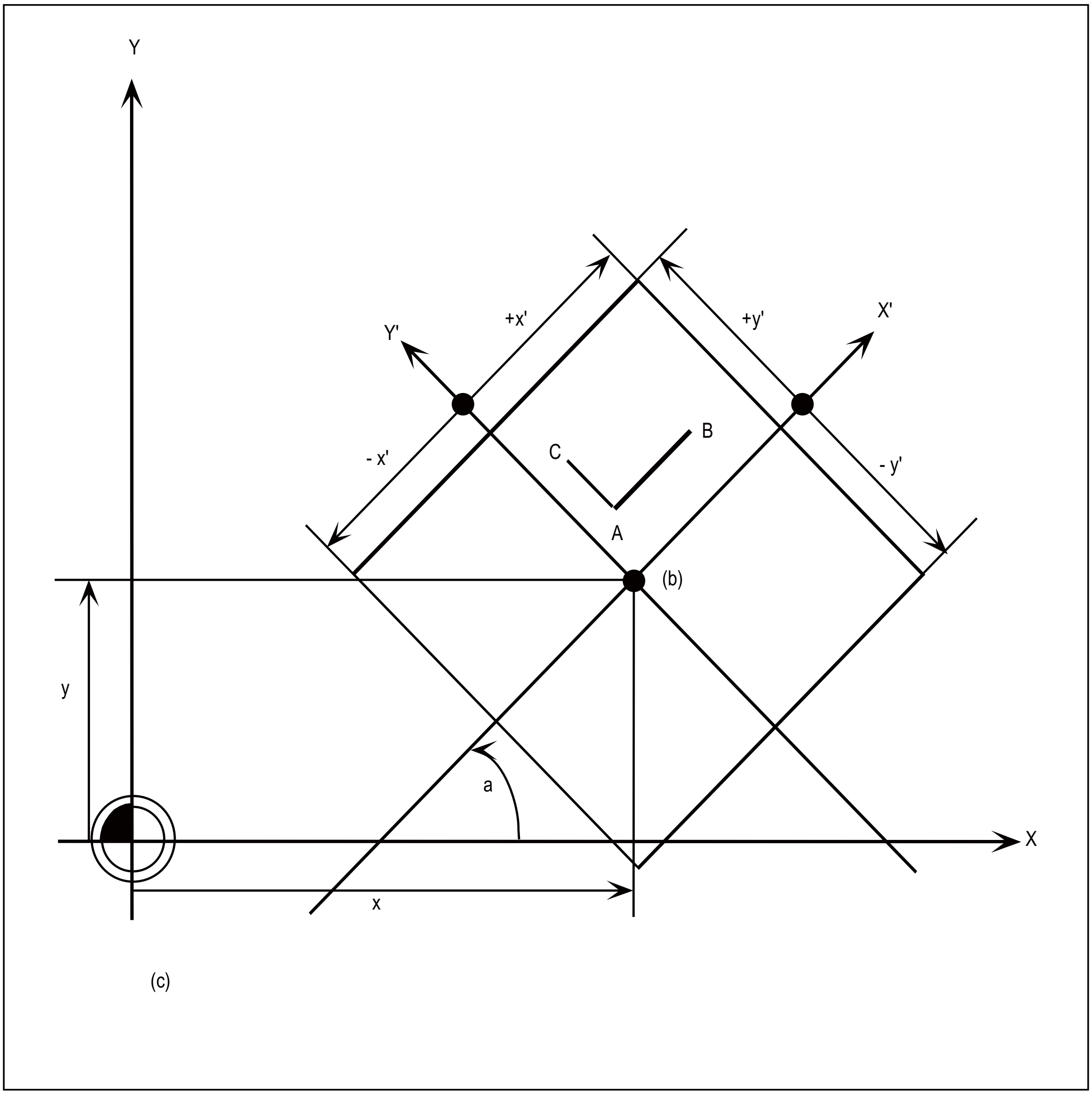
This function enables the feed directions and feed rates of the control axes to be controlled using commands from the user PLC.
The arbitrary feed function controls the movement of the axes at the specified rates while the start signal is output from the PLC to the NC system.
PLC operations can be performed even during manual operation or automatic operation, but they cannot be performed when an axis for which arbitrary feed has been assigned is executing a command from the NC system (that is, while the axis is moving).

17.3.2 Circular Feed in Manual Mode

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

By specifying a hypothetical coordinate on the machine coordinate from the user PLC, oblique linear interpolation or circular interpolation is executed with jog/handle feed, manual rapid traverse or incremental feed of either X-axis or Y-axis.
This function is valid only in the jog mode, handle mode, manual rapid traverse mode or incremental mode.
This function cannot be used in the other manual modes and automatic operation modes.
This function works for the X axis and Y axis. This function cannot be used for the other NC axes and PLC axis. When this function is valid, all the axes other than X axis and Y axis move as usual.

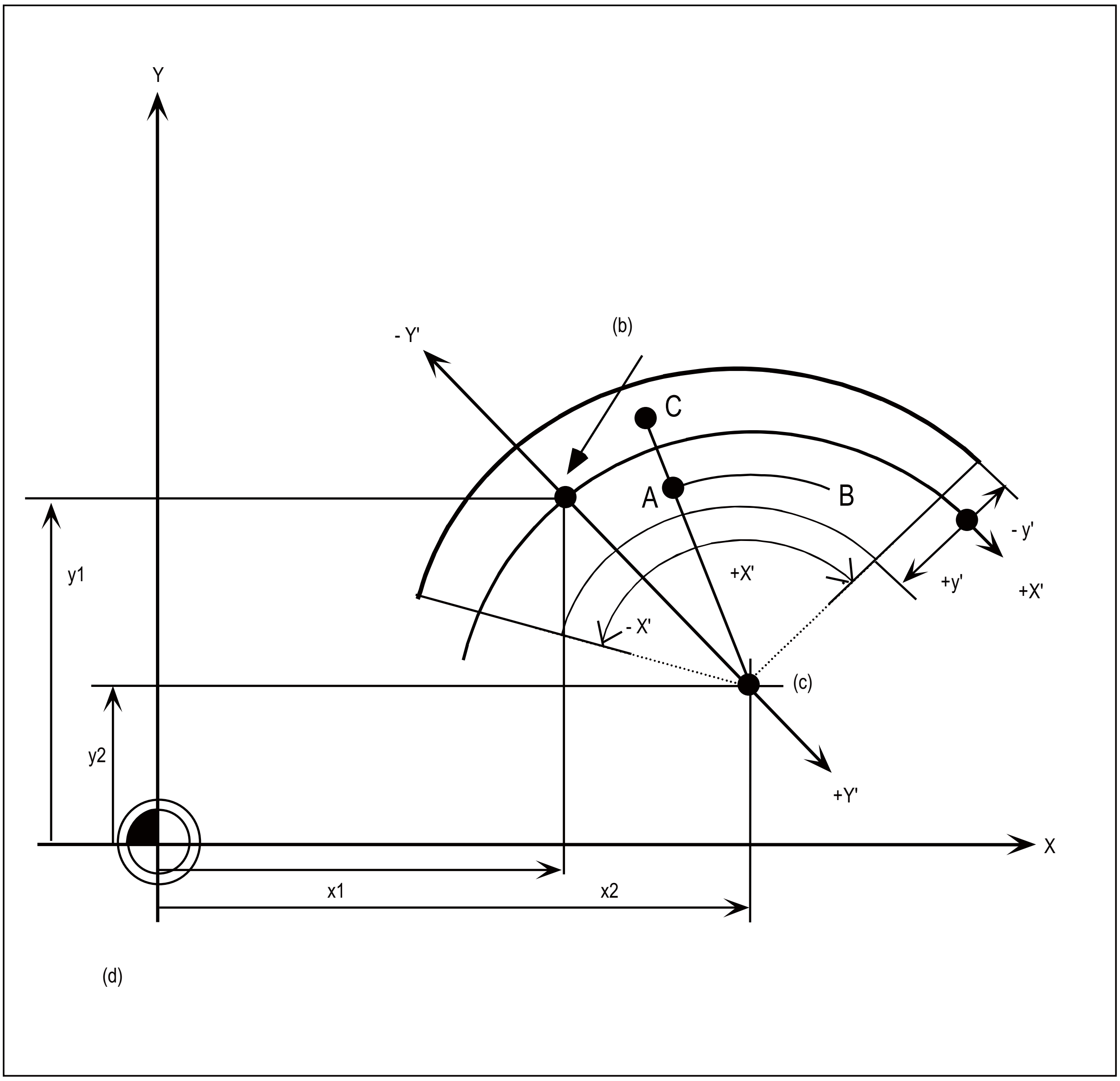
(1) Oblique linear interpolation



These are set in the R registers.
a : Gradient angle
x,y : Basic point coordinate
±x' : X' travel range
±y' : Y' travel range
(b) Basic point
(c) When the circular feed in manual mode (oblique linear interpolation) is valid:
- Jog/handle feed of X-axis moves in parallel with X'-axis (see above).
- Jog/handle feed of Y-axis moves in parallel with Y'-axis (see above).

(2) Circular interpolation

By specifying a hypothetical coordinate on the machine coordinate as shown in the figure below, jog/handle feed can be executed on the hypothetical coordinate.



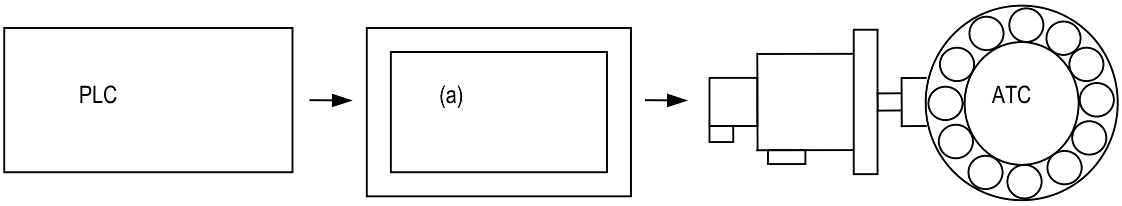
These are set in the R registers.
x1, y1 : Basic point coordinate
x2, y2 : Arc center coordinate
±x' : X' travel range
±y' : Y' travel range
(b) Basic point
(c) Arc center
(d) When the circular feed in manual mode (circular interpolation) is valid:

- Jog/handle feed of X-axis moves on the arc having the distance from the arc center to the current position as its radius.
- Jog/handle feed of Y-axis moves on the line connecting the current position and the arc center.

17.3.3 PLC Axis Control

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.



(a) PLC axis control

Item	Description
Number of control axes	Max. 2 axes
Simultaneously controlled axes	PLC control axis is controlled independently from NC control axes. A multiple number of PLC axes can be started simultaneously.
Command increment	Least command increment 0.001 mm (0.0001 inch) 0.0001 mm (0.00001 inch) 0.00001 mm (0.000001 inch) 0.000001 mm (0.0000001 inch) (Same as command increment for NC control axes)
Feed rate	0 to 1000000mm/min (0 ～ 100000 inch/min) (Speed is fixed regardless of the unit system.)
Movement commands	Incremental commands from current position Absolute commands for machine coordinate system 0 to ± 99999999 (0.001mm/0.0001inch)
Operation modes	Rapid traverse, cutting feed, jog feed (+) (-), reference position return feed (+) (-), handle feed
Acceleration/deceleration	Rapid traverse, jog feed Reference position return feed } Linear acceleration/deceleration Cutting feed } Exponential function acceleration/deceleration Handle feed } Step
Backlash compensation	Available
Stroke end	None
Soft limit	Available
Rotary axis command	Available For absolute commands……amount within 1 rotation (rotation by amount remaining after division into 360) For incremental commands……rotation by assigned amount
Inch/mm changeover	None Set to the command that corresponds to the feedback unit.
Position detector	Encoder (Absolute position can also be detected.)

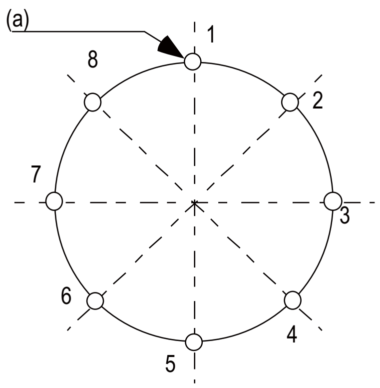
17.3.5 PLC Axis Indexing

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function is used to move the PLC axis to the position (station) registered on the table.
The destination (station) can be determined by equally dividing the valid axis stroke or by using an arbitrary coordinate that has been stored in advance. The PLC program can also be used to specify arbitrary values of the coordinate.

(1) Command methods

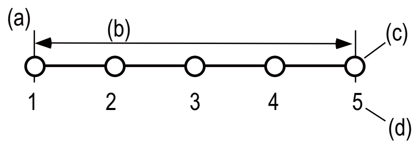
Station method (for rotary axis) : One rotation (360°)of the rotary axis is equally divided to determine the stations (positioning destinations). The maximum number of divisions is 360.



(a) Station

Figure 1. Setting 8 stations (8 divisions)

Station method (for linear axis) : A valid stroke is equally divided to determine the stations (positioning destinations). The maximum number of divisions is 359.



(a) Zero point

(b) Valid stroke length
(c) Station
(d) Station No.

Figure 2. Setting 5 stations

The zero point is station 1, and the final end of the valid stroke is station 5.
When using a linear axis, the No. of equal divisions is "number of stations - 1".

(2) Feed functions

Feed rate selection : Four different feed rates are set to be selected with the PLC program.

Acceleration/deceleration pattern : Four different acceleration/deceleration patterns are set to be selected with the PLC program. The patterns are all constant inclination acceleration/deceleration, which have options of linear or soft acceleration/deceleration.

Short-cut control : A least movement distance is automatically judged when a rotary axis is rotated.

(3) Operation functions

The following operation modes are available. Send a command from PLC to change the operation mode.

Automatic mode	:	The axis is positioned at the station No. designated by the start signal. When the start signal has turned OFF before the positioning is completed, the axis is positioned at the nearest station position. The arbitrary position command operation is also available: the positioning can be carried out to any position other than a station by directly commanding the positioning coordinates from the PLC.
Manual mode	:	While the start signal is ON, the axis is rotated at a constant speed in the designated direction. When the start signal has turned OFF, the axis is positioned at the nearest station position.
JOG mode	:	While the start signal is ON, the axis is rotated at a constant speed in the designated direction.
Manual handle mode	:	The axis is moved by the manual PLG.

(4) Operation support function

Position switch	:	A signal is output to the PLC interface when the machine has reached within the specified range.

17.4 PLC Interface

17.4.1 CNC Control Signal

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Control commands to the CNC system are assigned from the PLC. Input signals with an A/D conversion function and skip inputs that respond at high speed can also be used.

(1) Control signals

- Control signals for operations in automatic operation mode
- Control signals for operations in manual operation mode
- Control signals for program execution
- Control signals for interrupt operations
- Control signals for servo
- Control signals for spindle
- Control signals for mode selection
- Control signals for axis selection
- Control signals for feed rates

(2) Analog voltage control [M system]

When an analog voltage is input to an external connector used to connect CNC analog inputs, the data corresponding to the input voltage can be read out in the prescribed file register. This data can be used for load meter displays, thermal deformation compensation, etc. (Maximum 4 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 8 points)

For further details, refer to the PLC Interface Manual.

17.4.2 CNC Status Signal

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC. These signals can also be output as analog data by setting the data from the PLC in the R register.

Status output functions

(1) Controller operation ready
When the controller power is turned ON and the controller enters the operation ready status, the "Ready" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the controller power is supplied to when the controller ready status is entered.

(2) Servo operation ready
When the controller power is turned ON and the servo system enters the operation ready status, the "Servo ready" signal is output to the machine.
Refer to the PLC Interface Manual for details of the sequences from when the power is supplied to when the "Servo ready" signal is turned ON.

(3) In automatic operation
Generally, if the "cycle start" switch is turned ON in the automatic operation mode (memory, MDI), this signal is output until the reset state or emergency stop state is entered by the M02, M30 execution or the reset & rewind input to the controller using the reset button.

(4) In automatic start
The signal that denotes that the controller is operating in the automatic mode is output from the time when the cycle start button is pressed in the memory or MDI mode and the automatic start status has been entered until the time when the automatic operation is terminated in the automatic operation pause status entered by the "feed hold" function, block completion stop entered by the block stop function or resetting.

(5) In automatic pause
An automatic operation pause occurs and this signal is output during automatic operation from when the automatic pause switch is pressed ON until the automatic start switch is pressed ON, or during automatic operation when the mode select switch is changed from the automatic mode to the manual mode.

(6) In rapid traverse
The "In rapid traverse" signal is output when the command now being executed is moving an axis by rapid traverse during automatic operation.

(7) In cutting feed
The "In cutting feed" signal is output when the command now being executed is moving an axis by cutting feed during automatic operation.

(8) In tapping

The "In tapping" signal is output when the command now being executed is in a tapping modal which means that one of the statuses below is entered during automatic operation.

(a) G84, G88 (fixed cycle: tapping cycle)
(b) G84.1, G88.1 (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

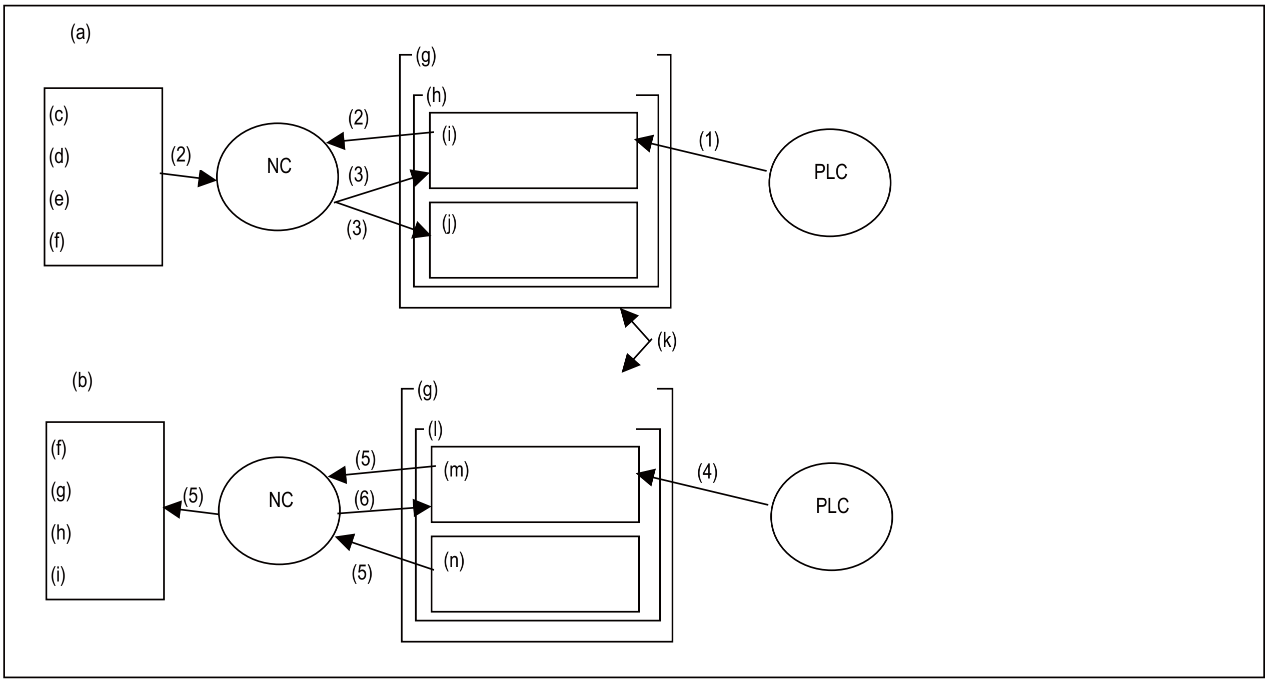
In the memory or MDI automatic operation, the "Movement command finish" signal is output when the command block in the machining program features a movement command and when that block command has been completed.
When the movement command and M, S, T or B command have been assigned in the same block, 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.

17.4.3 PLC Window

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

This function uses the "read window" or "write window" assigned to the R register's user area to read and write the CNC operation status, axis information, parameters and tool data, etc.

The area used for the "read window" and "write window" has 1500 points for the backup area and 100 points for the non-backup area.
Up to three window areas can be specified simultaneously for both "read window" area and "write window" area. 16 R registers are used for one read window or write window.
The read window is further divided into the "read control window" and "read data window". The write window is also divided into the "write control window" and "write data window".
Up to four data items can be successively read or written from the data designated in one read window or write window.



(a) <Reading>
(b) <Writing>
(c) Operation status
(d) Axis information
(e) Parameters
(f) Tool data
(g) R register (user area)
(h) Read window
(i) Read control window
(j) Read data window
(k) Designate arbitrary position with parameter (R register)
(l) Write window
(m) Write control window
(n) Write data window
(1) PLC turns control signal ON.
(2) NC receives control signal, and reads designated data in control window.
(3) Results of NC read are set in read control window and read data window.
(4) PLC turns control signal ON.
(5) NC receives control signal, and writes contents of write data window corresponding to designated data in control window.
(6) Results of NC write are set in write control window.

17.4.4 External Search

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

The program which is automatically operated can be searched from PLC. The program No., sequence No. and block No. can be assigned. Also the content currently searched can be read.

17.5 Machine Contact I/O

17.5.1 Additional DI/DO (DI: 32/DO: 32)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

17.5.2 Additional DI/DO (DI: 64/DO: 64)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

17.5.3 Remote IO 32/32

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

17.5.4 Remote IO 64/48

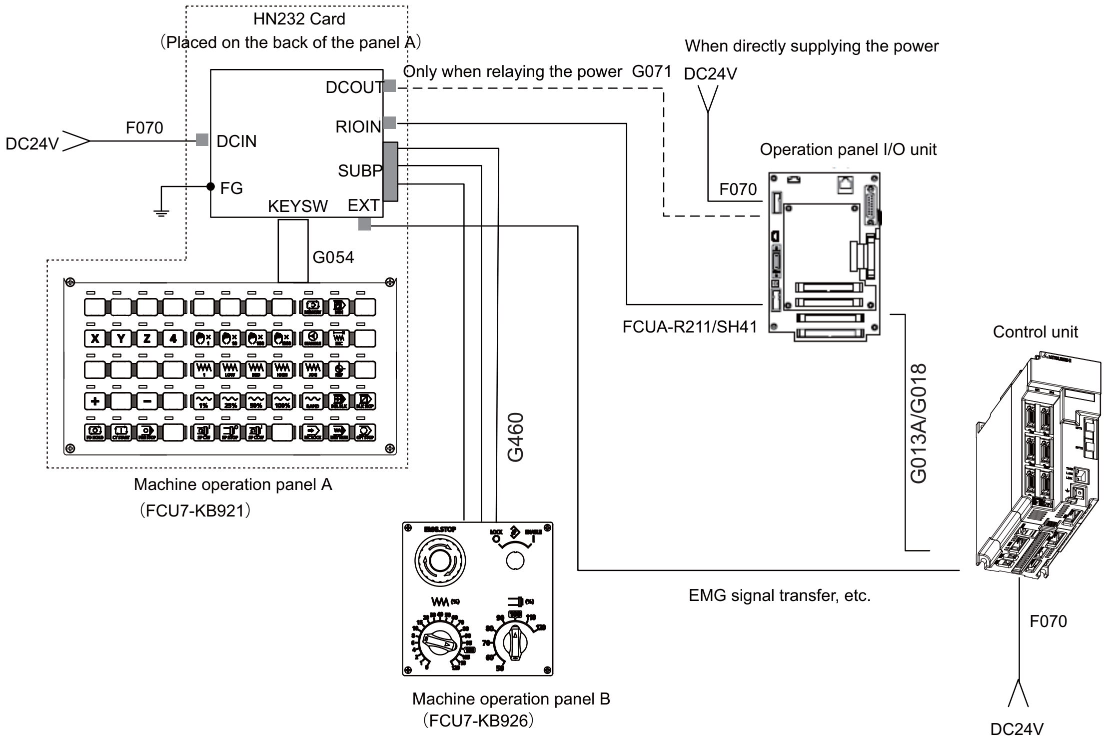
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□	□	□	□	□	□
L system	□	□	□	□	□	□

17.5.5 MITSUBISHI CNC Machine Operation Panel

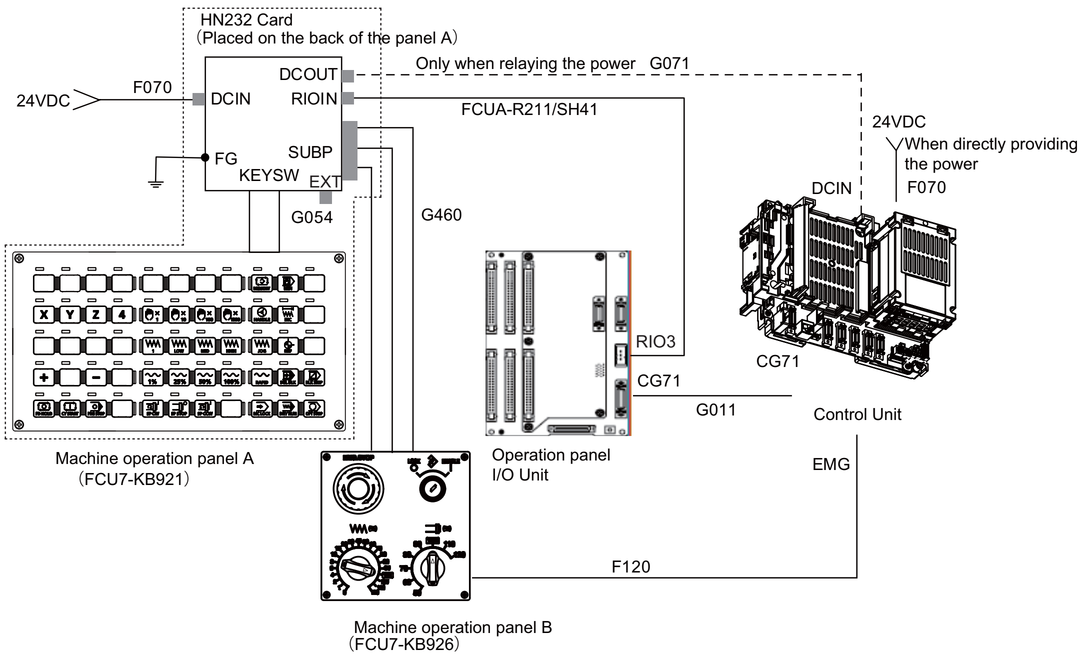
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

MITSUBISHI CNC machine operation panel can change its keyboard layout according to the specifications of the machine.
Also, a sample of the PLC program supporting the basic key board layout is available. By adding an interface to the machine, a PLC program which fits the machine can easily be created.
Wirings have been reduced as a result of directly connecting with the NC via the remote I/O link communication.

<Connection of M700VW>



<Connection of M700VS>



17.6 External PLC Link

17.6.3 CC-Link (Master/Slave)

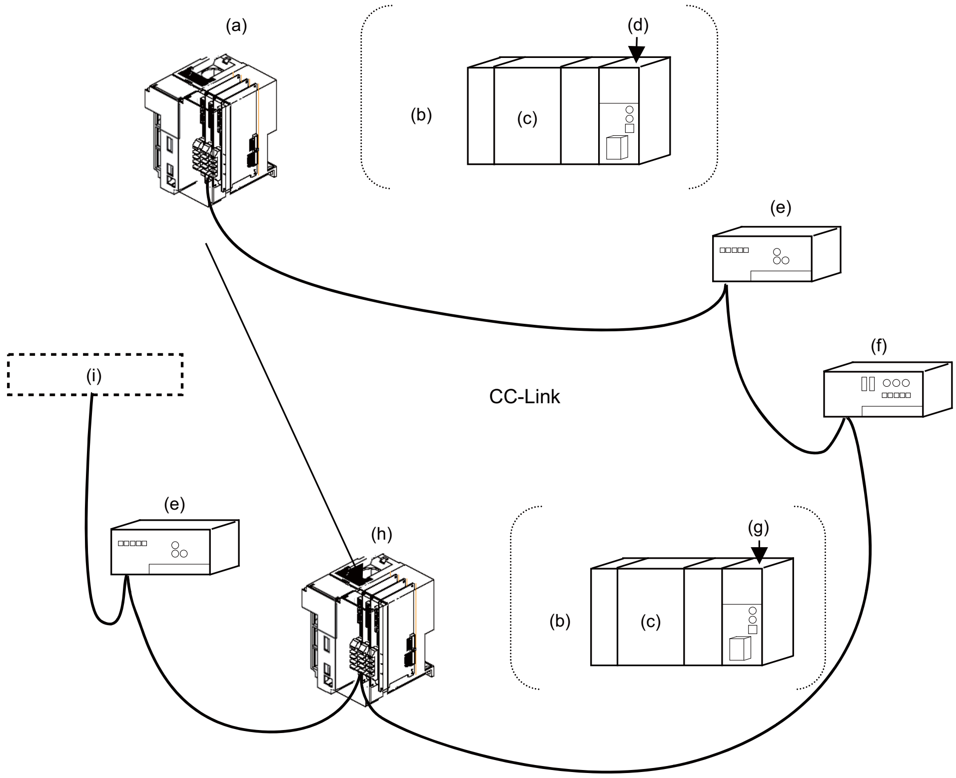
	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□ (when HN576/ HN577 is installed)	□ (when HN576/ HN577 is installed)	□ (when HN576/ HN577 is installed)	□ (when HN746 is installed)	□ (when HN746 is installed)	□ (when HN746 is installed)
L system	□ (when HN576/ HN577 is installed)	□ (when HN576/ HN577 is installed)	□ (when HN576/ HN577 is installed)	□ (when HN746 is installed)	□ (when HN746 is installed)	□ (when HN746 is installed)

NC unit can be directly connected to the network to serve as the master/local station of the MELSEC CC-Link.
CC-Link unit is required for this connection.
With this function, the GOT connection and the transient instruction with MELSEC A series cannot be used.

(1) Outline of CC-Link

- Distributing and installing each unit to the equipments such as conveyor line and mechanical device can simplify the wiring of the whole system.
- The ON/OFF data and numerical data such as input/output treated by each unit can be communicated easily and at high speed.
- The simple distribution system can be established by connecting several sequencer CPUs or NCs.
- Connecting the device equipments made by the partner manufacturer can flexibly support various systems.

(2) Outline drawing



(a) Master station (NC unit)
(b) or
(c) Sequencer CPU
(d) Master station
(e) Remote I/O station
(f) Remote device station
(g) Local station
(h) Local station (NC unit)
(i) Partner manufacturer product

Master station	This station controls the remote station and local station. One master station is required for one system.
Local station	This station contains the CPU and can communicate with the master and the other local stations.
Remote I/O station	Remote station that handles only bit information.
Remote device station	Remote station that handles bit information and word information.
Intelligent device station	This station allows the transient transmission. (Including local station)

(3) Performance specifications
M700V Series is compliant with CC-Link Ver.2.00.
<CC-Link Ver.2.00 specification>

Item		CC-Link master/local unit				
Max. number of link points per one system (Note 1)		Remote input/output (RX,RY)	8192 points each			
		Remote register (RWw) (Master St. → Remote/Local St.)	2048 points			
		Remote register (RWr) (Remote/Local St. → Master St.)	2048 points			
Extended cyclic setting			1 time setting	2 times setting	4 times setting	8 times setting
Number of link points per one remote/local station	1 station occupied	Remote input/output (RX,RY)	32 points each	32 points each	64 points each	128 points each
		Remote register (RWw) (Master St. → Remote/Local St.)	4 words	8 words	16 words	32 words
		Remote register (RWr) (Remote/Local St. → Master St.)	4 words	8 words	16 words	32 words
	2 stations occupied	Remote input/output (RX,RY)	64 points each	96 points each	192 points each	384 points each
		Remote register (RWw) (Master St. → Remote/Local St.)	8 words	16 words	32 words	64 words
		Remote register (RWr) (Remote/Local St. → Master St.)	8 words	16 words	32 words	64 words
	3 stations occupied	Remote input/output (RX,RY)	96 points each	160 points each	320 points each	640 points each
		Remote register (RWw) (Master St. → Remote/Local St.)	12 words	24 words	48 words	96 words
		Remote register (RWr) (Remote/Local St. → Master St.)	12 words	24 words	48 words	96 words
	4 stations occupied	Remote input/output (RX,RY)	128 points each	224 points each	448 points each	896 points each
		Remote register (RWw) (Master St. → Remote/Local St.)	16 words	32 words	64 words	128 words
		Remote register (RWr) (Remote/Local St. → Master St.)	16 words	32 words	64 words	128 words
Number of occupied stations (Number of local stations)		Station 1 to station 4 (The station No. can be switched with parameters.)				
Baud rates		156kbps / 625kbps / 2.5Mbps / 5Mbps / 10Mbps can be selected. (Baud rate can be selected with parameters.)				
Communication method		Polling method				
Synchronization method		Flame synchronization method				
Encode method		NRZI method				
Transmission path method		Bus (EIA RS485 compliant)				
Transmission format		HDLC compliant				
Illegal control method		CRC($X^{16} + X^{12} + X^5 + 1$)				

Item	CC-Link master/local unit
Max. number of connection units	64 units Note that the following two conditions must be satisfied. Condition 1: - $(a+a2+a4+a8)+(b+b2+b4+b8) \times 2+(c+c2+c4+c8) \times 3+(d+d2+d4+d8) \times 4 \leqq 64$ - $(a \times 32+a2 \times 32+a4 \times 64+a8 \times 128)+(b \times 64+b2 \times 96+b4 \times 192+b8 \times 384)+(c \times 96+c2 \times 160+c4 \times 320+c8 \times 640)+(d \times 128+d2 \times 224+d4 \times 448+d8 \times 896) \leqq 8192$ - $(a \times 4+a2 \times 8+a4 \times 16+a8 \times 32)+(b \times 8+b2 \times 16+b4 \times 32+b8 \times 64)+(c \times 12+c2 \times 24+c4 \times 48+c8 \times 96)+(d \times 16+d2 \times 32+d4 \times 64+d8 \times 128) \leqq 2048$ a : Number of units when 1 station is occupied in 1 time extended cyclic setting b : Number of units when 2 stations are occupied in 1 time extended cyclic setting c : Number of units when 3 stations are occupied in 1 time extended cyclic setting d : Number of units when 4 stations are occupied in 1 time extended cyclic setting a2: Number of units when 1 station is occupied in 2 times extended cyclic setting b2: Number of units when 2 stations are occupied in 2 times extended cyclic setting c2: Number of units when 3 stations are occupied in 2 times extended cyclic setting d2: Number of units when 4 stations are occupied in 2 times extended cyclic setting a4: Number of units when 1 station is occupied in 4 times extended cyclic setting b4: Number of units when 2 stations are occupied in 4 times extended cyclic setting c4: Number of units when 3 stations are occupied in 4 times extended cyclic setting d4: Number of units when 4 stations are occupied in 4 times extended cyclic setting a8: Number of units when 1 station is occupied in 8 times extended cyclic setting b8: Number of units when 2 stations are occupied in 8 times extended cyclic setting c8: Number of units when 3 stations are occupied in 8 times extended cyclic setting d8: Number of units when 4 stations are occupied in 8 times extended cyclic setting Condition 2: - $\{(16 \times A)+(54 \times B)+(88 \times C)\} \leqq 2304$ - A:Number of remote I/O stations $\leqq$ 64 units - B:Number of remote device stations $\leqq$ 42 units - C:Number of local stations, standby master stations and intelligent device stations $\leqq$ 26 units
Remote station No.	1 to 64
Max. total cable length and each cable length between stations	CC-Link Ver.1.10 compliant cable (When a 110Ω of terminating resistance is used.) (Note 2) Baud rates / Cable length between stations / Max. total cable length 156kbps / 20cm or more / 1200m 625kbps / 20cm or more / 900m 2.5Mbps / 20cm or more / 400m 5Mbps / 20cm or more / 160m 10Mbps / 20cm or more / 100m
Connection cable	CC-Link Ver.1.10 compliant cable (3-core twisted pair cable with shield) (Note 3)
RAS function (Note 4)	- Automatic link refresh function - Sub-station isolation function - Link special relay/error detection by register

(Note 1) If the points which can be reserved as the device for the CC-Link in the NC side does not reach 8192 points, the number of points which can be reserved in the NC side is the max. number of link points per one system.
This applies for the remote register (RWw, RWr), as well.

(Note 2) When CC-Link Ver.1.00 compliant cables are mixed, the cable length between stations and the maximum total length of the cable should follow the specifications for CC-Link Ver.1.00.

(Note 3) This can be used with the cables made by different manufacturers as long as they are Ver.1.10 compliant cables.

(Note 4) "RAS" is short for "Reliability", "Availability" and "Serviceability".

(4) Usable functions

Of all the CC-Link functions, the following functions can be used with the NC.

(○ : Available ×: Not available － : Not relevant)

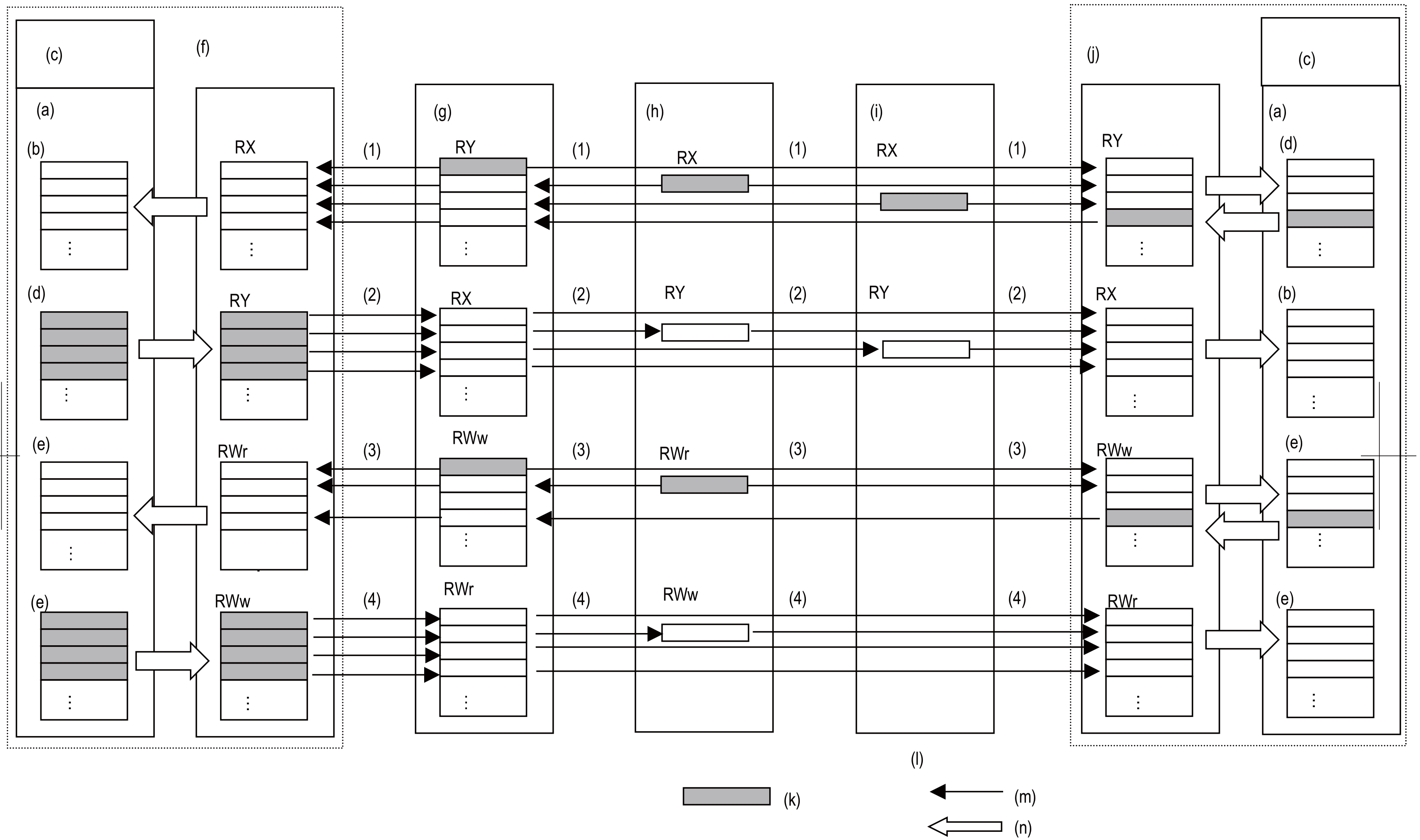
	Function item		MELSEC		NC unit	
			Master station	Local station	Master station	Local station
Method	Ver.1.00		○	○	○	○
	Ver.2.00		○	○	○	○
Master function	Communication between master station and remote I/O station		○	－	○	－
	Communication between master station and remote device station		○	－	○	－
	Communication 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 E^2PROM		○	－	○ (Note 2)	－
	Setting of input data status from data link trouble station		○	○	○	○
	Unit resetting by sequence program		○	○	○	○
	Data link stop/restart		○	○	○	○
	Parameter registration function		○	○	○ (Note 2)	○ (Note 2)
	Automatic refresh function		○	○	○	○
	Scan synchronization function	Synchronous mode	○	－	○	－
		Asynchronous mode	○	○	○	○
Setting & display function	LED diagnosis status		16-point display (A1SJ61QBT11)		16-point display	
	Station number setting		Unit front panel switches		Parameters (Note 2)	
	Baud rate setting					
	Mode setting switch					
	Condition setting				H/W switches	
RAS function	Automatic link refresh function		○	○	○	○
	Sub-station isolation function		○	－	○	－
	Data link status check (SB/SW)		○	○	○	○
	Off-line test		○	○	○	○
	On-line test		○	○	○	○
	Monitor diagnosis		○	○	×	×
	Standby master function (Note 3)		○	－	○	－
	Temporary error cancel station designation function		○	－	○	－
Dedicated instruction	READ instruction / SREAD instruction (Note 1)		○	○	○	○
	WRITE instruction / SWRITE instruction (Note 1)		○	○	○	○
	RIRD instruction / RIWT instruction (Note 1)		○	○	×	×

(Note 1) The transient instruction cannot be used.

(Note 2) The parameter for the CC-Link is set with the GX Developer.

(Note 3) With this function, when an error occurs in the master station, switch to the standby master station so that data link can be continued. (The NC unit cannot be used as the standby master station.)

(5) Communication data flow
The flow of data communicated by the CC-Link's link scan is as follows.
(The master station and local station of MELSEC CPU can be also mixed.)



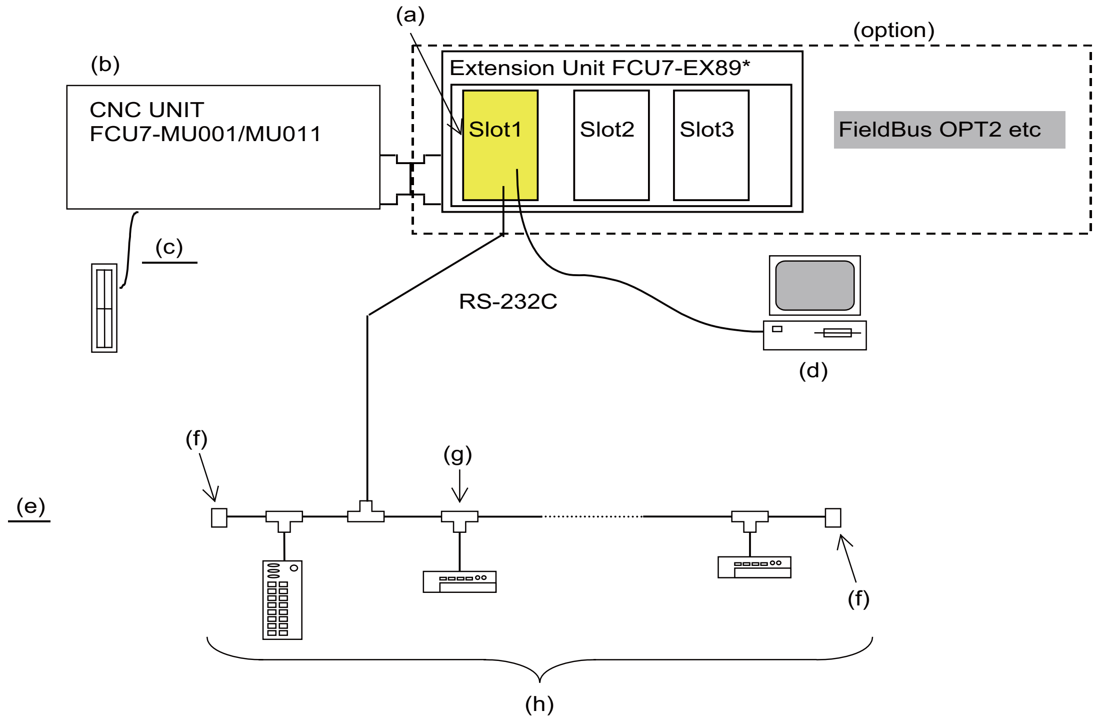
(a) Devices
(b) X device, etc.
(c) Built-in PLC
(d) Y device, etc.
(e) R register, etc.
(f) NC (Master station)
(g) Local station
(h) Remote device station
(i) Remote I/O station
(j) NC (Local station)
(k) Transmission data
(l) <Flow of data>
(m) Risk scan
(n) Automatic refresh (When master station/local station is NC.)

(1) By executing a link scan, data in the remote I/O station and remote device station's remote input (RX) and in the local station's remote output (RY) is transmitted to the master station's remote input (RX) and the local station's remote output (RY).
(2) By executing a link scan, data in the master station's remote output (RY) is transmitted to the remote I/O station and remote device station's remote output (RY) and the local station's remote input (RX).
(3) By executing a link scan, data in the remote device station's remote register (RWr) and the local station's remote register (RWw) is transmitted to the master station's remote register (RWr) and the local station's remote register (RWw).
(4) By executing a link scan, data in the master station's remote register (RWw) is transmitted to the remote device station's remote register (RWw) and the local station's remote register (RWr).

17.6.4 PROFIBUS-DP (Master)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	□ (when HN571 is installed)	□ (when HN571 is installed)	□ (when HN571 is installed)	—	—	—
L system	□ (when HN571 is installed)	□ (when HN571 is installed)	□ (when HN571 is installed)	—	—	—

By installing HN571 slave stations compatible with PROFIBUS-DP communication can be connected to input/output device.
Hilsher's Fieldbus communication control unit (COM module) is mounted on HN571, and NC operates as the master station. Up to 128 units can be connected with master station and slave stations combined.
Inputs/outputs of the devices from NC's PLC are all handled as bit device data. Up to 512 points can be input/output.
Maximum number of inputs/outputs for NC remote I/O unit is 768, whether or not HN571 is installed.



(i)

(a) HN571 for master station communication
(b) CNC control unit
(c) RIO communication
(d) Windows PC for parameter setting + Hilsher configurator
(e) Fieldbus communication
(f) Terminator
(g) Tapping
(h) 128 units (master + slave)
(i) Fieldbus connection outline

Parameters for the communication control unit installed in HN571 must be set with Hilsher configurator.
Refer to the Hilsher instruction manual.
"Bus synchronous" or "device controlled" of the hand-shake method cannot be used. If used, an error will occur.

17.6.5 DeviceNet (Master)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	□ (when HN747 is installed)	□ (when HN747 is installed)	□ (when HN747 is installed)
L system	—	—	—	□ (when HN747 is installed)	□ (when HN747 is installed)	□ (when HN747 is installed)

MELSEC-Q series I/O intelligent function unit can be connected with NC.
To enable this connection, the MELSEC-Q interface expansion unit (FCU7-HN747) must be installed.
Input/output signal status for each unit is referable and user-configurable, using CNC's PLC device.
The intelligent function unit enables MELSEC-Q series to communicate with field network devices, using PLC exclusive instruction.

17.6.6 FL-net

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	—	—	—	□ (when HN747 is installed)	□ (when HN747 is installed)	□ (when HN747 is installed)
L system	—	—	—	□ (when HN747 is installed)	□ (when HN747 is installed)	□ (when HN747 is installed)

MELSEC-Q series I/O intelligent function unit can be connected with NC.
To enable this connection, the MELSEC-Q interface expansion unit (FCU7-HN747) must be installed.
Input/output signal status for each unit is referable and user-configurable, using CNC's PLC device.
The intelligent function unit enables MELSEC-Q series to communicate with field network devices, using PLC exclusive instruction.

17.7 Installing S/W for Machine Tools

17.7.1 Customization (NC Designer)

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

Custom release is an optional function and allows the user-original window to display as an HMI screen or another application.

(Note) The separate software is required.

Custom release includes, mainly, F0 release and menu release:

F0 release : Custom release screen (Note 1) can be assigned to function keys (F0, SEP, window display, window selection).

When a function key is pressed, the assigned custom release screen will be displayed.

"NC Designer interpreter method", "NCDesigner compilation method" and "Executing file registration method" can be used for registration.

Menu release : Custom release window (Note 2) can be registered in the main menu of the monitor screen, setup screen and edit screen.

Main menu contents of the monitor, setup and edit screen can be rearranged.

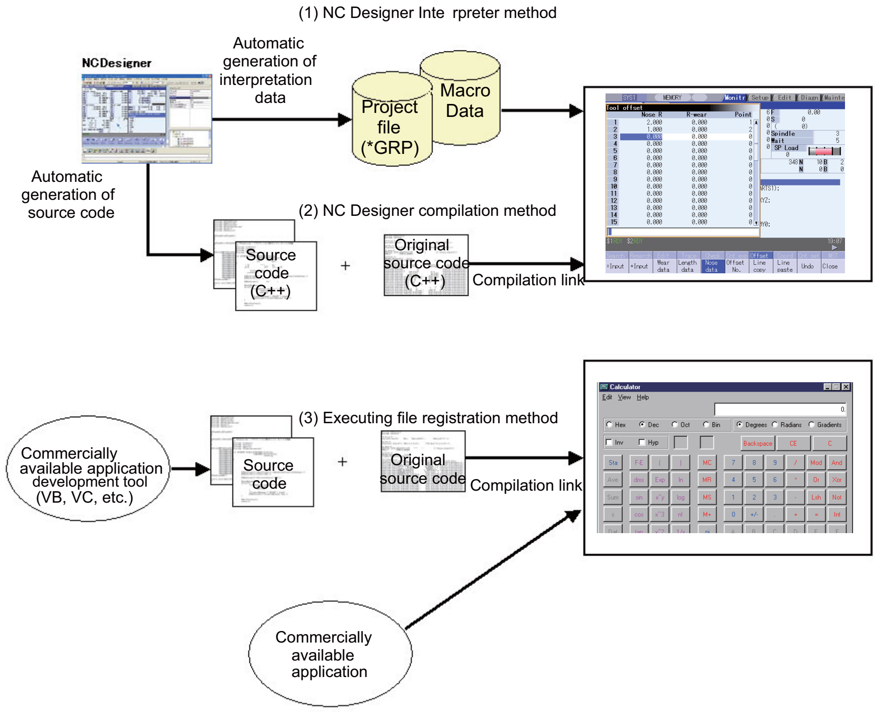
When the main menu in which custom release window is registered is pressed, the custom release window will be displayed.

"NC Designer interpreter method", "NC Designer compilation method" and "Executing file registration method" can be used for registration.

Depending on the conditions, display/non-display of the custom menu can be changed.

(Note 1) An HMI screen originally created with NC Designer by the user or an execution file prepared by the user.

(Note 2) An HMI window originally created with NC Designer by the user or an execution file prepared by the user.



17.7.1.1 Customization Data Storage Capacity [MByte]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	Depending on hard disk space	Depending on hard disk space	Depending on hard disk space	6	6	6
L system	Depending on hard disk space	Depending on hard disk space	Depending on hard disk space	6	6	6

17.7.1.2 Customization Workpiece Data Size [MByte]

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	6	6	6	6	6	6
L system	6	6	6	6	6	6

17.7.2 User-defined Key

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

This function allows an arbitrary character string to be assigned to one key and makes it easy to input the fixed phrases.
Register correspondences between SHIFT keys and character strings in a configuration file (keydef.txt).
The maximum number of registerable key types is 8 for ABC layout and 10 for QWERTY layout.
The maximum number of characters used per definition is 256. The definition is disregarded when it is set exceeding the number.

In the configuration file (Keydef.txt), a character string should be defined as follows:
n [character string] ;
n(Registration No.): 0 to 7 (SHIFT+A to SHIFT+H (ABC layout))
: 0 to 9 (SHIFT+Q to SHIFT+G (QWERTY layout))
Example:
0[G28X0Y0]; ← When SHIFT+A and a key is input, a character string "G28X0Y0" is input.
1[G28Z0]; ← When SHIFT+B and a key is input, a character string "G28Z0" is input.

17.7.3 EZSocket I/F

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(Note) The separate software is required.
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.

17.7.4 APLC Release

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

(Note) The separate software is required.
APLC (Advanced Programmable Logic Controller) release is a function that allows the user-generated C language module to be called from NC.
Control operations that are difficult to express in ladder language can be created with C language.
PLC ladder can easily be customized by converting a part of PLC ladder that has grown complex and bulky into C language module.

There are three methods where APLC is executed. It is possible to use the three in combination.

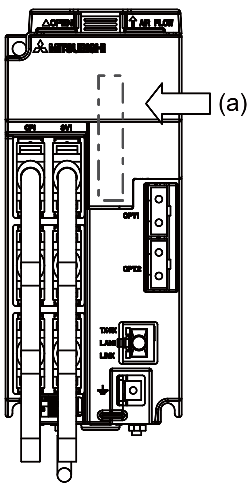
- Type 1 : By registering the start address of the module to be processed asynchronously with PLC, C language module will be executed in a constant frequency. The processing frequency is 7.1 (ms).

- Type 2 : By registering the start address of the module to be processed synchronously with PLC, C language module will be started synchronizing with PLC. Note that C language will not be called if PLC is incorrect, even if the process has been registered.
- Type 3: By registering the start address of the module to be called from the sequence program and processed, C language module will be started from the sequence program upon S.CALL instruction.

[Hardware configuration]

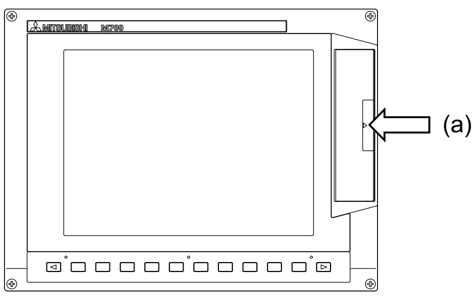
This function will be activated by installing C language module into a compact flash or built-in F-ROM. Compact flash interface is located on the control unit. Up to 120KB data can be stored in built-in F-ROM. When C language module is installed in compact flash, C language inside the compact flash is executed. (The module inside the built-in F-ROM is not executed.)

There is a list of compact flash cards which are tested by us in "5 Hardware Specifications" of "Precautions for Use of Commercially Available CF Cards".



700VW

(a) Compact flash interface



700VS

(a) Compact flash interface

[Software configuration]

The names of directory, file and initialize function, where C language modules are stored, are fixed.

(Note) Incorrect hardware or software configuration disables operations regardless of option function.

17.7.5 Custom API Library

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(Note) The separate software is required.

Reading/writing of each information within NC unit is possible by using custom API library.

17.8 Others

17.8.1 System Lock

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	△	△	△	△	△	△
L system	△	△	△	△	△	△

With this function, when release code is not input before the specified deadline, SA (servo ready completed) is forcibly set to OFF.
(Note) The detriment cannot be guaranteed if the release code should be decoded.

17.8.2 CNC Remote Operation Tool

17.8.2.1 NC Monitor

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(Note) The separate software is required.
NC Monitor is a PC compatible software tool that monitors information in NC unit connected with the Ethernet.

17.8.2.2 NC Explorer

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

(Note) The separate software is required.
NC Explorer is a software tool to operate the machining data files of each NC unit connected with a host personal computer by Ethernet connection from the Explorer on the host personal computer.

17.8.3 Automatic Operation Lock

	M720VW	M730VW	M750VW	M720VS	M730VS	M750VS
M system	○	○	○	○	○	○
L system	○	○	○	○	○	○

Automatic operation lock function prevents the falsification of APLC(C language module: herein after called APLC) by a third party.
Automatic operation will be prohibited when illegal APLC is installed by authenticating APLC with the password for APLC authentication which is registered to NC unit beforehand using this function.
Refer to "17.7.4 APLC Release" for details on the APLC.

Revision History

Date of revision	Manual No.	Revision details
Nov. 2008	IB(NA)1500916-A	First edition created.
Jan. 2009	IB(NA)1500916-B	The descriptions of M700VW Series were added to “General Specifications” The following sections were added to “Functional Specifications”. - 1.3.4.1 CF Card in Control Unit - 1.3.6 Hard Disk Mode - 6.1.3 Color display (10.4-type LCD TFT/WindowsXPe) - 6.1.5 Color touch-panel display (10.4-type LCD TFT/WIndowsXPe) - 6.1.7 Color touch-panel display (15-type LCD TFT/WindowsXPe) - 7.2.2.1 I/F for CF Card in Control Unit - 7.2.4 Hard Disk I/F - 7.2.5 Floppy Disk I/F - 7.4.1 Handy Terminal Connection - 11.2.4.2 3D Solid Program Check - 15.3.8 3D Machine Interference Check - 16.1.3.1 Index/Positioning Servo : MR-J2-CT - 16.1.3.1.1 Servo motor : HC-SF/HC-RF (16 kp/rev) - 16.1.3.1.2 Servo motor : HA-FF/HC-MF (8 kp/rev) - 17.2.8 Auxiliary Axis Control (J2-CT) - 17.2.13 Inclined Surface Machining Command - 17.6.4 PROFIBUS-DP (Master) - 17.7.1.1 Customization data storage capacity [MByte] - 17.7.1.2 Customization workpiece data size [MByte] - 17.8.2.1 NC Monitoring Tool - 17.8.2.2 NC Explorer Other contents were added/revised/deleted according to specification.
Apr.2010	IB(NA)1500916-C	Revisions were made to support M700V S/W version G1. The following chapters were added. - 6.1.7 Color touch-panel display (15-type LCD TFT/WindowsXPe) - 6.2.16 User selectable menu configuration - 6.2.17 PC-NC network automatic connection - 6.2.18 Device open parameter - 6.2.19 SRAM open parameter - 8.1.11 Spindle speed clamp - 9.3.1.7 Number of tool compensation sets (99 x number of part systems) sets - 12.1.8.4 Control axis superimposition - 14.2.1.3 Tool life management III - 16.1.1.1.6 Servo Motor: HF-KP □□ JW04(260kp/rev) - 16.1.1.4 MDS-DM-V3/SPV2/SPV3(200V) - 16.1.1.4.1 Servo Motor: HF □□ -A48(260kp/rev) - 16.1.1.4.2 Servo Motor: HF □□ -A51(1000kp/rev) - 16.1.1.4.3 Servo Motor: HF-KP □□ JW04-S6(260kp/rev) - 16.1.2.4 MDS-D-SP2(200V) - 16.1.2.5 MDS-DM-SPV2/SPV3(200V) - 17.2.13 Inclined surface machining command - 17.2.14 3-dimensional tool radius compensation for 5-axis machining - 17.2.15 Workpiece installation error compensation - 17.5.5 MITSUBISHI CNC machine operation panel Other contents were added/revised/deleted according to specification.
Nov.2011	IB(NA)1500916-D	Revisions were made to support M700V S/W version H0. The following chapters were added. - 11.2.4.3 Graphic Check Rotary Axis Drawing - 11.2.5.2 Graphic Trace Rotary Axis Drawing - 12.1.2.4.6 8000 Sets - 12.1.2.4.12 (7900+100 × Number of Part System)Sets - 12.3.10 Machining Condition Selection I(1st Part System Only) - 15.4.11 Application Error Detection - 17.1.4.2 Large PLC Capacity - 17.2.16 Manual Feed for 5-axis Machining (Continue to the next page)

		(Continued from the previous page) - 17.6.5 DeviceNet(Master) - 17.6.6 FL-net - 17.8.3 Automatic Operation Lock Other contents were added/revised/deleted according to specification.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)
Central Region Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Michigan Service Satellite
ALLEGAN, MICHIGAN 49010, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-269-673-4092

Ohio Service Satellite
LIMA, OHIO 45801, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
CLEVELAND, OHIO 44114, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

Minnesota Service Satellite
MINNEAPOLIS, MINNESOTA 55413, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650

West Region Service Center
5665 PLAZA DRIVE, CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-220-4796 / FAX: +1-714-229-3818

East Region Service Center
200 COTTONTAIL LANE SOMERSET, NEW JERSEY 08873, U.S.A.
TEL: +1-732-560-4500 / FAX: +1-732-560-4531

Pennsylvania Service Satellite
ERIE, PENNSYLVANIA 16510, U.S.A.
TEL: +1-814-897-7820 / FAX: +1-814-987-7820

Massachusetts Service Satellite
BOSTON, MASSACHUSETTS 02108, U.S.A.
TEL: +1-508-216-6104

South Region Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GEORGIA 30097, U.S.A.
TEL: +1-678-258-4500 / FAX: +1-678-258-4519

Texas Service Satellites
GRAPEVINE, TEXAS 76051, U.S.A.
TEL: +1-817-251-7468 / FAX: +1-817-416-5000
FRIENDSWOOD, TEXAS 77546, U.S.A.
TEL: +1-832-573-0787 / FAX: +1-678-573-8290

Florida Service Satellite
WEST MELBOURNE, FLORIDA 32904, U.S.A.
TEL: +1-321-610-4436 / FAX: +1-321-610-4437

Canada Region Service Center
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-475-7728 / FAX: +1-905-475-7935

Mexico City Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 / FAX: +52-55-9171-7649

Monterrey Service Satellite
MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 / FAX: +52-81-8365-4171

Brazil Region Service Center
ACESSO JOSE SARTORELLI, KM 2.1 CEP 18550-000, BOITUVA-SP, BRAZIL
TEL: +55-15-3363-9900 / FAX: +55-15-3363-9911

Brazil Service Satellites
PORTO ALEGRE AND CAXIAS DO SUL BRAZIL
TEL: +55-15-3363-9927
SANTA CATARINA AND PARANA STATES
TEL: +55-15-3363-9927

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V. (EUROPE FA CENTER)
GOTHAER STRASSE 10, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 / FAX: +49-2102-486-5910

Germany Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-3270-010 / FAX: +49-711-3270-0141

France Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France (Lyon) Service Satellite
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center
VIALE COLLEONI 7-PALAZZO SIRIO CENTRO DIREZIONALE COLLEONI,
20041 AGRATE BRIANZA MILANO ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

Italy (Padova) Service Satellite
VIA SAVELLI 24 - 35129 PADOVA ITALY
TEL: +39-039-60531-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-27-6100 / FAX: +44-1707-27-8992

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420
08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-630-4700 / FAX: +48-12-630-4727

Poland (Wroclaw) Service Center
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 / FAX: +48-71-333-77-53

Turkey Service Center
BAYRAKTAR BULVARI, NUTUK SOKAK NO.5, YUKARI DUDULLU
ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
TECHNOLOGICKA 374/6,708 00 OSTRAVA-PUSTKOVEC, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

Sweden Service Center
STRANDKULLEN, 718 91 FROVI, SWEDEN
TEL: +46-581-700-20 / FAX: +46-581-700-75

Bulgaria Service Center
4 A. LYAPCHEV BOUL., 1756 - SOFIA, BULGARIA
TEL: +359-2-8176000 / FAX: +359-2-9744061

Ukraine (Kharkov) Service Center
APTEKARSKIY LANE 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +38-57-732-7744 / FAX: +38-57-731-8721

Ukraine (Kiev) Service Center
4-B, M. RASKOVOYI STR., 02660 KIEV, UKRAINE
TEL: +38-044-494-3355 / FAX: +38-044-494-3366

Belarus Service Center
703, OKTYABRSKAYA STR., 16/5, 220030 MINSK, BELARUS
TEL: +375-17-210-4626 / FAX: +375-17-227-5830

South Africa Service Center
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625 SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

Denmark Service Center
KARETMAGERVEJ. 7A, DK-7000, FREDERICIA, DENMARK
TEL: +45-7620-7514

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)

Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Indonesia Service Center
THE PLAZZA OFFICE TOWER, 28TH FLOOR JL.M.H. THAMRIN KAV.28-30, JAKARTA, INDONESIA
TEL: +62-21-2992-2333 / FAX: +62-21-2992-2555

Malaysia (KL) Service Center
60, JALAN USJ 10 /1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN, MALAYSIA
TEL: +60-3-5631-7605 / FAX: +60-3-5631-7636

Malaysia (Johor Baru) Service Center
NO. 16, JALAN SHAH BANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 / FAX: +60-7-557-3404

Vietnam Service Center-1
ROOM 1004, 1005, FLOOR 10, 255 TRAN HUNG DAO CO GIANG WARD, DIST. 1, HCMC, VIETNAM
TEL: +84-8-3838-6931 / FAX: +84-8-3838-6932

Vietnam Service Center-2
LOT G10 - AREA 4 - HIEP BINH CHANH WARD - THU DUC DISTRICT - HCMC, VIETNAM
TEL: +84-8-2240-3587 / FAX: +84-8-3726-7968

Vietnam (Hanoi) Service Center
5FL, 59 - XA DAN STR., DONG DA DIST., HN, VIETNAM
TEL: +84-4-3573-7646 / FAX: +84-4-3573-7650

Philippines Service Center
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

MITSUBISHI ELECTRIC AUTOMATION (THAILAND) CO., LTD. (THAILAND FA CENTER)

BANG-CHAN INDUSTRIAL ESTATE NO.111 SOI SERITHAI 54
T.KANNAYAO, A.KANNAYAO, BANGKOK 10230, THAILAND
TEL: +66-2906-8255 / FAX: +66-2906-3239

Thailand Service Center
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1, FLOOR 7
RAMA III RD., BANGPONGPANG, YANNAWA, BANGKOK 10120, THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-9750

INDIA

MITSUBISHI ELECTRIC INDIA PVT. LTD.

India Service Center
2nd FLOOR, DLF BUILDING No.9B, DLF CYBER CITY
DLF PHASE-III, GURGAON 122002, HARYANA
TEL: +91-124-4630300 / FAX: +91-124-4630399

India (Bangalore) Service Center
FIRST & SECOND FLOOR, AVR BASE, MUNICIPAL NO.BC-308,
HENNURE BANASWADI ROAD, HRBR RING ROAD, BANGALORE-560 043, INDIA
TEL: +91-80-4020-1600 / FAX: +91-80-4020-1699
Chennai satellite office
Coimbatore satellite office

India (Pune) Service Center
TEL: +91-998-7997651
Baroda satellite office

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA LTD.

Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269 / FAX: +61-2-9684-7245

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)

China (Shanghai) Service Center
1-3,5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2308-2830
China (Ningbo) Service Dealer
China (Wuxi) Service Dealer
China (Jinan) Service Dealer
China (Wuhan) Service Satellite

China (Beijing) Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-3907
China (Beijing) Service Dealer

China (Tianjin) Service Center
B-2 801/802, YOUYI BUILDING, NO.50 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017
China (Shenyang) Service Satellite
China (Changchun) Service Satellite

China (Chengdu) Service Center
ROOM 407-408, OFFICE TOWER AT SHANGRI-LA CENTER, NO. 9 BINJIANG DONG ROAD,
JINJIANG DISTRICT, CHENGDU, SICHUAN 610021, CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China (Shenzhen) Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8218-4776
China (Xiamen) Service Dealer
China (Dongguan) Service Dealer

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)

Korea Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9602 / FAX: +82-2-3664-8668

Korea Taegu Service Satellite
603 CRYSTAL BUILDING 1666, SANBYEOK-DONG, BUK-KU, DAEGU, 702-010, KOREA
TEL: +82-53-604-6047 / FAX: +82-53-604-6049

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)

Taiwan (Taichung) Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan (Taipei) Service Center
3RD. FLOOR, NO.122 WUKUNG 2ND RD., WU-KU HSIANG, TAIPEI HSIEN, TAIWAN R.O.C.
TEL: +886-2-2299-2205 / FAX: +886-2-2298-1909

Taiwan (Tainan) Service Center
2F(C),1-1, CHUNGHWA-RD., YONGKANG CITY, TAINAN HSIEN, TAIWAN R.O.C.
TEL: +886-6-313-9600 / FAX: +886-6-313-7713

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 CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BUILDING,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M700V Series
MODEL CODE	100-208
Manual No.	IB-1500916

IB(NA)1500916(ENG)-D(1111)MEE
Made in JAPAN

Specifications are subject to change without notice.