MITSUBISHI ELECTRIC

Changes for the Better

MITSUBISHI CNC

C70

Specifications Manual

IB-1500259(ENG)-C

Introduction

This manual describes the specifications of CNC C70.
To safely use this CNC module, thoroughly study the "Precautions for Safety" on the next page before use.

Details described in this manual
At the beginning of each item, a table indicating it's specification according to the model.

⚠ CAUTION

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

General precautions
(1) When the contents of this manual is updated, the version (A, B, …) on the cover will be incremented.
(2) In this manual, the machining center system is described as "M system" and the lathe system is described as "L system".

Precautions for Safety

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

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

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

⚠ **DANGER**
Not applicable in this manual.

⚠ **WARNING**
Not applicable in this manual.

⚠ **CAUTION**
1. Items related to product and manual ⚠ The items that are not described in this manual must be interpreted as "not possible". ⚠ This manual is written on the assumption that all option functions are added. ⚠ Some functions may differ or some functions may not be usable depending on the NC system (software) version. **2. Items related to start up and maintenance** ⚠ Follow the power specifications (input voltage range, frequency range, momentary power failure time range) described in this manual. ⚠ Follow the environment conditions (ambient temperature, humidity, vibration, atmosphere) described in this manual. ⚠ If the parameter is used to set the temperature rise detection function to invalid, overheating may occur, thereby disabling control and possibly resulting in the axes running out of control, which in turn may result in machine damage and/or bodily injury or destruction of the unit. It is for this reason that the detection function is normally left "valid" for operation.

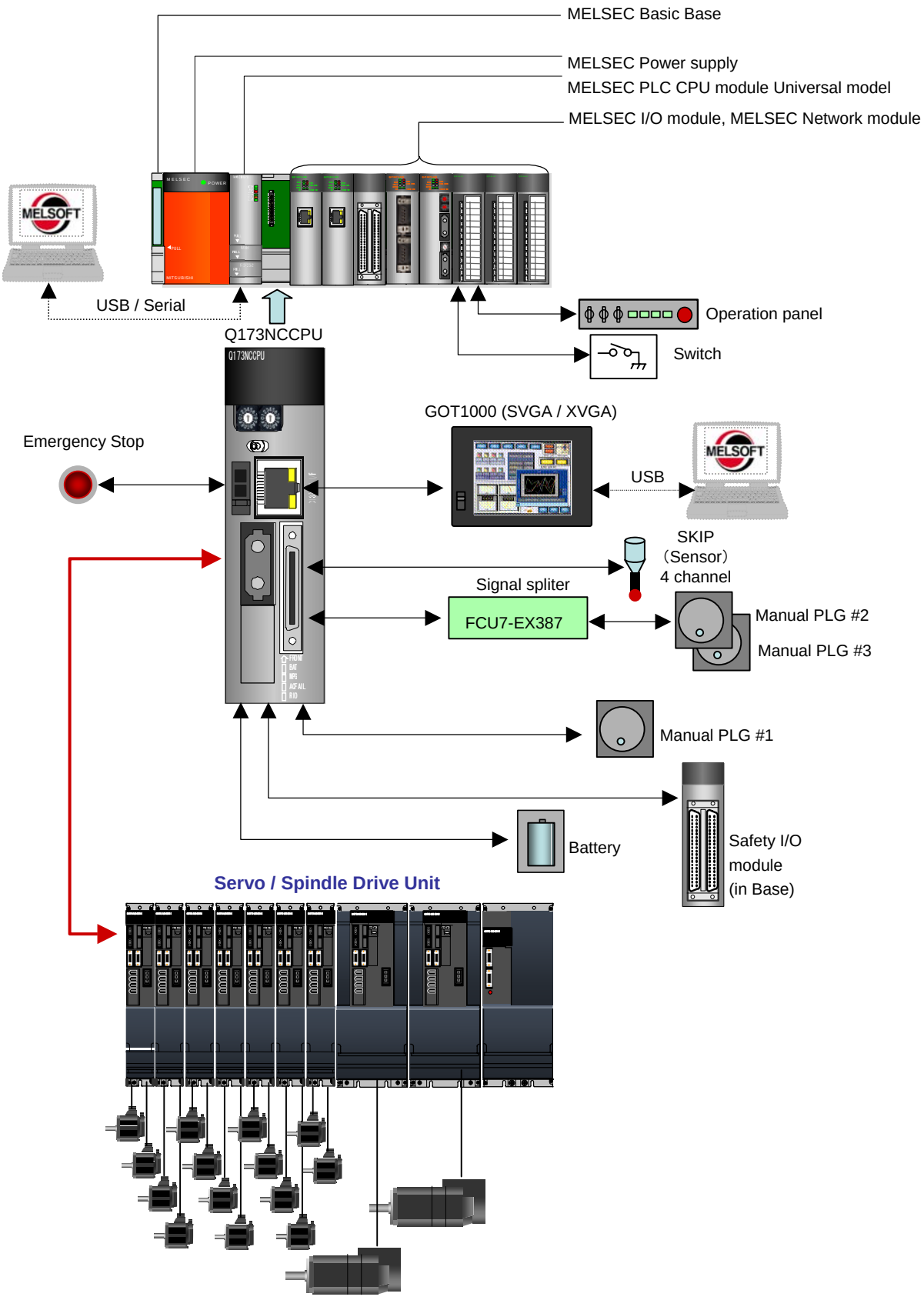
CONTENTS

I. GENERAL SPECIFICATIONS

II. FUNCTIONAL SPECIFICATIONS

I. GENERAL SPECIFICATIONS

1. System Configurations



2. General Specifications

General specifications of NC CPU are shown below.

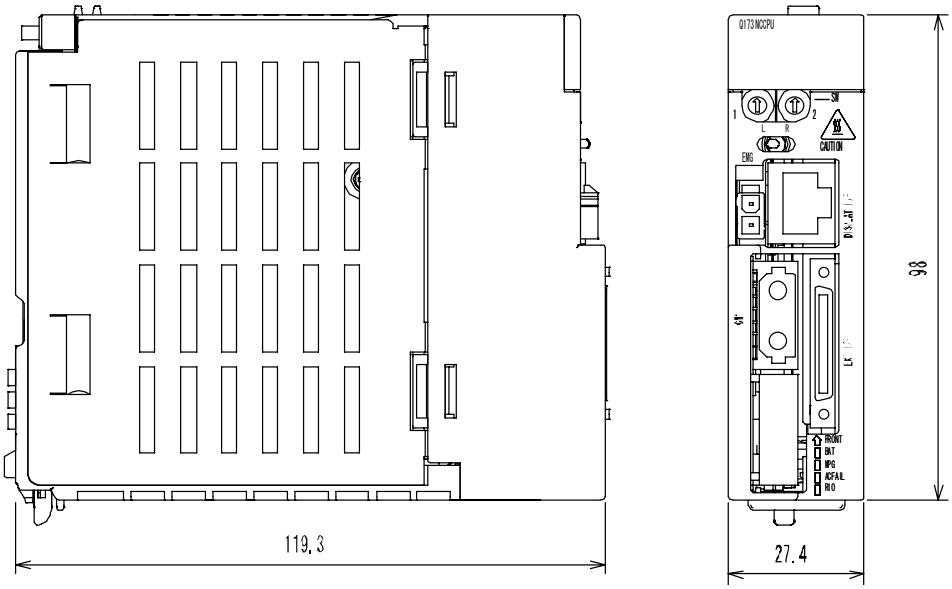
<table>
<tr><th>Item</th><th colspan="5">Specifications</th></tr>
<tr><td>Working ambient temperature</td><td colspan="5">0 to 55°C</td></tr>
<tr><td>Storage ambient temperature</td><td colspan="5">-25 to 75°C (-20 to 60°C for M700 series)</td></tr>
<tr><td>Working ambient humidity</td><td colspan="5">5 to 95% RH (with no dew condensation)</td></tr>
<tr><td>Storage ambient humidity</td><td colspan="5">5 to 95% RH (with no dew condensation)</td></tr>
<tr><td rowspan="8">Vibration resistance</td><td rowspan="8">Conforming to JIS B 3502 IEC61131-2</td><td colspan="3">Under intermittent vibration</td><td>Sweep count</td></tr>
<tr><td>Frequency</td><td>Acceleration</td><td>Amplitude</td><td rowspan="6">10 times each in X, Y, Z directions (for 80 min.)</td></tr>
<tr><td>10 to 57Hz</td><td>-</td><td>0.075mm</td></tr>
<tr><td>57 to150Hz</td><td>9.8m/s^2</td><td>-</td></tr>
<tr><td colspan="3">When there is a continuous vibration</td></tr>
<tr><td>Frequency</td><td>Acceleration</td><td>Amplitude</td></tr>
<tr><td>10 to 57Hz</td><td>-</td><td>0.035mm</td></tr>
<tr><td>57 to150Hz</td><td>4.9 m/s^2</td><td>-</td></tr>
<tr><td>Impact resistance</td><td colspan="5">Conforming to JIS B 3502, IEC61131-2
(147m/s^2 [15.0G], 3 times in each of 3 directions X, Y, Z)</td></tr>
<tr><td>Working atmosphere</td><td colspan="5">No corrosive gases</td></tr>
<tr><td>Altitude</td><td colspan="5">2000m or less</td></tr>
<tr><td>Installing</td><td colspan="5">Inside control panel</td></tr>
<tr><td>Overvoltage Category (Note 1)</td><td colspan="5">II or less</td></tr>
<tr><td>Pollution Class (Note 2)</td><td colspan="5">2 or less</td></tr>
</table>

(Note 1) This indicates the section of the power supply to which the equipment is assumed to be connected between the public electrical power distribution network and the machinery within premises. Category II applies to equipment for which electrical power is supplied from fixed facilities.
The surge voltage withstand level for up to the rated voltage of 300 V is 2500V.

(Note 2) This index indicates the degree to which conductive material is generated in terms of the environment in which the equipment is used.
In the environment corresponding to "Pollution level 2", basically only nonconductive pollution occurs, however temporary conductivity may occur due to the occasional condensing.

3. Outline Drawing

3.1 NC CPU Model Q173NCCPU



4. Servo/Spindle Drive System

(1) Power supply regenerative type

MDS-D-V1/V2 Series	MDS-DH-V1/V2 Series
200VAC (50Hz)/200 to 230VAC (60Hz) +10% -15%	380 to 440VAC (50Hz)/380 to 480VAC (60Hz) ±10%
MDS-D-V1 1st axis servo drive unit MDS-D-V2 2nd axis servo drive unit MDS-D-SP Spindle drive unit MDS-D-CV Power supply unit	MDS-DH-V1 1st axis servo drive unit MDS-DH-V2 2nd axis servo drive unit MDS-DH-SP Spindle drive unit MDS-DH-CV Power supply unit

(2) Resistance regenerative type

MDS-D-SVJ3/SPJ3 Series
200VAC (50Hz)/200 to 230VAC (60Hz) +10% -15%
MDS-D-SVJ3 1st axis servo drive unit MDS-D-SPJ3 Spindle drive unit

5. CNC Signals (PLC Interface Signals)

The CNC signal includes the following signals.
Refer to "PLC Interface Manual" for detail.

Bit Type Input Signals (CNC->PLC)

System State

24 hours continuous operation
Safety signal unconfirmed after compare error
Controller ready completion
Servo ready completion
Door open enable
In spindle synchronization
Spindle rotation speed synchronization completion
Spindle phase synchronization completion
Chuck close confirmation
Battery warning
Battery alarm
NC alarm 1
NC alarm 2 (Servo alarm)
In door interlock

Axis State

Servo ready
Axis selection
In axis plus motion
In axis minus motion
1st reference position reached
2nd reference position reached
3rd reference position reached
4th reference position reached
Near reference position
NC axis up-to-speed
Zero point initialization set completed
Zero point initialization set error completed
In zero point initialization
Zero point initialization incomplete
Current limit reached
Unclamp command
In-position

Part System State

In jog mode
In handle mode
In incremental mode
In manual arbitrary feed mode
In reference position return mode
In automatic initial set mode
In program operation mode (In memory mode)
In MDI mode
In automatic operation "run"
In automatic operation "start"
In automatic operation "pause"
In "reset"
In manual arbitrary feed
In rewind
Motion command completion
All axes in-position
All axes smoothing zero
Manual arbitrary feed completion
External search finished
In rapid traverse
In cutting feed
In tapping
In thread cutting
In synchronous feed
In constant surface speed
In skip
In reference position return
F 1-digit commanded
In tool life management output
Tool life over
NC alarm 3 (Program error)
NC alarm 4 (Operation error)
Search & start (error)
Search & start (search)
Illegal axis selected
F 1-digit No. code 1
F 1-digit No. code 2
F 1-digit No. code 4
M code independent output M00
M code independent output M01
M code independent output M02
M code independent output M30
M function strobe 1
M function strobe 2
M function strobe 3
M function strobe 4
Manual numerical command
Tool change position return completion
New tool change
T function strobe 1
T function strobe 2
T function strobe 3
T function strobe 4
2nd M function strobe 1
2nd M function strobe 2
2nd M function strobe 3
2nd M function strobe 4
S function strobe 1
S function strobe 2
S function strobe 3
S function strobe 4
S function strobe 5
S function strobe 6
S function strobe 7
Position switch 1
Position switch 2
Position switch 3
Position switch 4
Position switch 5
Position switch 6
Position switch 7
Position switch 8
Tap retract possible
No. of work machining over
Absolute position warning
Position switch 9
Position switch 10
Position switch 11
Position switch 12
Position switch 13
Position switch 14
Position switch 15
Position switch 16

Spindle State

S command gear No. illegal
S command max./min. command value over
S command no gear selected
Spindle speed upper limit over
Spindle speed lower limit over
Spindle gear shift command 1
Spindle gear shift command 2
Current detection
Speed detection
In spindle alarm
Zero speed
Spindle up-to-speed
Spindle in-position
In L coil selection
Spindle ready-ON
Spindle servo-ON
In spindle forward run
In spindle reverse run
Z-phase passed
Position loop in-position
In torque limit

Data Type Input Signals (CNC->PLC)

System State

Speed monitor door open possible
CRT display information
Emergency stop cause
User macro output #1132 (Controller -> PLC)
User macro output #1133 (Controller -> PLC)
User macro output #1134 (Controller -> PLC)
User macro output #1135 (Controller -> PLC)
CNC software version code
Battery drop cause
Temperature warning cause
Spindle synchronization phase error 1
Spindle synchronization phase error 2
Spindle synchronization phase error output
Spindle synchronization Phase error monitor
Spindle synchronization Phase error monitor (lower limit)
Spindle synchronization Phase error monitor (upper limit)
Spindle synchronization Phase offset data

Part System State

External search status
M code data 1
M code data 2
M code data 3
M code data 4
S code data 1
S code data 2
S code data 3
S code data 4
T code data 1
T code data 2
T code data 3
T code data 4
2nd M function data 1
2nd M function data 2
2nd M function data 3
2nd M function data 4
Tool No.
Group in tool life management
No. of work machining(current value)
Near reference position (per reference position)
Tool life usage data
No. of work machining(maximum value)
Error code output
S code data 5
S code data 6
S code data 7
User Macro output #1132 (Controller -> PLC)
User Macro output #1133 (Controller -> PLC)
User Macro output #1134 (Controller -> PLC)
User Macro output #1135 (Controller -> PLC)

Axis State

Thermal expansion compensation amount

Spindle State

Spindle command rotation speed input
Spindle command final data (Rotation speed)
Spindle actual speed

Bit Type Output Signals (PLC->CNC)

System Command

Contactor shutoff test signal
Integration time input 1
Integration time input 2
Data protect key 1
Data protect key 2
Data protect key 3
CRT changeover completion
Display changeover $1
Display changeover $2
PLC emergency stop
Door open I
Door open II
PLC axis control buffering mode valid
PLC axis 1st handle valid
PLC axis 2st handle valid
PLC axis 3st handle valid
Spindle synchronization cancel
Chuck close
Spindle synchronization
Spindle phase synchronization
Spindle synchronous rotation direction
Phase shift calculation request
Phase offset request
Error temporary cancel
PLC axis near point detection 1
PLC axis near point detection 2
PLC axis near point detection 3
PLC axis near point detection 4
PLC axis near point detection 5
PLC axis near point detection 6
PLC axis near point detection 7
PLC axis control valid 1
PLC axis control valid 2
PLC axis control valid 3
PLC axis control valid 4
PLC axis control valid 5
PLC axis control valid 6
PLC axis control valid 7

Axis Command

Control axis detach
Servo OFF
Mirror image
External deceleration +
External deceleration -
Automatic interlock +
Automatic interlock -
Manual interlock +
Manual interlock -
Automatic machine lock
Manual machine lock
Feed axis selection +
Feed axis selection -
Manual/Automatic simultaneous valid
Control axis detach 2
Current limit changeover
Droop release request
Zero point initialization set mode
Zero point initialization set start
Unclamp completion

Part System Command

Jog mode
Handle mode
Incremental mode
Manual arbitrary feed mode
Reference position return mode
Automatic initialization mode
Program operation mode (Memory mode)
MDI mode
Automatic operation "start" command (Cycle start)
Automatic operation "pause" command (Feed hold)
Single block
Block start interlock
Cutting block start interlock
Dry run
Error detect
NC reset 1

NC reset 2
Reset & rewind
Chamfering
Automatic restart
External search strobe
M function finish 1
M function finish 2
Tool length measurement 1
Tool length measurement 2 (L system)
Synchronization correction mode
Macro interrupt
Rapid traverse
Manual absolute
Recalculation request
Optional block skip 1
Reference position selection code 1
Reference position selection code 2
Reference position selection method
Optional block skip 2
Optional block skip 3
Optional block skip 4
Optional block skip 5
Optional block skip 6
Optional block skip 7
Optional block skip 8
Optional block skip 9
1st handle axis selection code 1
1st handle axis selection code 2
1st handle axis selection code 4
1st handle axis selection code 8
1st handle axis selection code 16
1st handle valid
2nd handle axis selection code 1
2nd handle axis selection code 2
2nd handle axis selection code 4
2nd handle axis selection code 8
2nd handle axis selection code 16
2nd handle valid
3rd handle axis selection code 1
3rd handle axis selection code 2
3rd handle axis selection code 4
3rd handle axis selection code 8
3rd handle axis selection code 16
3rd handle valid
Override cancel
Manual override method selection
Miscellaneous function lock
Tap retract
Reference position retract
Cutting feedrate override code 1
Cutting feedrate override code 2
Cutting feedrate override code 4
Cutting feedrate override code 8
Cutting feedrate override code 16
2nd cutting feedrate override valid
Cutting feedrate override method selection
Rapid traverse override code 1
Rapid traverse override code 2
Rapid traverse override method selection
Manual feedrate code 1
Manual feedrate code 2
Manual feedrate code 4
Manual feedrate code 8
Manual feedrate code 16
Manual feedrate method selection
Feedrate least increment code 1
Feedrate least increment code 2
Jog synchronous feed valid
Jog handle synchronous
Current limit mode 1
Current limit mode 2
Handle/incremental feed multiplication code 1
Handle/incremental feed multiplication code 2
Handle/incremental feed multiplication code 4
Handle/incremental feed magnification method selection
Tool alarm 1 (M system)/Tool skip 1 (L system)
Tool alarm 2 (M system)
Usage data count valid
Tool life management input (M system)
Tool change reset
Manual arbitrary feed 1st axis selection code 1
Manual arbitrary feed 1st axis selection code 2
Manual arbitrary feed 1st axis selection code 4
Manual arbitrary feed 1st axis selection code 8
Manual arbitrary feed 1st axis selection code 16
Manual arbitrary feed 1st axis valid
Manual arbitrary feed 2nd axis selection code 1
Manual arbitrary feed 2nd axis selection code 2
Manual arbitrary feed 2nd axis selection code 4
Manual arbitrary feed 2nd axis selection code 8
Manual arbitrary feed 2nd axis selection code 16
Manual arbitrary feed 2nd axis valid
Manual arbitrary feed 3rd axis selection code 1
Manual arbitrary feed 3rd axis selection code 2
Manual arbitrary feed 3rd axis selection code 4
Manual arbitrary feed 3rd axis selection code 8
Manual arbitrary feed 3rd axis selection code 16
Manual arbitrary feed 3rd axis valid
Manual arbitrary feed smoothing off
Manual arbitrary feed axis independent
Manual arbitrary feed EX.F/MODAL.F
Manual arbitrary feed G0/G1
Manual arbitrary feed MC/WK
Manual arbitrary feed ABS/INC
Manual arbitrary feed stop
Manual arbitrary feed strobe
2nd reference position return interlock
Search & start

Spindle Command

Gear shift completion
Spindle override code 1
Spindle override code 2
Spindle override code 4
Spindle override method selection
Spindle gear selection code 1
Spindle gear selection code 2
Spindle stop
Spindle gear shift
Spindle orientation
Spindle forward run start
Spindle reverse run start
Spindle forward run index
Spindle reverse run index
Spindle orientation command
L coil selection
Torque limit 1
Torque limit 2
Torque limit 3

Data Type Output Signals (PLC->CNC)

System Command

Speed monitor mode
User macro input #1032 (PLC -> Controller)
User macro input #1033 (PLC -> Controller)
User macro input #1034

(PLC -> Controller)
User macro input #1035
(PLC -> Controller)
PLC version code
1st axis index
2nd axis index
3rd axis index
4th axis index
5th axis index
6th axis index
7th axis index
8th axis index
9th axis index
10th axis index
11th axis index
12th axis index
13th axis index
14th axis index
15th axis index
16th axis index
Spindle synchronization Basic spindle selection
Spindle synchronization Synchronous spindle selection
Spindle synchronization Phase shift amount
PLC version code (method 2)

Part System Command

1st cutting feedrate override
2nd cutting feedrate override
Rapid traverse override
Manual feedrate
Handle/incremental feed magnification
2nd handle feed magnification
3rd handle feed magnification
Manual arbitrary feed 1st axis travel amount
Manual arbitrary feed 2nd axis travel amount
Manual arbitrary feed 3rd axis travel amount
OT ignored
Near-point dog ignored
Tool group No. designation
Synchronization control operation method
Search & start program No.
Workpiece coordinate offset measurement compensation No.
Selected tool No.
External search device No.
External search program No.
External search sequence No.
External search block No.
User Macro input #1032
(PLC -> Controller)
User Macro input #1033
(PLC -> Controller)
User Macro input #1034
(PLC -> Controller)
User Macro input #1035
(PLC -> Controller)

Axis State

External machine coordinate system compensation data
Each axis reference position selection
Thermal expansion offset compensation amount
Thermal expansion max. compensation amount

Spindle Command

Spindle command rotation speed output
S command override
Multi-point orientation position data

Classified Under Purpose
(CNC->PLC)
(PLC->CNC)

PLC axis state
PLC axis control
Window result information
Window command
Data registered to magazine for M system
Tool life management (M system)
Safety observing

II. FUNCTIONAL SPECIFICATIONS

1. Control Axes

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

1.1 Control Axes

1.1.1 Number of Basic Control Axes (NC axes)

M system : ○ 3 axes **L system : ○ 3 axes**

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

A number of axes that are within the maximum number of control axes, and that does not exceed the maximum number given for the NC axis, spindle and PLC axis can be used.
For example, if 16 NC axes are used, this alone is the maximum number of control axes, so a spindle and PLC axis cannot be connected.

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

M system : 16 axes **L system : 16 axes**

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

M system : 16 axes **L system : 16 axes**

Max. number of spindles

M system : 7 axes **L system : 4 axes**

Max. number of PLC axes

M system : 7 axes **L system : 7 axes**

1.1.4 Number of Simultaneous Contouring Control Axes

Simultaneous control of up to four axes or less is possible in the same part system.
However, for actual use, the machine tool builder specification will apply.

M system : 4 axes **L system : 4 axes**

1.1.5 Max. Number of NC Axes in a Part System

M system : 8 axes **L system : 8 axes**

1.2 Control Part System

1.2.1 Standard Number of Part Systems

M system : 1 part system **L system : 1 part system**

1.2.2 Max. Number of Part Systems

M system : Δ7 part systems **L system : Δ3 part systems**

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

1.3 Control Axes and Operation Modes

1.3.2 Memory Mode

M system : ○ **L system : ○**

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

1.3.3 MDI Mode

M system : ○ **L system : ○**

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

2. Input Command

2.1 Data Increment

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

M system : ○ L system : ○

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

M system : Δ L system : Δ

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

(1) The least input increment indicates the increment handled in the internal processing of the controller. The counter and tool offset data, etc., input from the screen is handled with this increment. This increment is applied per part system (all part systems, PLC axis).

Increment type	Input increment (parameter)	Metric unit system		Inch unit system	
		Linear axis (Unit = mm)	Rotary axis (Unit = °)	Linear axis (Unit = inch)	Rotary axis (Unit = °)
Least input increment	B	0.001	0.001	0.0001	0.001
	C	0.0001	0.0001	0.00001	0.0001

(Note 1) The inch and metric systems cannot be used together.

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

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

(Note 1) The inch and metric systems cannot be used together.

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

2.2 Unit System

2.2.1 Inch/Metric Changeover

M system : Δ **L system : Δ**

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

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

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

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

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

2.3 Program Format

2.3.1 Program Format

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

2.3.1.1 Format 1 for Lathe (G code list 2, 3)

M system : - **L system : ○**

2.3.1.2 Format 2 for Lathe (G code list 4, 5)

M system : - **L system : ○**

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

M system : ○ **L system : -**

2.4 Command Value

2.4.1 Decimal Point Input I, II

M system : ○ **L system : ○**

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

(1) Decimal point input type I (When parameter #1078 Decpt2 is 0.)
When axis coordinates and other data are supplied in machining program commands, the assignment of the program data can be simplified by using the decimal point input. The minimum digit of a command not using a decimal point is the same as the least command increment.
Usable addresses can be applied not only to axis coordinate values but also to speed commands and dwell commands.
The decimal point position serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in a time designation of dwell command.

(2) Decimal point input type II (When parameter #1078 Decpt2 is 1.)
As opposed to type I, when there is no decimal point, the final digit serves as the millimeter unit in the metric mode, as the inch unit in the inch mode and as the second unit in the time designation.
The "." (point) must be added when commands below the decimal point are required.

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

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

2.4.2 Absolute/Incremental Command

M system : O **L system : O**

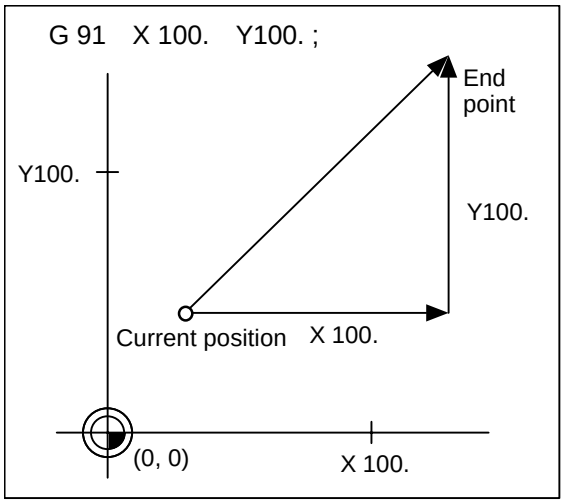
(1) M system
When axis coordinate data is issued in a machining program command, either the incremental command method (G91) that commands a relative distance from the current position or the absolute command method (G90) that moves to a designated position in a predetermined coordinate system can be 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 value command)
G91 ... Incremental command (incremental value command)
These G codes can be commanded multiple times in one block.

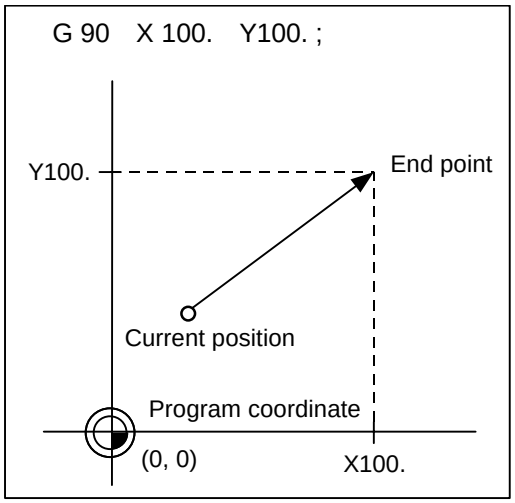
Example

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

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

(Incremental value command) (Absolute value command)





(2) L system

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

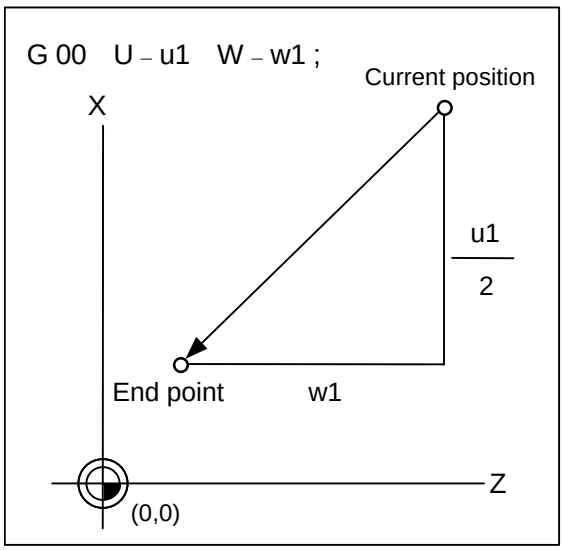
When issuing an incremental value command, the axis address to be commanded as the incremental axis name is registered in the parameter. However, the arc radius designation (R) and arc center designation (I, J, K) always use incremental designations.

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

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

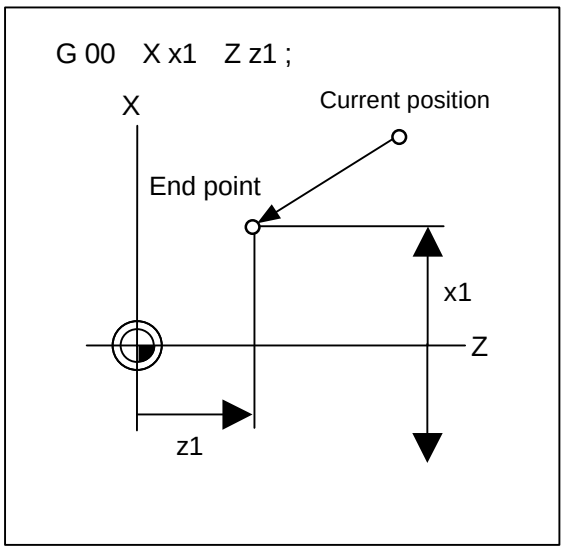
Example	G00	X100.	W200.	;
		Absolute value	Incremental value	

(Incremental value command)



The above drawing shows the case for the diameter command.

(Absolute value command)



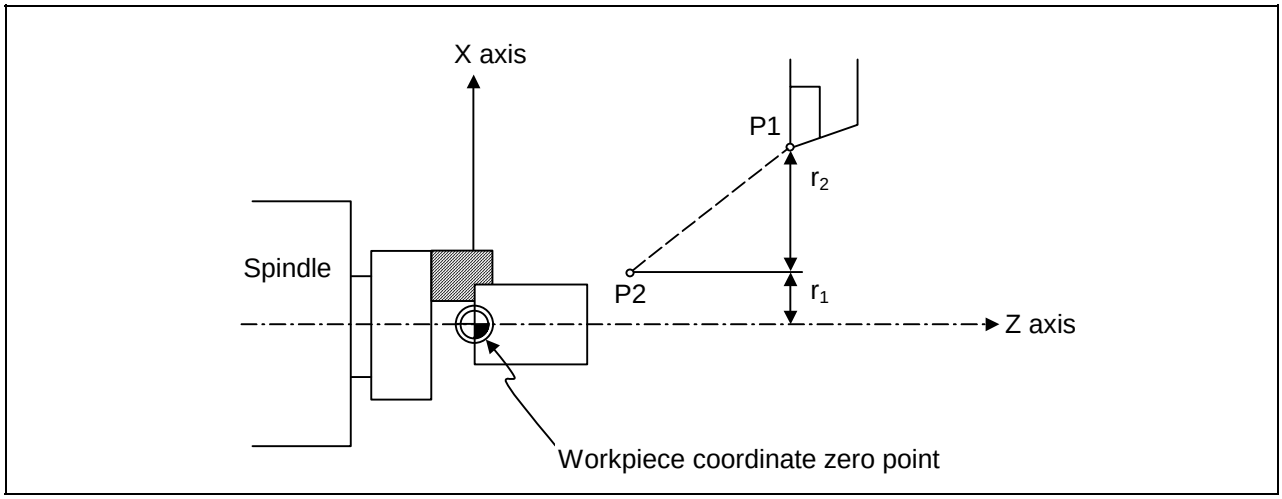
The above drawing shows the case for the diameter command.

(Note 1) In addition to the above command method using the above axis addresses, the absolute value command and incremental value command can be switched by commanding the G code (G90/G91). (Select with the parameters.)

2.4.3 Diameter/Radius Designation

M system : - **L system : O**

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

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

Radius and diameter commands

3. Positioning/Interpolation

3.1 Positioning

3.1.1 Positioning

M system : O **L system : O**

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

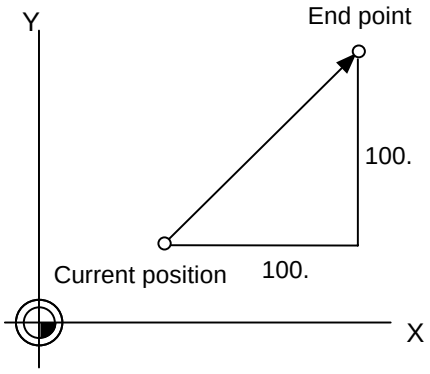
G00 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously) x1, y1, z1: numerical values denoting the position data

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

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

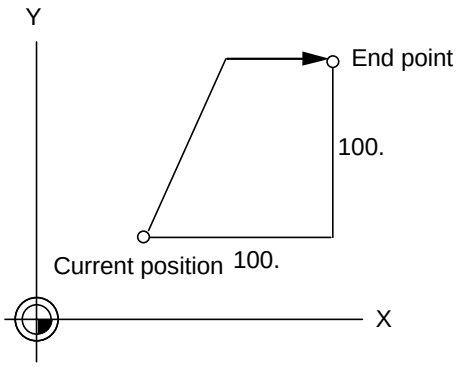
(Example) Linear type (Moves lineary to the end point.)

G 00 G 91 X 100. Y 100. ;



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

G 00 G 91 X 100. Y 100. ;



(Note 1) If the acceleration/deceleration conditions differ between the axes, the path will not be linear to the end point even when using the linear type.

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

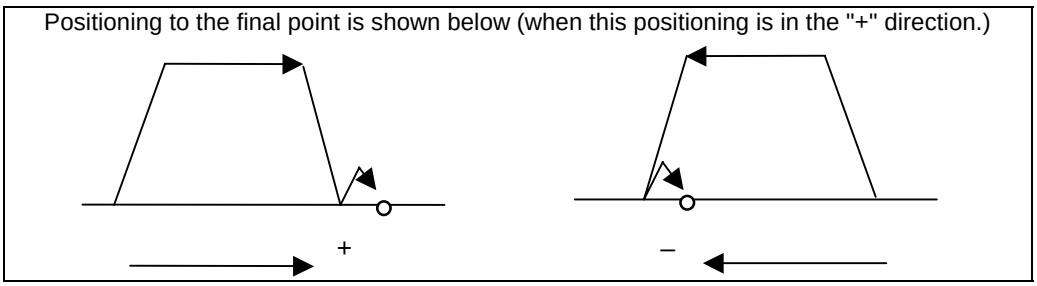
3.1.2 Unidirectional Positioning

M system : O **L system : -**

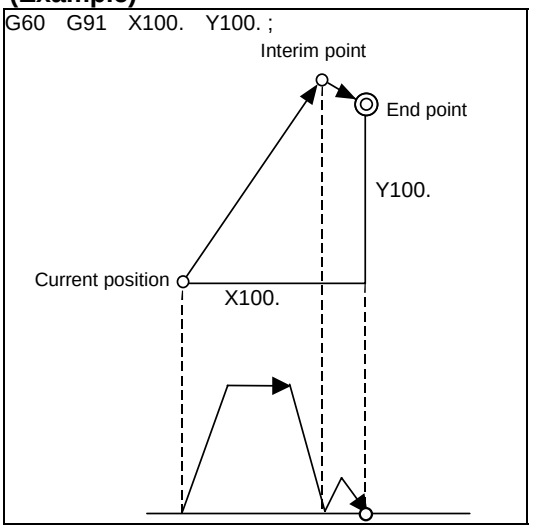
The G60 command always moves the tool to the final position in a direction determined with parameters. The tool can be positioned without backlash.

G60 Xx1 Yy1 Zz1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)
x1, y1, z1: numerical values denoting the position data

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



(Example)



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

(Note 1) The processing of the above pattern will be followed even for the machine lock and Z-axis command cancel.
(Note 2) 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

M system : O **L system : O**

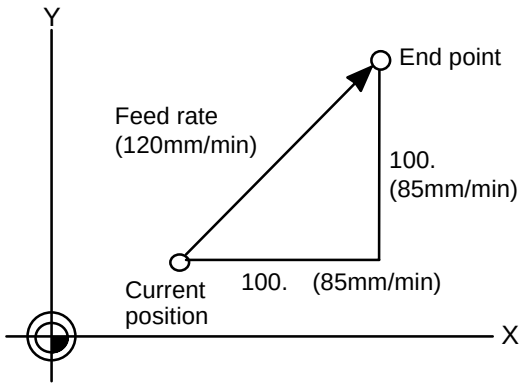
Linear interpolation is a function that moves a tool linearly by the movement command value supplied in the program at the cutting feed rate designated by the F code.

G01 Xx1 Yy1 Zz1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W simultaneously)
x1, y1, z1 : numerical values denoting the position data
f1 : numerical value denoting the feed rate data

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

(Example)

G01 G91 X100. Y100. F120 ;



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

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

3.2.2 Circular Interpolation (Center/Radius Designation)

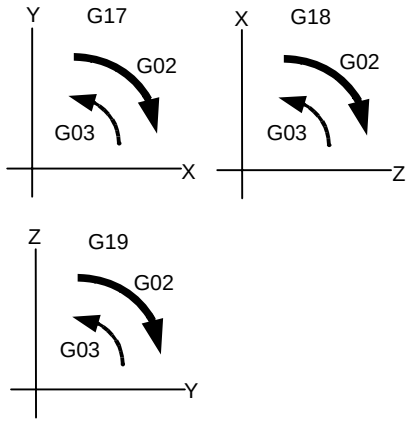
M system : O **L system : O**

(1) Circular interpolation with I, J, K commands
This function moves a tool along a circular arc on the plane selected by the plane selection G code with movement command value supplied in the program.

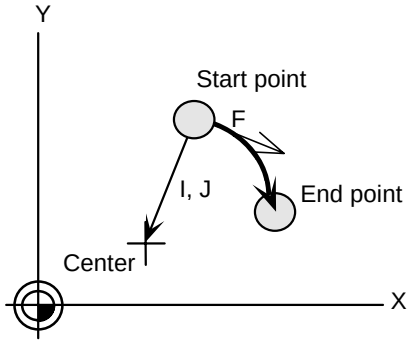
G02(G03) Xx1 Yy1 Ii1 Jj1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)	
G02, G03	: Arc rotation direction
Xx1, Yy1	: End point coordinate values
Ii1, Jj1	: Arc center coordinate values
Ff1	: Feed rate

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

The direction of the arc rotation is specified by G02 or G03.
G02: Clockwise (CW)
G03: Counterclockwise (CCW)
The plane is selected by G17, G18 or G19.
G17: XY plane
G18: ZX plane
G19: YZ plane



(Example) See below for examples of circular commands.



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

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

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

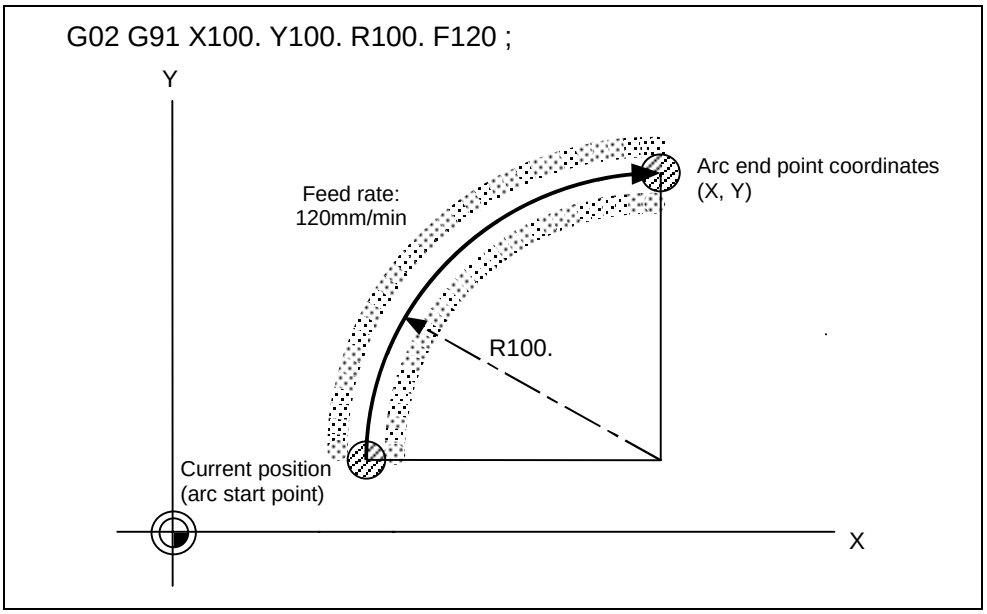
(2) R-specified circular interpolation
Besides the designation of the arc center coordinates using the above-mentioned I, J and K commands, arc commands can also be issued by designating the arc radius directly.

G02(G03) Xx1 Yy1 Rr1 Ff1 ; (Also possible for additional axes A, B, C, U, V, W)
G02, G03 : Arc rotation direction Xx1, Yy1 : End point coordinate values Rr1 : Arc radius Ff1 : Feed rate

G02 or G03 is used to designate the direction of the arc rotation.
The arc plane is designated by G17, G18 or G19.

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

(Example)



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

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

3.2.3 Helical Interpolation

M system : Δ **L system : -**

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

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

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

(1) The arc plane is designated by G17, G18 or G19.
(2) G02 or G03 is used to designate the direction of the arc rotation.
(3) Absolute or incremental values can be assigned for the arc end point coordinates and the end point coordinate of the linear axis, but incremental values must be assigned for the arc center coordinates.
(4) The linear interpolation axis is the other axis which is not included in the plane selection.
(5) Command the speed in the component direction that represents all the axes combined for the feed rate. Pitch l1 is obtained by the formula below.

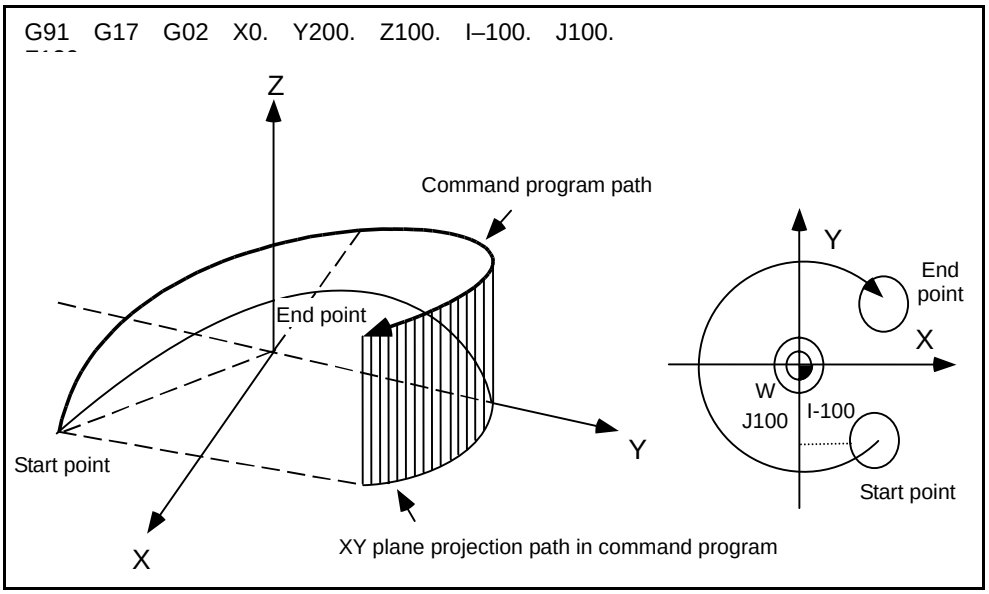
$l1 = z1/((2\pi \bullet p1 + \theta)/2\pi)$

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

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

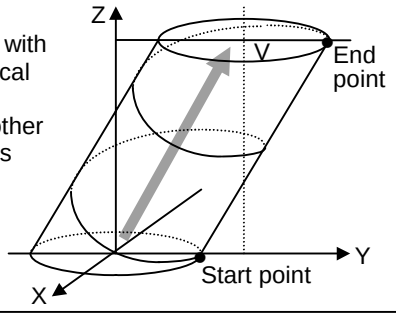
The combination of the axes which can be commanded simultaneously depends on the specifications. The axes can be used in any combination under the specifications. The feed rate is controlled so that the tool always moves at a speed along the circumference of the circle.

(Example)



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

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



4. Feed

4.1 Feed Rate

4.1.1 Rapid Traverse Rate (m/min)

M system : 1000 **L system : 1000**

[M system]

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

- Rapid Traverse Rate setting range

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

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

[L system]

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

- Rapid Traverse Rate setting range

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

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

4.1.2 Cutting Feed Rate (m/min)

M system : 1000 **L system : 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).

- Cutting feed rate setting range

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

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

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

[L system]

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

- Cutting Feed Rate setting range

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

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

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

4.1.3 Manual Feed Rate (m/min)

M system : 1000 **L system : 1000**

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

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

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

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

4.1.4 Rotary Axis Command Speed Tenfold

M system : O **L system : O**

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

M system : O **L system : O**

[M system]

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

Metric input (mm)

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

Inch input (inch)

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

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

[L system]

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

Metric input (mm)

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

Inch input (inch)

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

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

4.2.2 Feed per Revolution

M system : Δ **L system : O**

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

[M system]

Metric input (mm)

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

Inch input (inch)

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

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

[L system]

Metric input (mm)

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

Inch input (inch)

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

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

4.2.4 F1-digit Feed

M system : O **L system : O**

When the "F1digt" parameter is ON, the feed rate registered by parameter in advance can be assigned by designating a single digit following address F.
There are six F codes: F0 and F1 to F5. The rapid traverse rate is applied when F0 is issued which is the same as the G00 command. When one of the codes F1 to F5 is issued, the cutting feed rate set to support the code serves as the valid rate command. When a command higher than F5 is issued, it serves as a regular direct command with feed rate value of 5 digits following address F.
When an F1-digit command has been issued, the "In F1-digit" external output signal is output.

4.3 Override

4.3.1 Rapid Traverse Override

M system : ○ **L system : ○**

(1) Code method
Four levels of override (1%, 25%, 50% and 100%) can be applied to manual or automatic rapid traverse using the external input signal supplied.
Code method commands are assigned as combinations of bit signals from the PLC.

(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) Code method and value setting method can be selected by PLC processing.

4.3.2 Cutting Feed Override

M system : ○ **L system : ○**

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

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

4.3.3 2nd Cutting Feed Override

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.

4.3.4 Override Cancel

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 (memory and MDI).

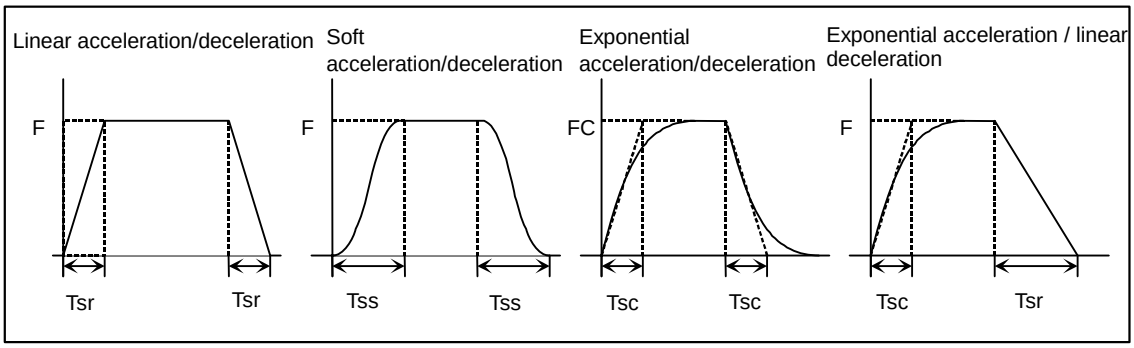
(Note 1) The override cancel signal is not valid for manual operation.
(Note 2) When the cutting feed override or second cutting feed override is 0%, the 0% override takes precedence and the override is not canceled.
(Note 3) The override cancel signal is not valid for rapid traverse.

4.4 Acceleration/Deceleration

4.4.1 Automatic Acceleration/Deceleration after Interpolation

M system : ○ **L system : ○**

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

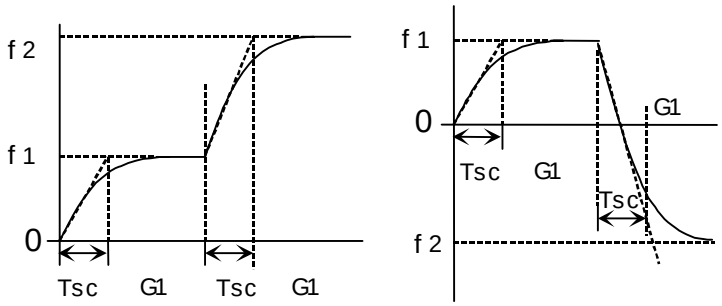


(Note 1) The rapid traverse feed acceleration/deceleration patterns are effective for the following: G00, G27, G28, G29, G30, rapid traverse feed in manual run, JOG, incremental feed, return to reference position.

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

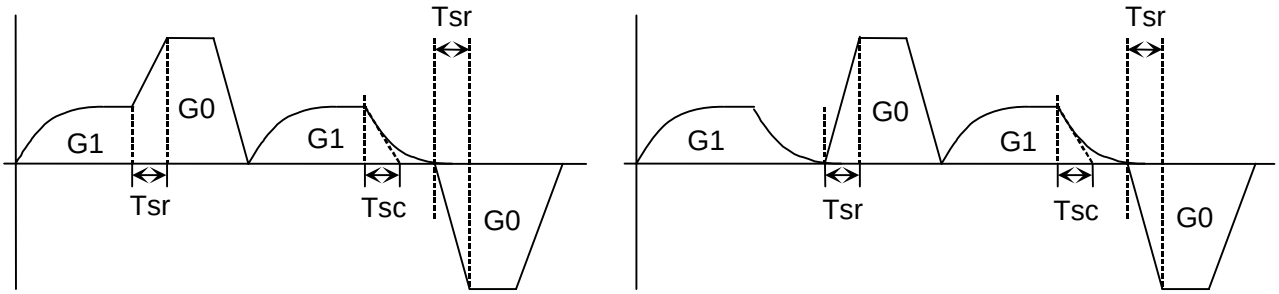
Acceleration/Deceleration during Continuing Blocks

(1) Continuous G1 blocks



The tool does not decelerate between blocks.

(2) Continuous G1-G0 blocks



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

4.4.2 Rapid Traverse Constant Inclination Acceleration/Deceleration

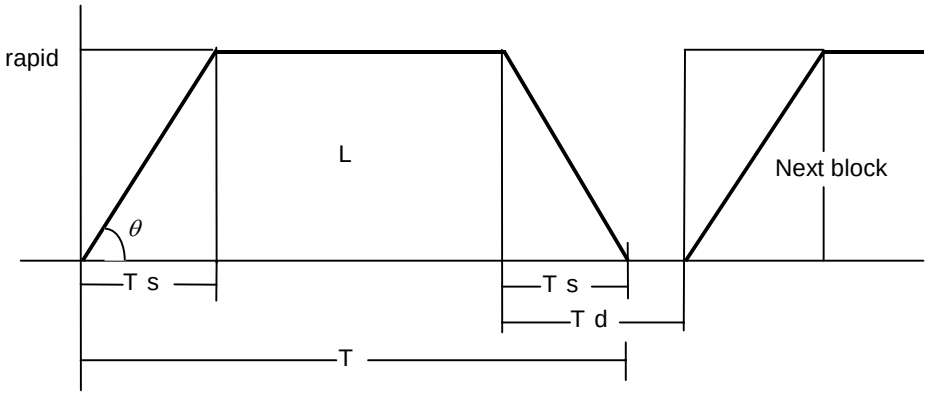
M system : O **L system : O**

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

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

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

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



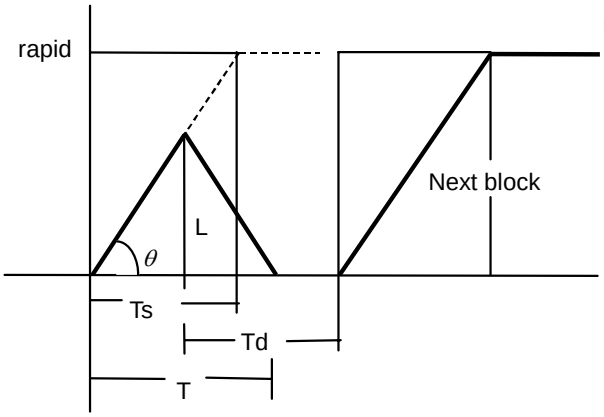
rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

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

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

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

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



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

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

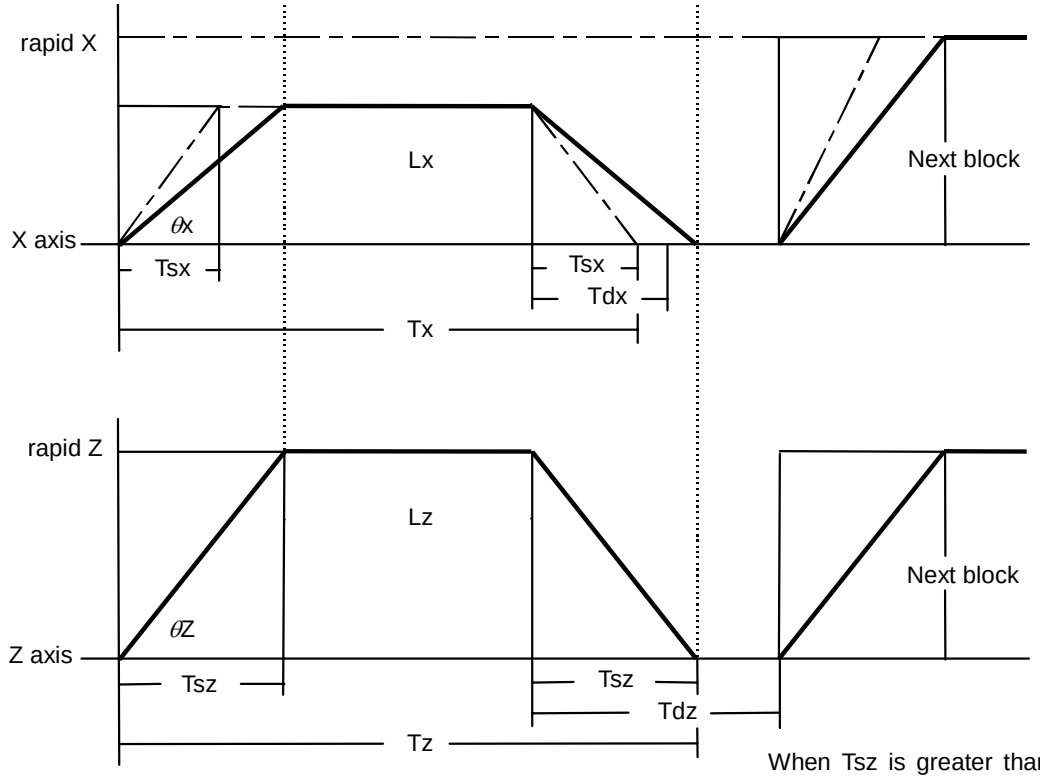
$$Td = \frac{T}{2} + (0 \sim 1.7\ ms)$$

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

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

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

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



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

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

4.5 Thread Cutting

4.5.1 Thread Cutting (Lead/Thread Number Designation)

M system : Δ **L system : ○**

(1) Lead designation
The thread cutting with designated lead are performed based on the synchronization signals from the spindle encoder.

G33 Zz1/Ww1 Xx1/Uu1 Qq1 Ff1/Ee1 ;	
G33	: Thread 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 tables below indicate the thread lead ranges.

[M system]

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

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

[L system]

Metric command		
Least input increment (mm)	**F (mm/rev)**	**E (mm/rev)**
0.001	0.0001 ~999.9999	0.00001 ~999.99999
0.0001	0.00001 ~99.99999	0.000001 ~99.999999

Inch command		
Least input increment (inch)	**F (inch/rev)**	**E (inch/rev)**
0.0001	0.000001 ~99.999999	0.000010 ~9.9999999
0.00001	0.0000001 ~9.9999999	0.00000001 ~0.99999999

The direction of the axis with a large movement serves as the reference for the lead.

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

G33 Zz1/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

The tables below indicate the thread number.

[M system]

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

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

[L system]

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

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

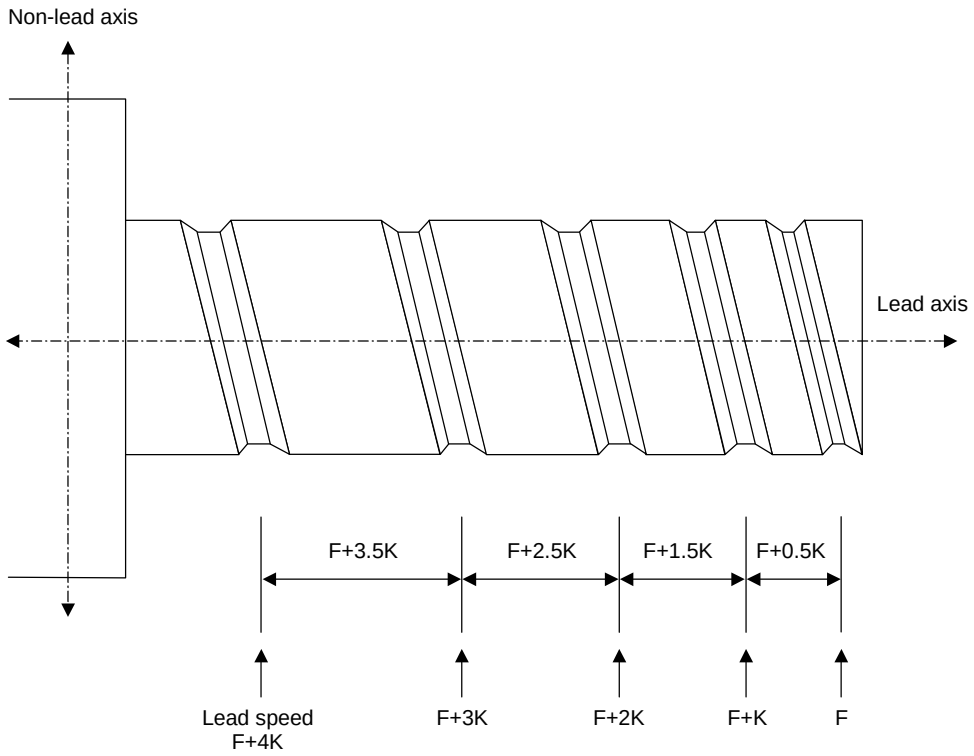
The number of thread per inch is commanded for both metric and inch systems, and the direction of the axis with a large movement serves as the reference.

4.5.2 Variable Lead Thread Cutting

M system : - **L system : O**

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

G34	**X/U__Z/W__F/E__K__;**
G34	: Variable lead thread cutting command
X/U	: Thread end point X coordinate
Z/W	: Thread end point Z coordinate
F/E	: Thread's basic lead
K	: Lead increment/decrement amount per thread rotation



4.5.3 Synchronous Tapping

4.5.3.1 Synchronous Tapping Cycle

M system : Δ **L system : Δ**

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

(1) Tapping pitch assignment

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

(2) Tapping thread number assignment

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

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

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

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

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

4.5.3.2 Pecking Tapping Cycle

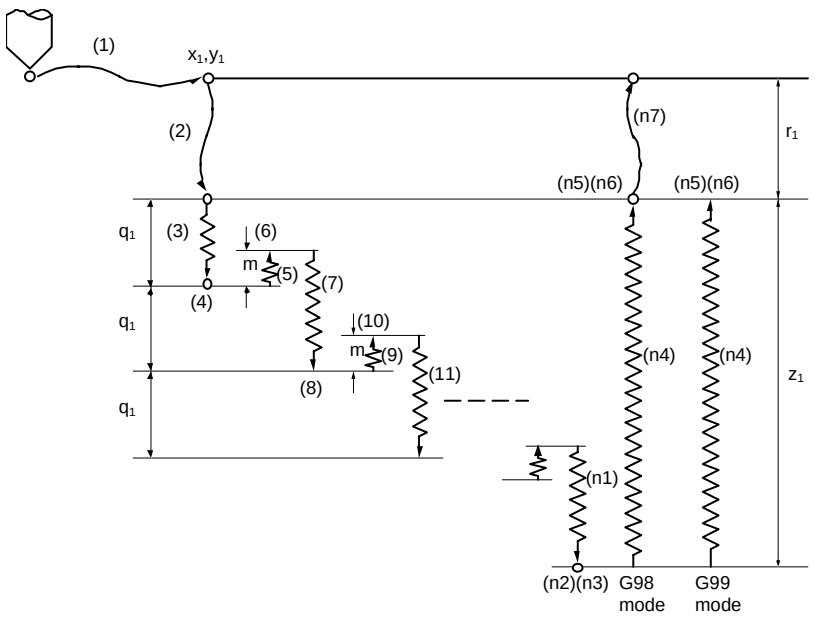
M system : Δ **L system : -**

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.
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
, Ii1	: In-position width of positioning axis
, Jj1	: In-position width of hole drilling axis
, Rr2	: Synchronization method selection (r2=1 synchronous, r2=0 asynchronous)

(Note 1) When ",R0" is commanded, F address is regarded as cutting feedrate.



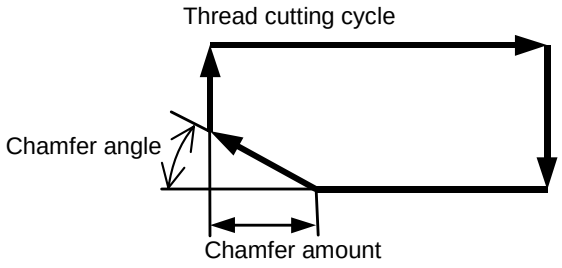
(1)	G0 Xx1 Yy1 ,Ii1
(2)	G0 Zr1
(3)	G1 Zq1 Ff1
(4)	M4 (Spindle reverse rotation)
(5)	G1 Z-m Ff1
(6)	M3 (Spindle forward rotation)
(7)	G1 Z(q1+m) Ff1
(8)	M4 (Spindle reverse rotation)
(9)	G1 Z-m Ff1
(10)	M3 (Spindle forward rotation)
(11)	G1 Z(q1+m) Ff1
:	:
(n1)	G1 Z(z1-q1*n) Ff1
(n2)	G4 Pp1
(n3)	M4 (Spindle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

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

4.5.4 Chamfering

M system : - **L system : O**

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

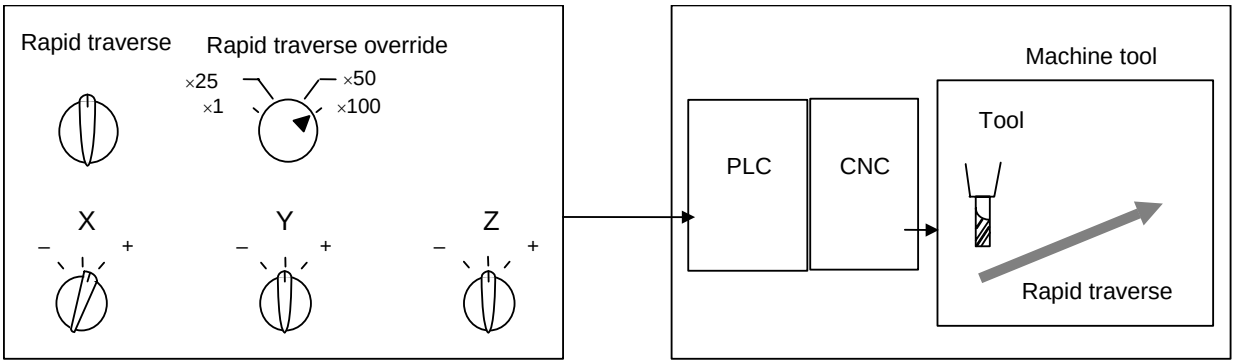


4.6 Manual Feed

4.6.1 Manual Rapid Traverse

M system : O **L system : O**

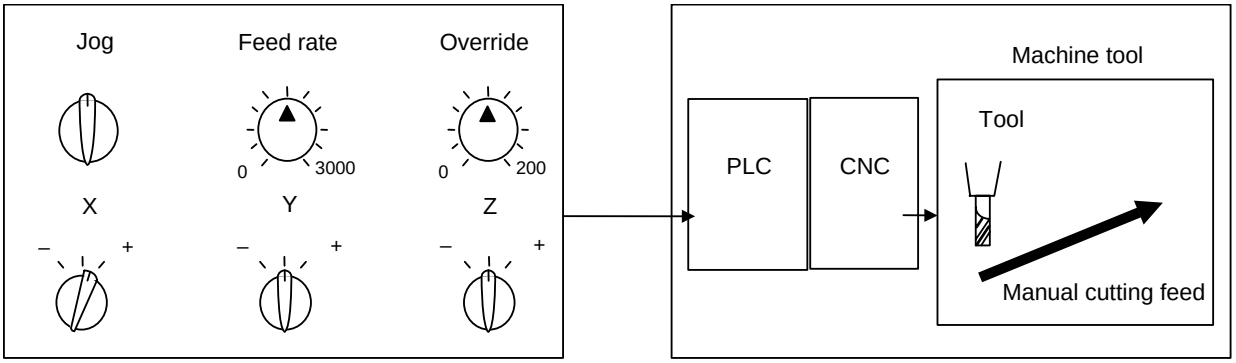
When the manual rapid traverse mode is selected, the tool can be moved at the rapid traverse rate for each axis separately. Override can also be applied to the rapid traverse rate by means of the rapid traverse override function.
Rapid traverse override is common to all part systems.



4.6.2 Jog Feed

M system : O **L system : O**

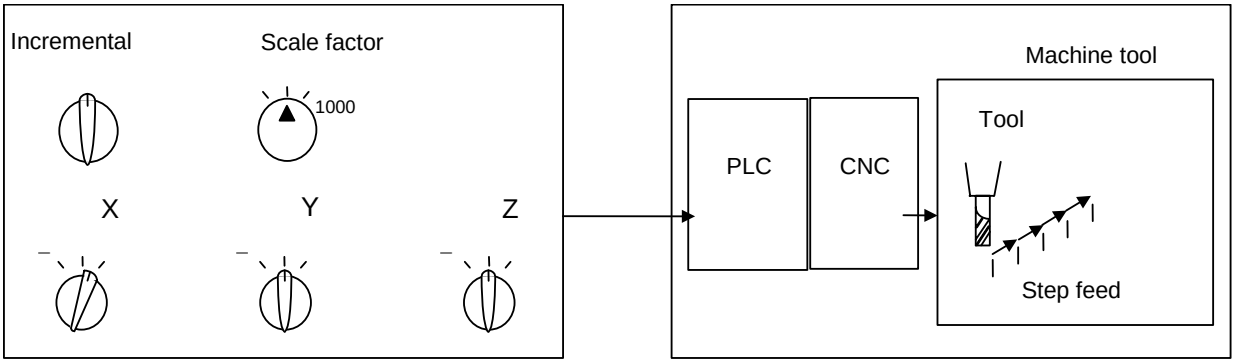
When the jog feed mode is selected, the tool can be moved in the axis direction (+ or –) in which the machine is to be moved at the per-minute feed. The jog feed rate is common to all part systems.



4.6.3 Incremental Feed

M system : ○ **L system : ○**

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



4.6.4 Handle Feed

M system : Δ **L system : Δ**

(1-axis)
In the handle feed mode, the machine can be moved in very small amounts by rotating the manual pulse generator. The scale can be selected from X1, X10, X100, X1000 or arbitrary value.
(Note 1) The actual movement amount and scale may not match if the manual pulse generator is rotated quickly.

(3-axes)
In the handle feed mode, individual axes can be moved in very small amounts either separately or simultaneously by rotating the manual pulse generators installed on each of the axes.
(Note 1) The actual movement amount and scale may not match if the manual pulse generator is rotated quickly.

4.7 Dwell

4.7.1 Dwell (Time-based Designation)

M system : O **L system : O**

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 G04 Pp1 ;		
G94	:	Asynchronous
G04	:	Dwell
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.)

5. Program Memory/Editing

5.1 Memory Capacity

Machining programs are stored in the NC memory.

5.1.1 Memory Capacity (Number of Programs Stored)

(Note 1) The tape length will be the total of two part systems when using the 2-part system specifications.

40 m (15KB) (64 programs)

M system : ○ **L system : ○**

80 m (30KB) (128 programs)

M system : Δ **L system : Δ**

160 m (60KB) (200 programs)

M system : Δ **L system : Δ**

320 m (125KB) (200 programs)

M system : Δ **L system : Δ**

600 m (230KB) (400 programs)

M system : Δ **L system : Δ**

5.2 Editing

5.2.1 Program Editing

M system : ○ **L system : ○**

The following editing functions are possible.

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

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

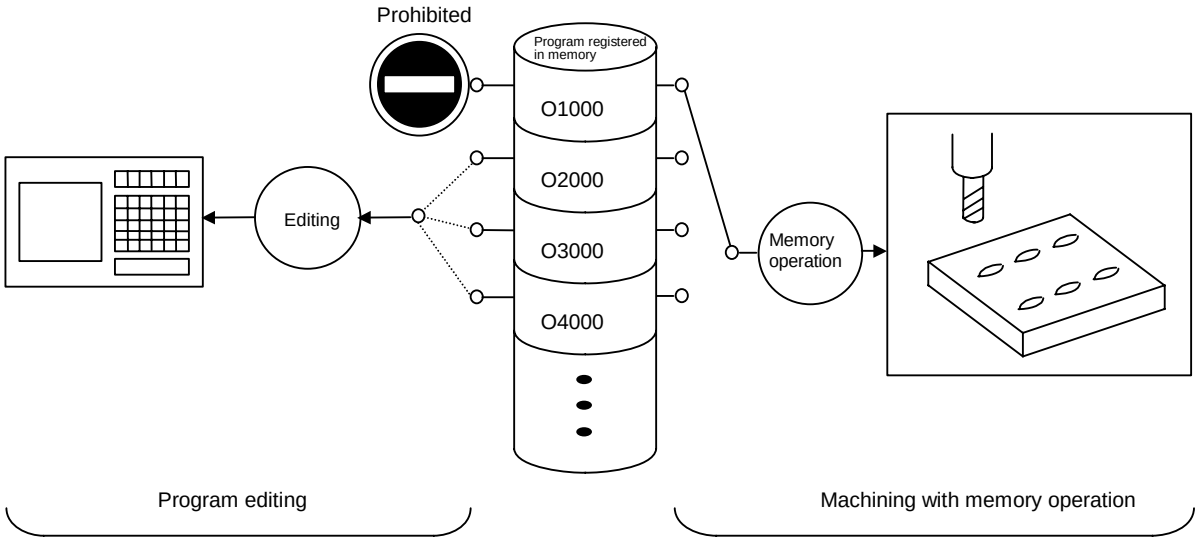
(3) Program copying
  (a) Machining programs stored in the controller memory can be copied, condensed or merged.
  (b) The program No. of the machining programs in the memory can be changed.

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

5.2.2 Background Editing

M system : O **L system : O**

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



(1) The data of the machining programs being used in memory operation can be displayed and scrolled on the setting and display unit, but data cannot be added, revised or deleted.
(2) The editing functions mentioned in the preceding section can be used at any time for machining programs which are not being used for memory operation.
This makes it possible to prepare and edit the next program for machining, and so the machining preparations can be made more efficiently.
(3) The machining program will not be searched as the operation target even when searched in the edit screen.

6. Operation and Display

6.1 Structure of Operation/Display Panel

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

6.1.1 MITSUBISHI Graphic Operation Terminal (GOT)

GT1595-XTBD/GT1595-XTBA
M system : □ L system : □

GT1585V-STBD/GT1585V-STBA
M system : □ L system : □

GT1585-STBD/GT1585-STBA
M system : □ L system : □

GT1575V-STBD/GT1575V-STBA
M system : □ L system : □

GT1575-STBD/GT1575-STBA
M system : □ L system : □

6.2 Operation Methods and Functions

6.2.2 Absolute Value/Incremental Value Setting

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.
- Tool compensation amount screen
- Coordinate system offset screen

6.2.5 Displayed Part System Switch

M system : ○ L system : ○

The part system displayed on the screen can be changed with the SHIFT , $ keys.
The number of displayed part systems is counted by one each time the SHIFT , $ 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.

6.3 Display Methods and Contents

6.3.1 Status Display

M system : ○ **L system : ○**

The status of the program now being executed is indicated.
(1) Display of G, S, T, M commands and 2nd miscellaneous command modal values
(2) Feed rate display
(3) Tool offset number and offset amount display
(4) Real speed display (Note 1)

(Note 1) 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

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 Position Display, Screen Display (Operation Screen Display)

M system : ○ **L system : ○**

Various information related to operation, such as the axis counter, speed display and MSTB command are displayed on the Position Display screen. The following operations regarding operation can be executed.
(1) Operation search
(2) Setting of common variables
(3) Setting of local variables
(4) Counter zero
(5) Origin zero
(6) Manual numeric command, etc.

6.3.4 Tool Compensation/Parameter Screen Display (Preparation Screen Display)

M system : ○ **L system : ○**

Tool/workpiece related settings, user parameter settings, manual numeric command issuing and tool length measurements can be carried out on the Tool Compensation/Parameter screen.

6.3.5 Program Screen Display (Edit Screen Display)

M system : ○ **L system : ○**

Machining program searching, creating and editing (addition, deletion, change), program list display and MDI editing can be carried out on the Program screen.

6.3.6 Alarm Diagnosis Screen Display (Diagnosis Screen Display)

M system : ○ **L system : ○**

The following operations related to the CNC diagnosis can be carried out on the Diagnosis screen.
(1) Display of hardware, software and drive unit configuration
(2) Operation monitor of servo and spindle drive unit
(3) Diagnosis of NC input/output signal (interface diagnosis)
(4) Display of operation history
(5) Display of alarm / stop code history list
(6) Data sampling for maintenance
(7) Deleting, copying and list displaying of machining program

6.3.7 Maintenance Screen Display

M system : ○ **L system : ○**

Refer to "6.3.6 Alarm Diagnosis Screen Display (Diagnosis Screen Display)".

6.3.8 Additional Language

6.3.8.1 Japanese

M system : ○ **L system : ○**

6.3.8.2 English

M system : ○ **L system : ○**

6.3.8.7 Chinese

(a) Simplified Chinese Characters

M system : Δ **L system : Δ**

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 compact flash card mounted on GOT (MITSUBISHI graphic operation terminal).

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

M system : ○ **L system : ○**

Tool offset data input / output

M system : ○ **L system : ○**

Common variable input / output

M system : ○ **L system : ○**

Parameter input / output

M system : ○ **L system : ○**

History data output

M system : ○ **L system : ○**

7.2 Input/Output I/F

7.2.3 Ethernet I/F

M system : ○ **L system : ○**

Various data can be input and output with display I/F.

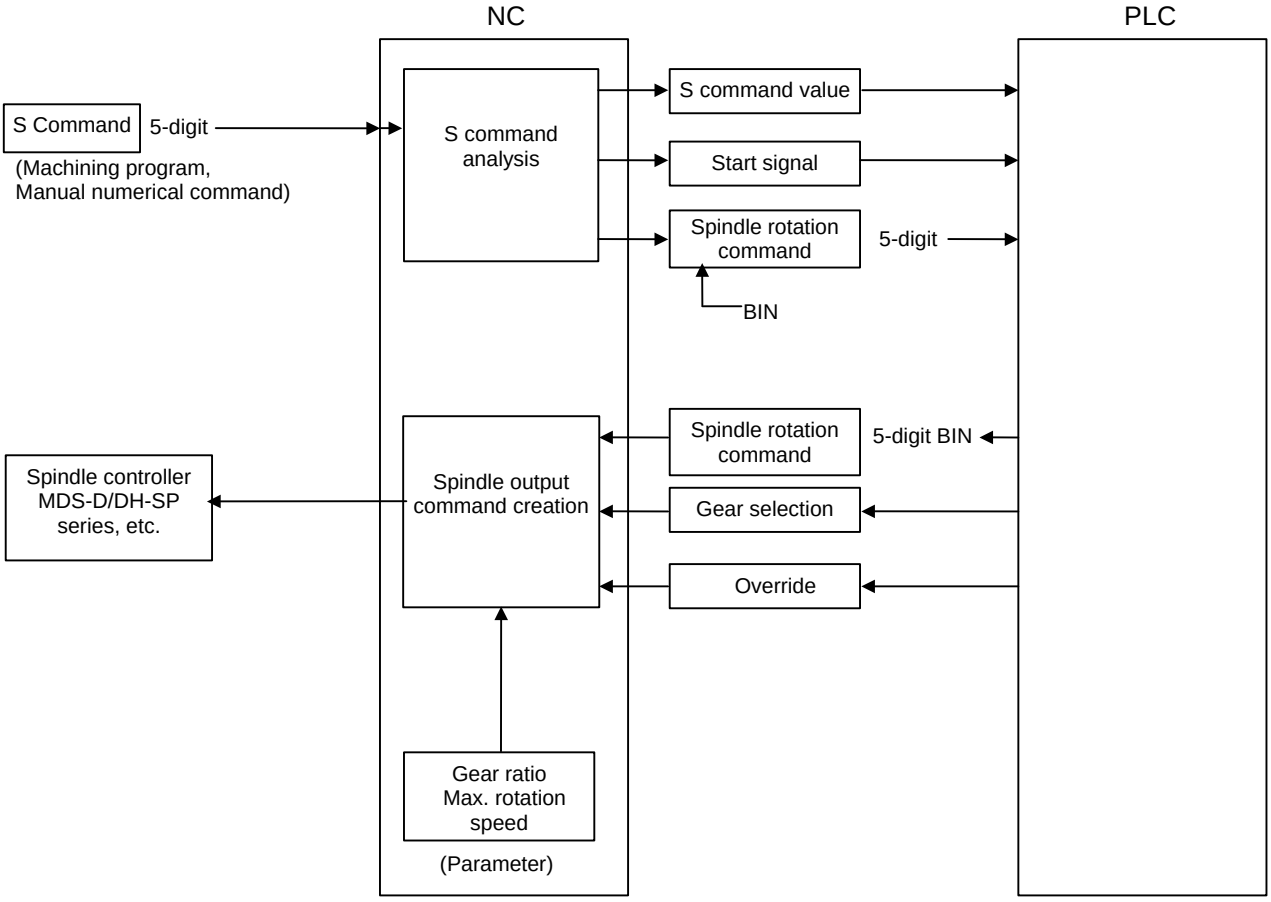
8. Spindle, Tool and Miscellaneous Functions

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 (S-99999999 to S99999999) is commanded, a signed 32-bit binary data and start signal will be output to the PLC.
When multiple spindle control "Sn = ****" method, up to seven sets of S commands can be commanded in one block.
Processing and complete sequences must be incorporated on the PLC side for all S commands.



(1) The override can be designated as 50% to 120% in 10% increments or 0 to 200% in 1% increments. 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.

(Note 1) S command can be commanded by eight digits. However, setting range of the parameter highest rotation speed and rotation speed limit, etc. are five digits or less. So, S command which can be substantially controlled are five digits or less.
(Note 2) The display of S command is five digits or less display on some screens.

8.1.1.1 Spindle Digital I/F

M system : ○ **L system : ○**

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

8.1.1.3 Coil Switch

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

M system : ○ **L system : ○**

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

8.1.2 S Code Output

M system : ○ **L system : ○**

When an 8-digit number following address S (S-99999999 to S99999999) is commanded, a 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 process (presence of DEN signal process), 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.

(Note) The display of S command is five digits or less display on some screens.

8.1.3 Constant Surface Speed Control

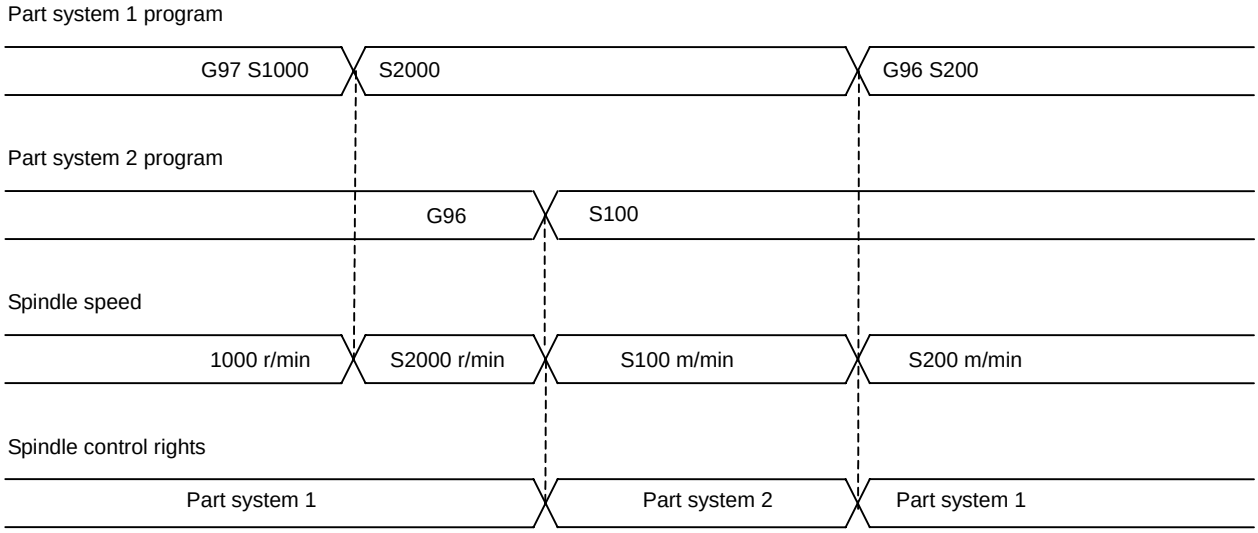
M system : Δ **L system : Δ**

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

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

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

(Note 1) If there is only one spindle, the spindle will not operate normally if the constant surface speed control command, S command or spindle related M command is commanded randomly from each part system. These commands must be commanded from only one certain part system, or commanded simultaneously with standby.
The controller will execute the following control for the constant surface speed control and S commands. The part system from which an S command was issued last will have the spindle control rights. That part system will judge whether the constant surface speed command mode is valid or canceled, and will execute spindle control.



8.1.4 Spindle Override

M system : O **L system : O**

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

(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) 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%.

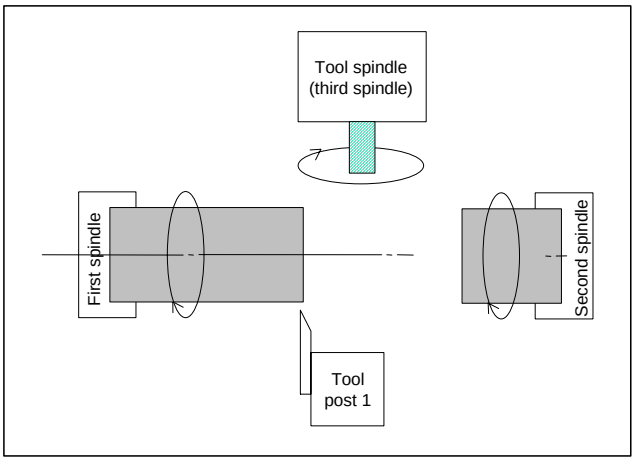
(Note 1) Selection between code method and value setting method can be designated by user PLC processing.

8.1.5 Multiple-spindle Control

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

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

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



8.1.5.1 Multiple-spindle Control I

M system : O **L system : O**

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

Command format

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

(2) Spindle control commands (using an extended word address (SO=******))
In addition to using the "S******" S commands, it is also possible to assign commands which differentiate the applicable spindle among the first through seventh spindles by using the SO=******.
The S command can be issued from a machining program for any part system.
The number of spindle axes differs according to the model, so check the specifications.
The C6 T and L System and C64 T System cannot control multiple spindles in one part system.

Command format

SO=****;**

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

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

8.1.6 Spindle Orientation

M system : O **L system : O**

(1) 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 point is the Z-phase position when using encoder orientation (PLG and external encoder) or the proximity switch neighborhood when using the proximity switch method.

(2) 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) Orientation is possible only when the gear ratio is 1:1 for the PLG orient. (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.)

8.1.8 Spindle Synchronization

8.1.8.1 Spindle Synchronization I

M system : Δ **L system : Δ**

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

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

Command format

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

G113;

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

G114.1 H__ D__ R__ A__ ;
H__ : Selects the basic spindle.
D__ : Selects the spindle to be synchronized with the basic spindle.
E__ : Designates the synchronized spindle phase shift amount.
A__ : Designates the spindle synchronization acceleration/deceleration time constant.

8.1.8.2 Spindle Synchronization II

M system : O **L system : O**

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

Whereas the spindle synchronization I executes the selection of the spindles to be synchronized, the start of the synchronization and other settings with G code in the machining program, this function designates all these 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.

8.1.11 Spindle Speed Clamp

M system : O **L system : O**

The spindle rotation speed is clamped between maximum rotation speed and minimum rotation speed.

8.2 Tool Functions (T)

8.2.1 Tool Functions (T Command)

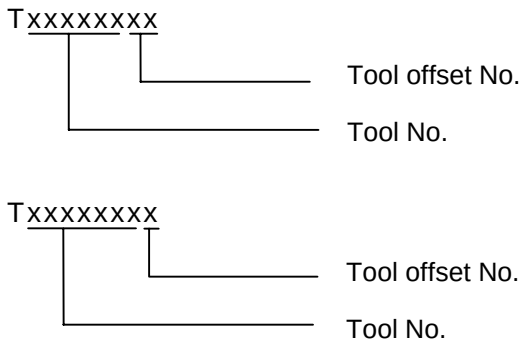
(1) M system

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

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

(2) L system

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



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

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

8.3 Miscellaneous Functions (M)

8.3.1 Miscellaneous Functions

M system : O **L system : O**

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

Apart from the above signals, various special independent signals are also output for the following signals.

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

Respective processing and complete sequences must be incorporated on the PLC side for all M commands from M00000000 to M99999999.
M98 and M99 have specific purposes and can not be used.

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

8.3.2 Multiple M Codes in 1 Block

M system : O **L system : O**

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

(Note 1) The code data and start signals of all the M commands in the same block are transferred simultaneously from the controller to the PLC, and so high-speed machine control can be done by the PLC processing sequence.

8.3.3 M Code Independent Output

M system : O **L system : O**

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

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

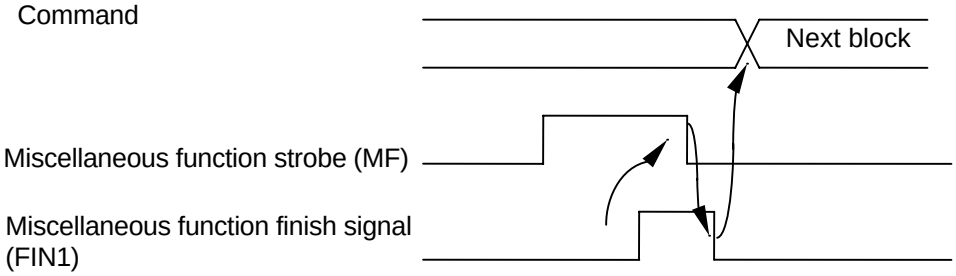
If movement or dwell command exists in the same block as these M commands, this signal is output upon completion of the movement or dwell command.

8.3.4 Miscellaneous Function Finish

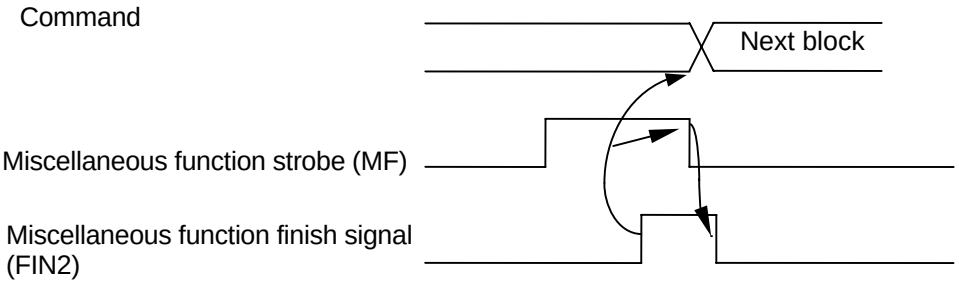
M system : O **L system : O**

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

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



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



8.4 2nd Miscellaneous Function (B)

8.4.1 2nd Miscellaneous Function

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) 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

M system : O **L system : O**

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

(1) M system

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

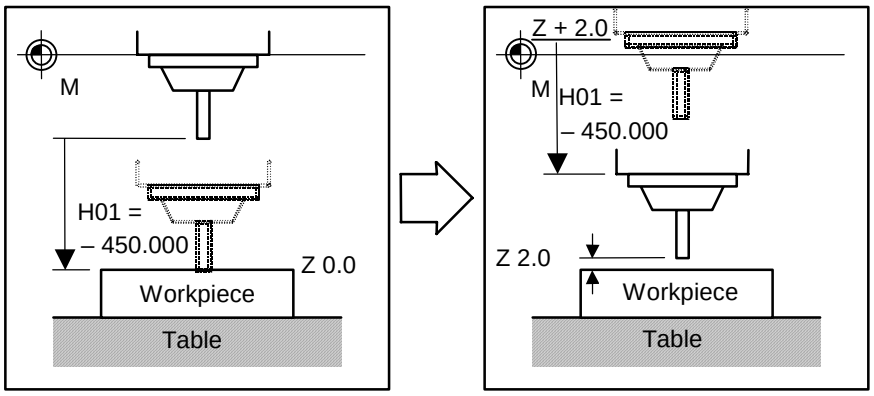
The offset direction is determined by the G command.
G43: Forward direction (z1 + h1)
G44: Reverse direction (z1 – h1)
Offset can be canceled by the following G commands.
G49;
G43 H0;
G44 H0;

(Note 1) When the tool length offset axis is returned to the reference point, the offset of that axis is canceled.

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

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

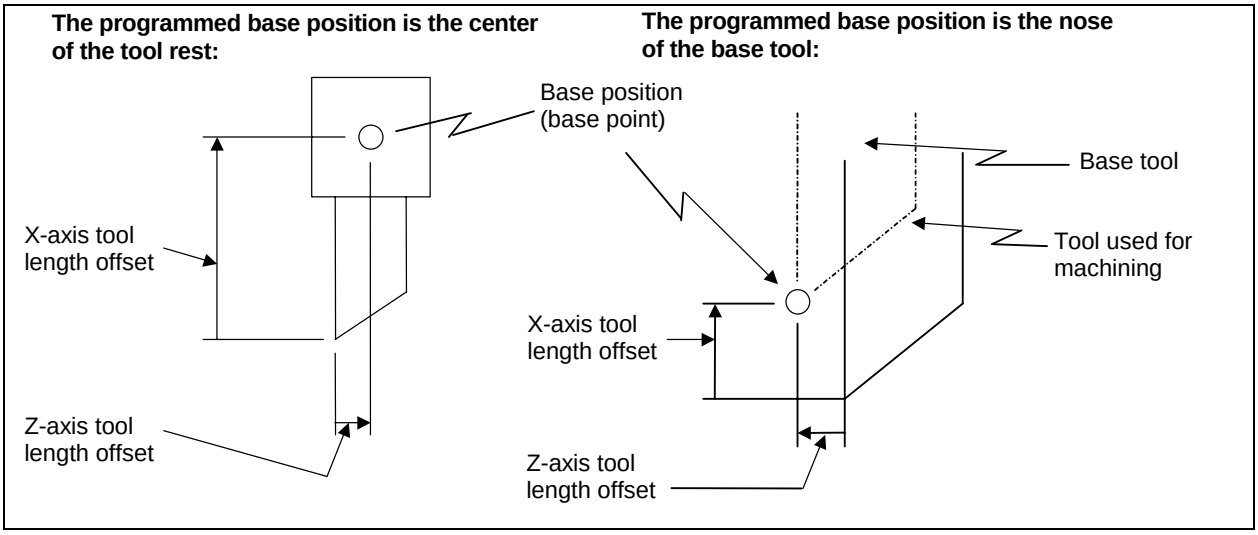
(Note) The tool length offset amount is set as a negative value such as H01 = –450.000.



(2) L system

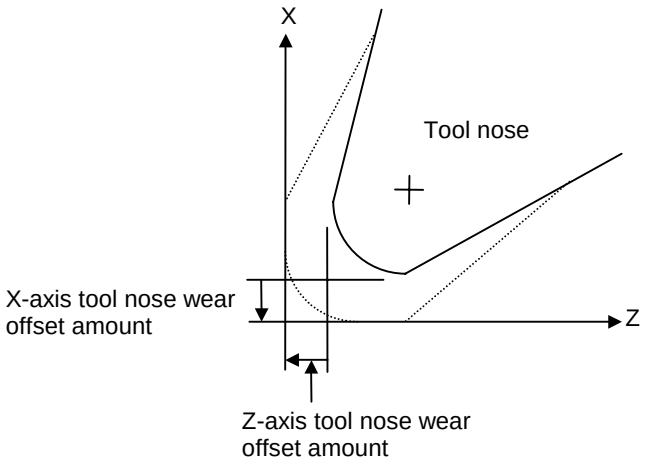
(a) Shape offset

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



(b) Wear offset

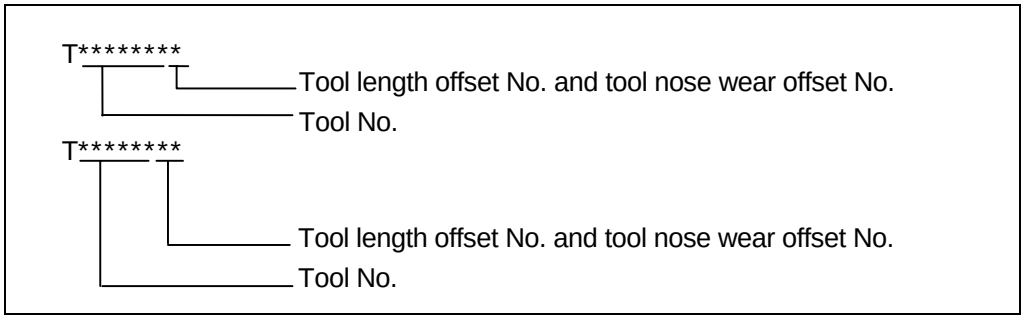
The wear of a tool nose can be offset.



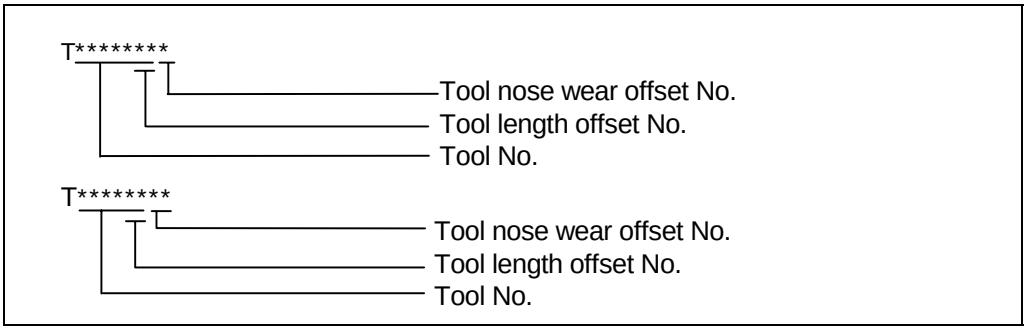
(c) Command format

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

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



2. Specifying tool length and wear offset Nos. separately



The tool offset for the L system is valid only for the X and Z axes.

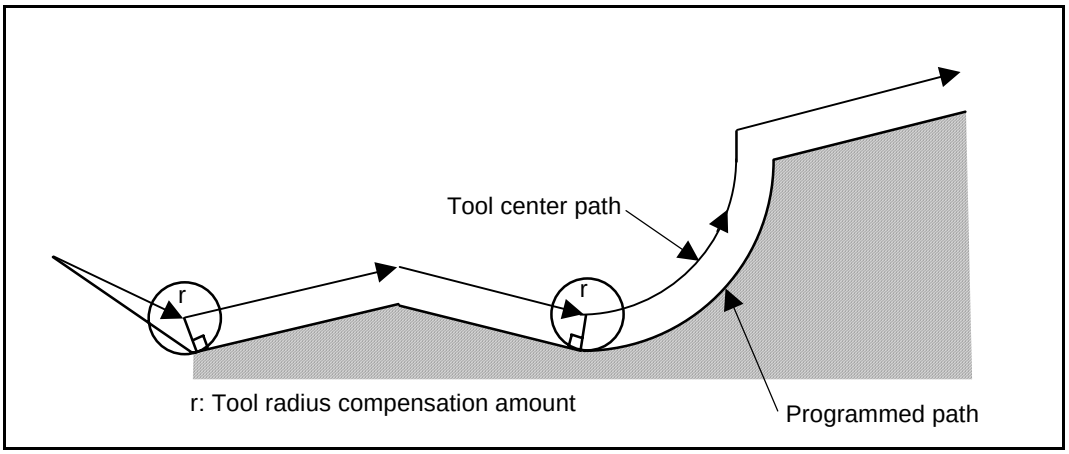
9.2 Tool Radius

9.2.1 Tool Radius Compensation

M system : O **L system : -**

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

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



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

G17 G01 G41 Xx1 Yy1 Dd1 ;

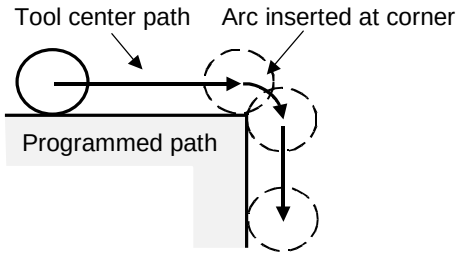
G17 : Compensation plane
G01 : Cutting command
G41 : Left compensation
Xx1,Yy1 : Movement axis
Dd1 : Compensation No.

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

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

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

G39 Xx1 Yy1 ;	
Xx1, Yy1	: Movement amount



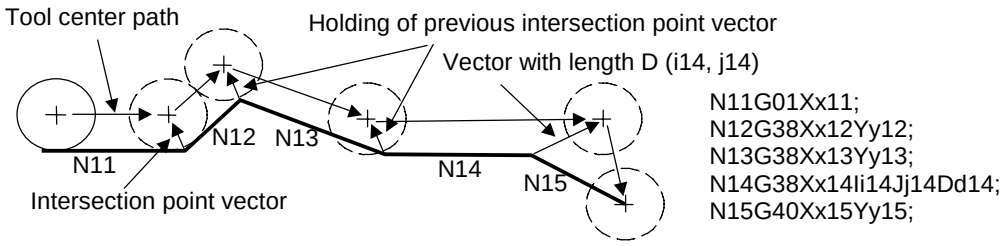
The compensation vector can be changed in following two ways.

G38 Xx1 Yy1 ;	
Xx1, Yy1	: Movement amount

The tool radius compensation vector amount and direction are retained.

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

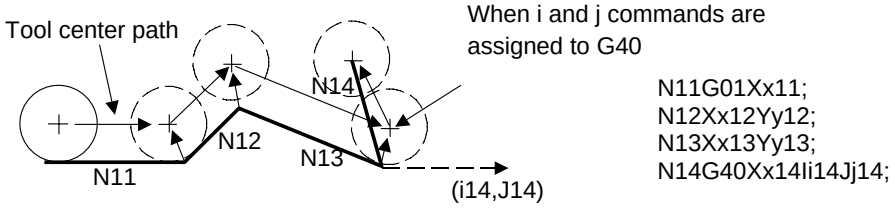
The tool radius compensation vector direction is updated by I and J.



The tool radius compensation is canceled by the following command.

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

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

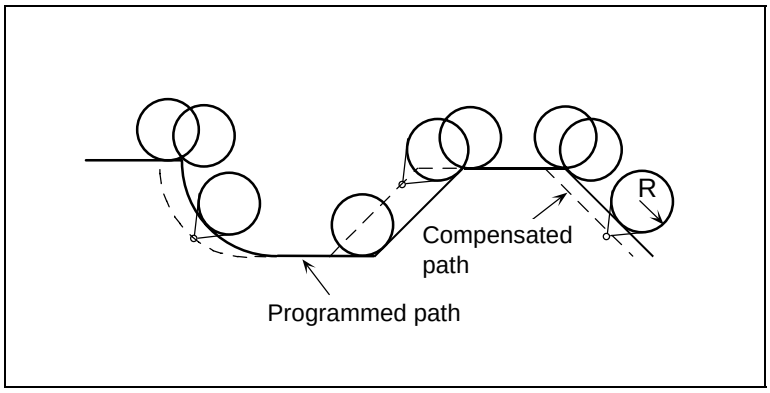


9.2.3 Tool Nose Radius Compensation (G40/G41/G42)

M system : - **L system : O**

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

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



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

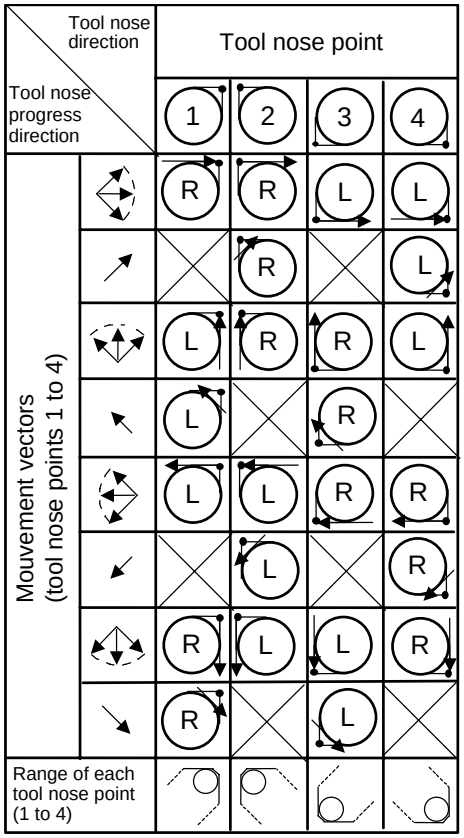
9.2.4 Automatic Decision of Nose Radius Compensation Direction (G46/G40)

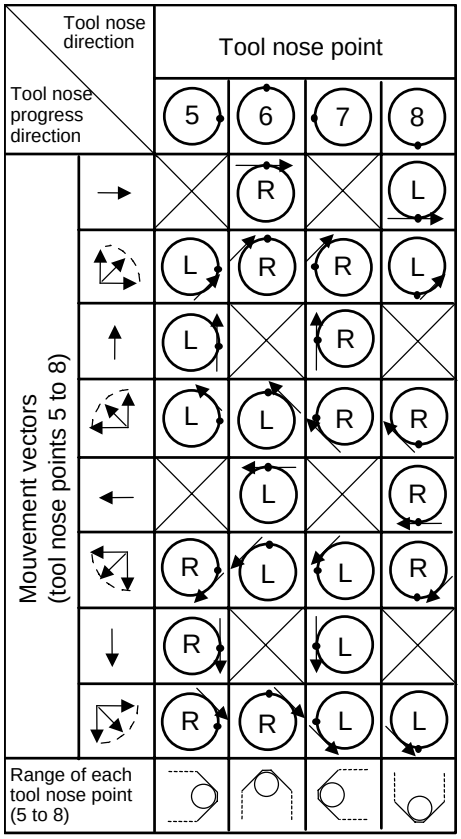
M system : - **L system : O**

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

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

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





9.3 Tool Offset Amount

9.3.1 Number of Tool Offset Sets

The number of tool offset sets is as follows.

9.3.1.2 40 sets

M system : ○ **L system : -**

9.3.1.3 80 sets

M system : △ (80/100) **L system : ○**

9.3.1.4 200 sets

M system : △ **L system : -**

The specifications of number of tool offset sets are following below.

Number of part systems / Tool offset sets for each part system	1st part system	2nd part system	3rd part system	4th part system	5th part system	6th part system	7th part system	Remarks
40 sets	40 sets	40 sets	40 sets	40 sets	40 sets	40 sets	40 sets	Standard for M system
80 sets	80 sets	80 sets	80 sets	40 sets	40 sets	40 sets	40 sets	
100 sets	100 sets	100 sets	80 sets *1	40 sets	40 sets	40 sets	40 sets	Standard for L system
200 sets	200 sets	100 sets	80 sets	40 sets	40 sets	40 sets	40 sets	

(Note 1) The number of sets in above table indicates the number of sets in each part system.
(Ex.) *1 indicates 80 sets for each part system in 3-part system when offset sets are addition specifications of 100 sets.
(Note 2) L system is three part systems or less.

9.3.2 Offset Memory

9.3.2.1 Tool Shape/Wear Offset Amount

M system : O **L system : O**

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

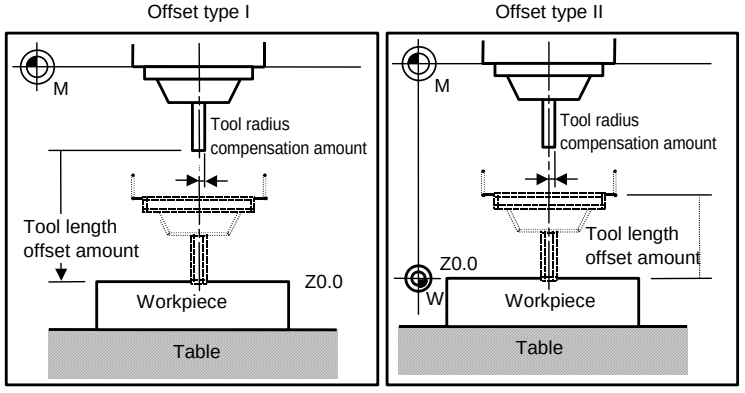
(1) Shape offset amount
The tool length offset amount, tool radius compensation amount, nose radius compensation amount, nose radius imaginary tool tip point or tool width can be set as the shape offset amount.
The compensation amount that can be set and used differs depending on whether offset amount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

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

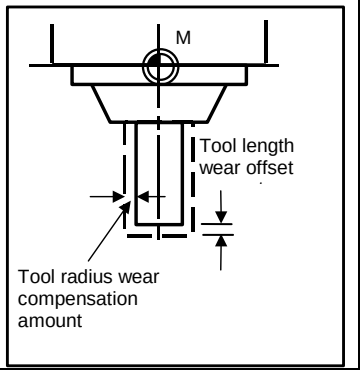
(a) Type 1: 1-axis offset amount [M system]
This is the value that is used by rotary tools.
As the tool length offset amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the longitudinal direction of the rotary tool is registered. The tool length offset amount is set as a minus value.
As the tool radius compensation amount, among the offset amounts for the position of the tool moving in the direction parallel to the control axis, the offset amount in the radial direction of the rotary tool is registered. The tool radius compensation amount is set as a plus value.
One offset amount data is registered in one offset number, and the offset Nos. are assigned using the address D or H commands. When a No. is assigned by a D address command, offset is provided in the form of the tool radius; when it is assigned by an H address command, it is provided in the form of the tool length.

(b) Type 2: 1-axis offset amounts/with wear offset [M system]
As with type 1, type 2 is for the offset amounts used by rotary tools.
With type 2, four kinds of offset amount data are registered in one offset No.: the tool length offset amount, tool length wear offset amount, tool radius compensation amount, and tool radius wear compensation amount.
When an offset No. is assigned by address D as the offset amount, the tool radius is compensated using the amount obtained by adding the tool radius compensation amount and tool radius wear compensation amount. Further, the tool length is offset using the amount obtained by adding the tool length offset amount and tool length wear offset amount.

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



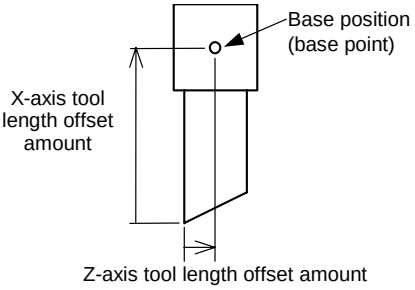
Wear offset amount when using type 2



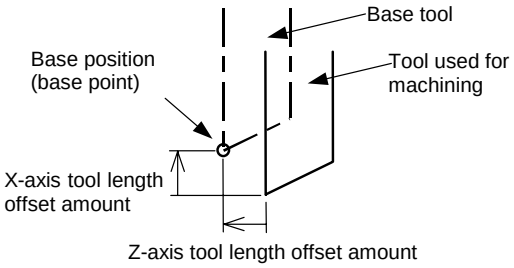
(c) Type 3: 2-axis offset amounts [L system]

Type 3 is for the offset amounts used by non-rotary tools.
As the offset amounts, the tool length along the X, Z axes and additional axis and the wear amount along each of these axes, the nose radius and nose radius wear amount, tool tip point P and tool width can be registered.
Offset is provided in the directions of the X, Z axes and additional axis from the base position in the program. Generally, the center of the tool rest or the tip of the base tool is used as the programmed base position.

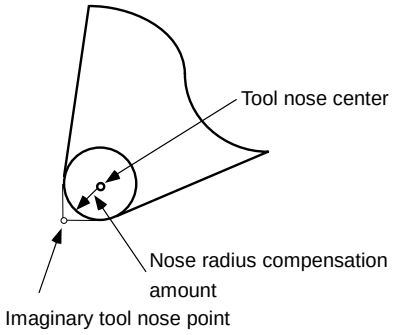
1. The programmed base position is the center of the tool rest:

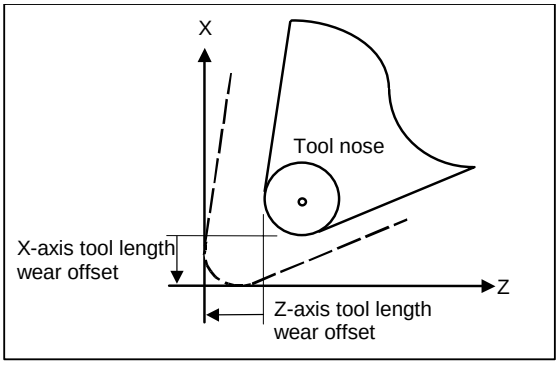


2. The programmed base position is the tip of the base tool:



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



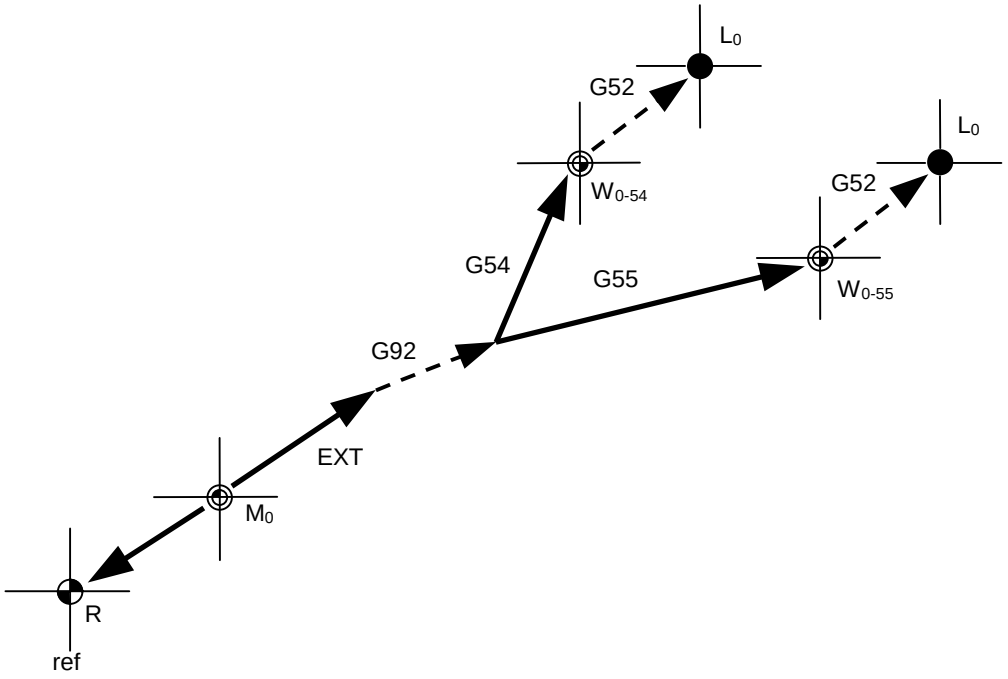


The X-axis tool length offset amount, Z-axis tool length offset amount and nose radius compensation amount are set as plus amounts.
The offset type (1, 2 or 3) is set using a parameter.

10. Coordinate System

10.1 Coordinate System Type and Setting

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



L_0	Local coordinate system zero point	⟶	Offset set with parameters
G52	Local coordinate system offset *1)	⇢	Offset set with program
W_{0-54}	Workpiece coordinate system zero point (G54)		(0 when power is turned ON)
W_{0-55}	Workpiece coordinate system zero point (G55)		
G54	Workpiece coordinate system (G54) offset *1)		*1)The G52 offset is available
G55	Workpiece coordinate system (G55) offset		independently for G54 to G59.
G92	G92 coordinate system shift		
EXT	External workpiece coordinate offset		
M_0	Machine coordinate system zero point		
ref	Reference point		

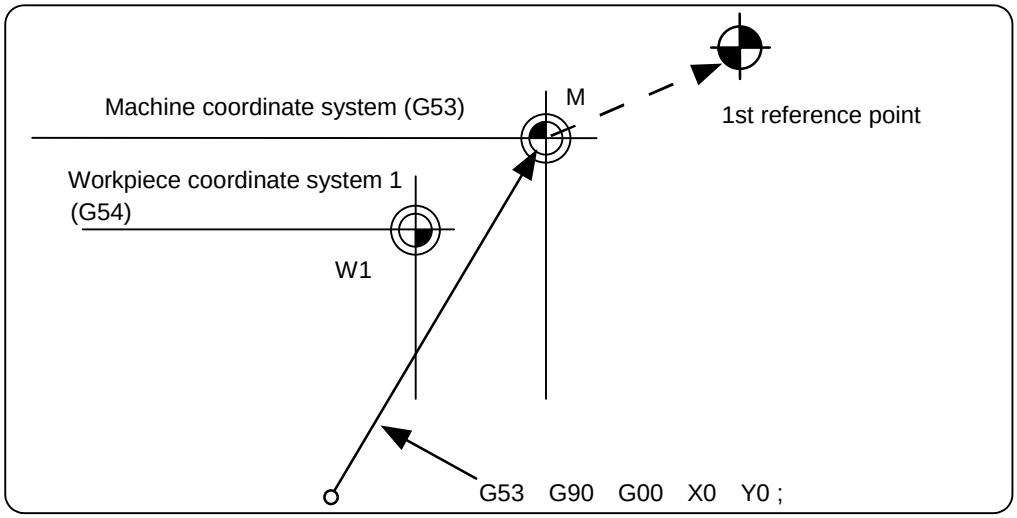
10.1.1 Machine Coordinate System

M system : O **L system : O**

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

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

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

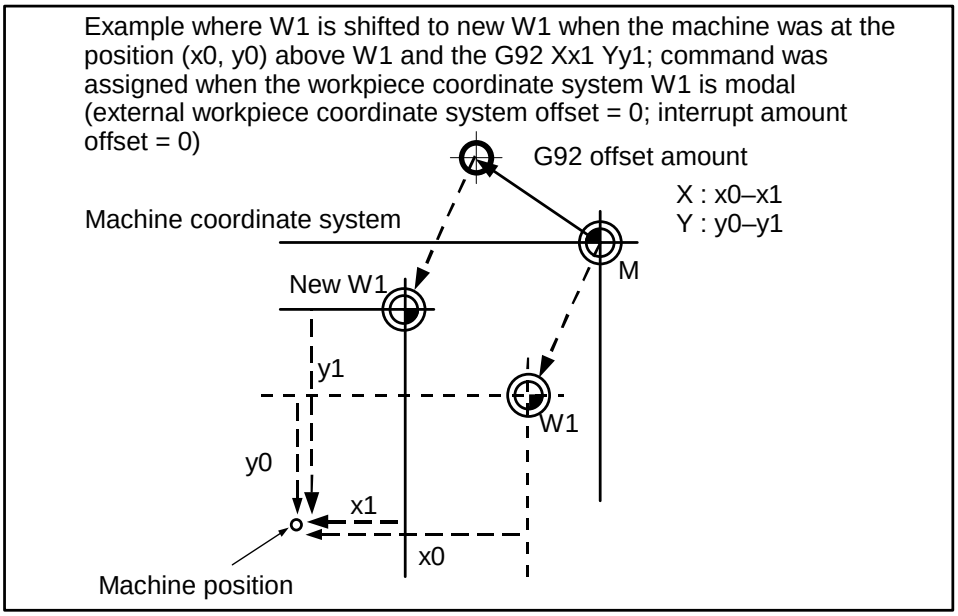


10.1.2 Coordinate System Setting

M system : O **L system : O**

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

Offset of coordinate system by G92 coordinate system setting

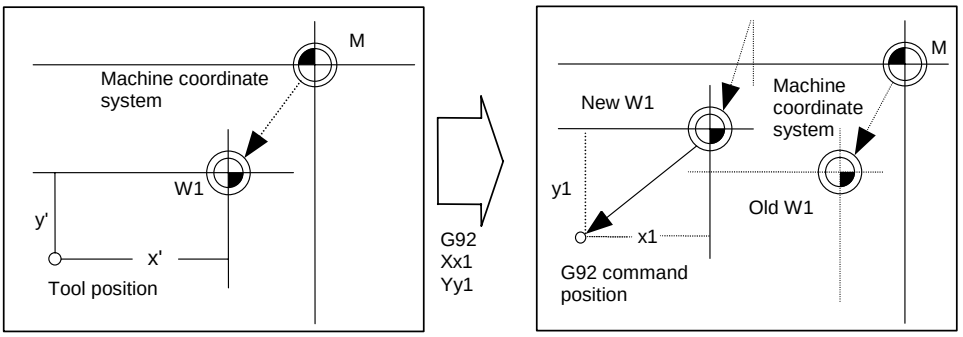


The shifted coordinate system is returned to its original position by dog-type reference point return or the program.

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

Coordinate system created by automatic coordinate system setting

Coordinate system after coordinate system setting by G92



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

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

10.1.3 Automatic Coordinate System Setting

M system : ○ **L system : ○**

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

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

The distances from the zero point of G53 machine coordinate system are set to the controller coordinate related parameters. Thus, where the No. 1 reference point is set in the machine is the base for the setting.

10.1.4 Workpiece Coordinate System Selection

(1) Workpiece coordinate system selection (6 sets)

M system : O **L system : O**

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.

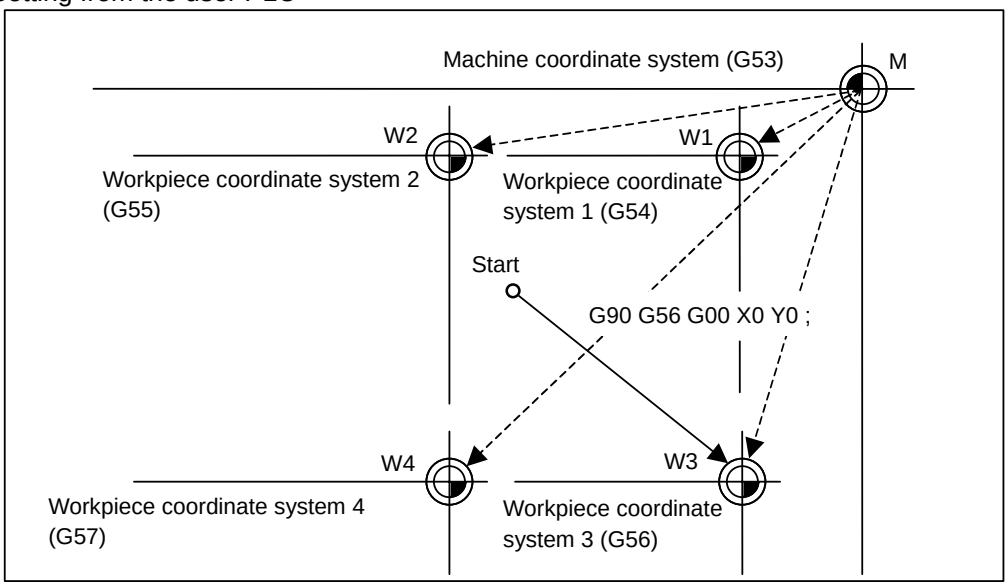
G code	Function
G54	Workpiece coordinate system 1 (W1)
G55	Workpiece coordinate system 2 (W2)
G56	Workpiece coordinate system 3 (W3)
G57	Workpiece coordinate system 4 (W4)
G58	Workpiece coordinate system 5 (W5)
G59	Workpiece coordinate system 6 (W6)

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

(G90) G54 G00 Xx1 Yy1 Zz1 ;
(G90) : (Absolute 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



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

M system : Δ **L system : -**

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

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

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

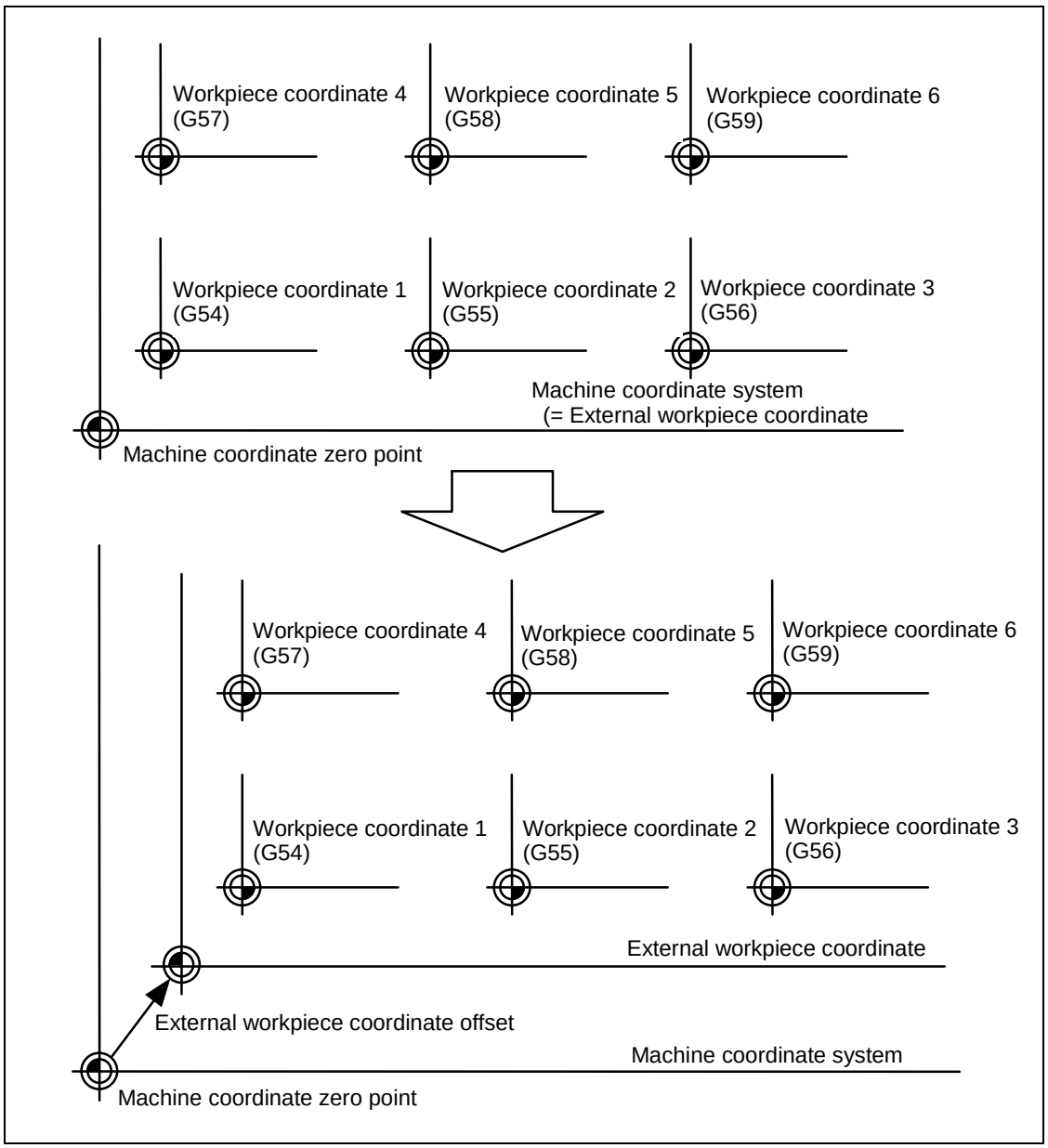
The workpiece coordinate zero points are provided as distances from the zero point of the machine coordinate system.
Settings can be performed in one of the following three ways:

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

10.1.5 External Workpiece Coordinate Offset

M system : O **L system : O**

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



10.1.7 Local Coordinate System

M system : O **L system : O**

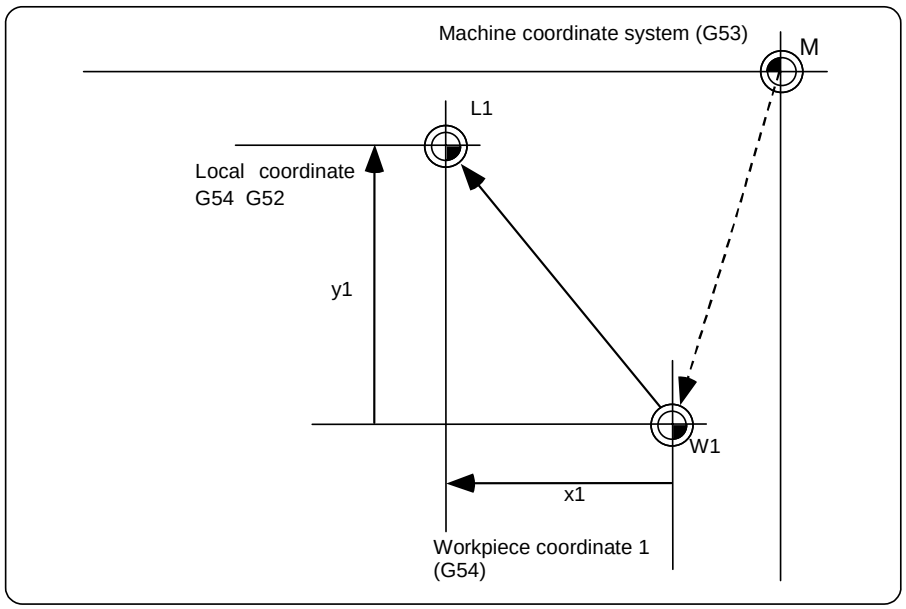
This function is for assigning a coordinate system on the workpiece coordinate system now being selected. This enables the workpiece coordinate system to be changed temporarily.
The local coordinate system can be selected independently on each workpiece coordinate system G54 to G59.

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

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

(G54) G52 Xx1 Yy1 Zz1 ;
(G54) : Workpiece coordinate system selection
G52 : Local coordinate system setting
Xx1, Yy1, Zz1 : Local coordinate offset amount

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



10.1.8 Coordinate System for Rotary Axis

M system : O **L system : O**

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

10.1.9 Plane Selection

M system : O **L system : O**

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

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

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

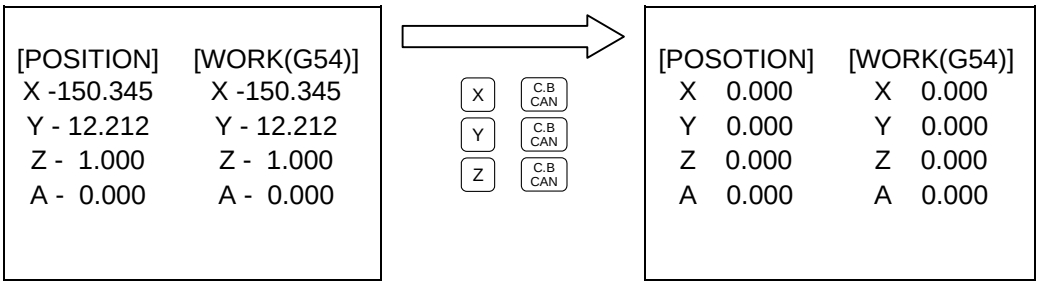
Example

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

10.1.10 Origin Set/Origin Cancel

M system : O **L system : O**

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



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

10.1.11 Counter Set

M system : O **L system : O**

Using the setting and display unit, the position counter display can be change to "0" by operating the screen.

(1) This operation is the same as the operation of "Origin Set", but press INPUT key instead of C.B CAN key.

(2) Only the [POSITION] counter display is changed to "0", and the other coordinate system counter displays are not changed.

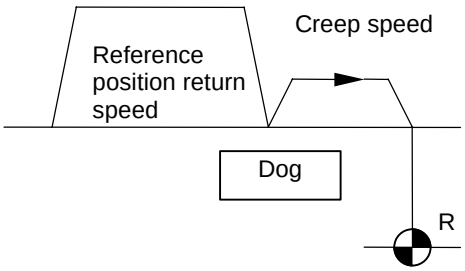
10.2 Return

10.2.1 Manual Reference Position Return

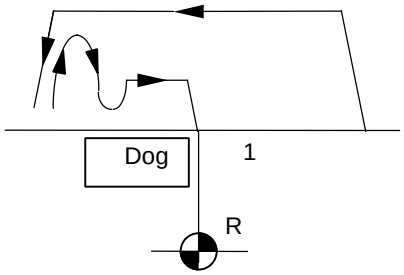
M system : ○ **L system : ○**

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

(a) Dog type

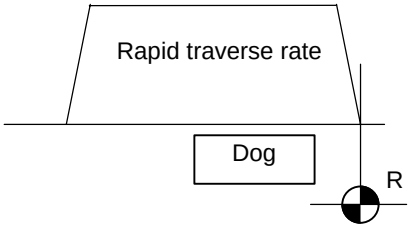


When starting in same direction as final advance direction



When starting in opposite direction as final advance direction

(b) High-speed type



(2) Differences according to detection method

	First return after power ON	Second return and following
Incremental position detection method	Dog-type	High-speed/Dog-type (switching by parameter)
Absolute position detection method	High-speed	High-speed

10.2.2 Automatic 1st Reference Position Return

M system : O **L system : O**

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

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

The G28 programming format is given below.

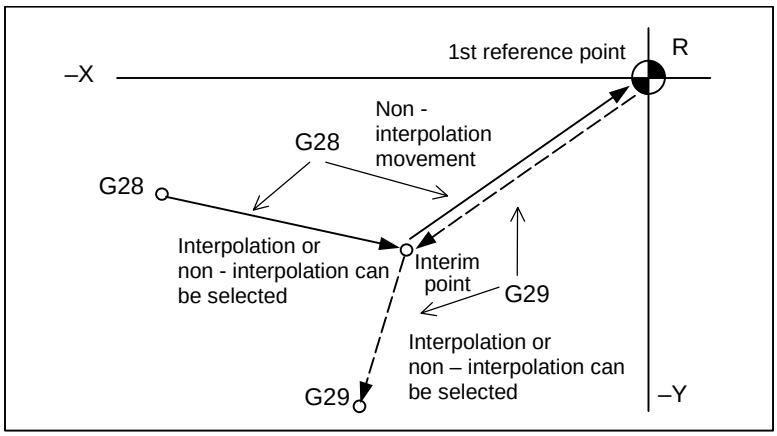
G28 Xx1 Yy1 Zz1 ;
G28 : Return command
Xx1, Yy1, Zz1 : Return control axes (interim point)

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

The G29 programming format is given below.

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

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



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

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

10.2.3 2nd, 3rd, 4th Reference Position Return; G30

M system : O **L system : O**

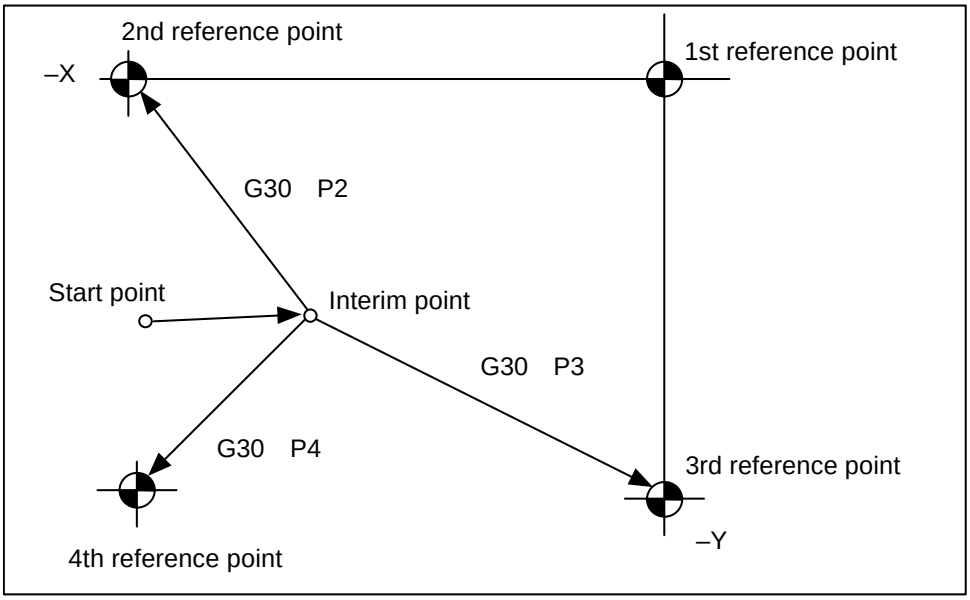
As with automatic 1st reference point return, commanding G30Pn during automatic operation enables the tool to be returned to the set points (2nd, 3rd or 4th reference points) characteristic to the machine. The 2nd, 3rd and 4th reference points can be set by parameters.

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

The G30 programming format is given below.

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

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



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

10.2.4 Reference Position Verification

M system : O **L system : O**

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

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

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

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

10.2.5 Absolute Position Detection

M system : Δ **L system : Δ**

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

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

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

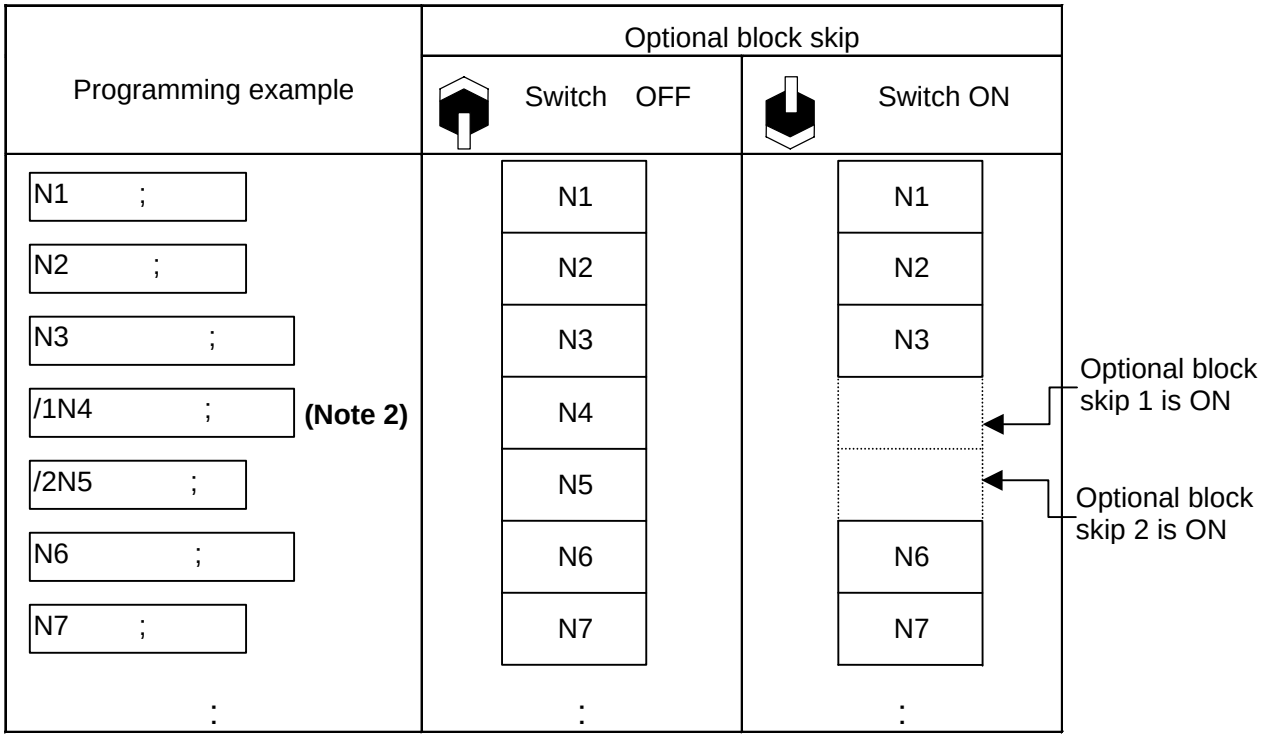
11. Operation Support Functions

11.1 Program Control

11.1.1 Optional Block Skip

M system : O **L system : O**

When "/n (n:1 to 9)" (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.

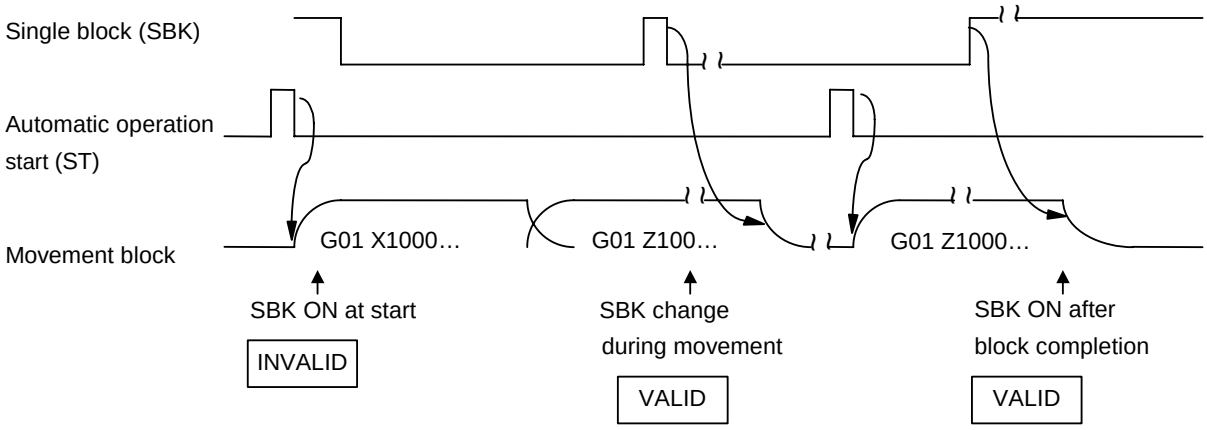


(Note 1) There are nine optional block skip switches corresponding to "/n (n:1 to 9)".
(Note 2) "1" of "1/N4" can be omitted.

11.1.2 Single Block

M system : O **L system : O**

The commands for automatic operation can be executed one block at a time (block stop) by turning ON the single block input signal. When the single block input signal is turned ON temporarily during continuous operation, the machine will stop after that block has been executed.
When operation is switched to another automatic operation mode (for example, memory operation mode to MDI operation mode) during continuous operation, the machine will stop after that block has been executed.
Single block in the multi-part system also functions as the above single block in each independent part system.



11.2 Program Test

11.2.1 Dry Run

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 selector switch OFF	Rapid traverse selector switch ON
G00, G27, G28, G29, G30, G60	Manual feed rate **(Note 1)**	Rapid traverse rate
G01, G02, G03	Manual feed rate	Cutting clamp speed

(Note 1) The dry run should be valid by the parameter setting.

11.2.2 Machine Lock

M system : ○ **L system : ○**

When the machine lock input signal is set to ON, the NC operations can be executed without actually moving the NC axis.
The command speed is the feed rate during machine lock.
Cutting override and rapid traverse override are valid.
The M, S, T and B commands are executed as usual, and so machine lock is completed by returning the FIN signal.

(1) Reference point return (manual, G28, G29, G30) is controlled as far as the interim point in the machine lock status but when the interim point is reached the counter is moved to the zero point and the block is completed.
(2) Machine lock is effective in the signal status applying when the axis has stopped.
(3) Block stop will be applied if the machine lock signal is turned ON and OFF or OFF and ON during automatic operation.
(4) On PLC programming, the signal for machine lock has for automatic operation and manual operation of each axis. Normally, all signals are simultaneously turned ON and OFF. However when Z axis cancellation function is executed, the machine lock signal for Z axis is turned ON and OFF.

11.2.3 Miscellaneous Function Lock

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.

11.3 Program Search/Start/Stop

11.3.1 Program Search

M system : O **L system : O**

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

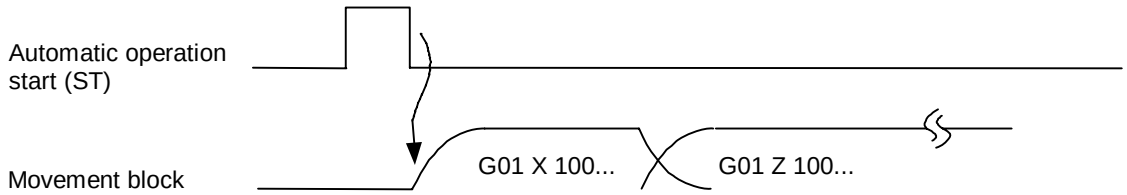
M system : O **L system : O**

Blocks can be indexed by setting the program No., sequence No. and block No. of the program to be operated automatically.
The searched program is displayed upon completion of the search.
Machining programs are stored in the memory inside the NC system.

11.3.5 Automatic Operation Start

M system : O **L system : O**

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

M system : O **L system : O**

This function enables the controller to be reset.

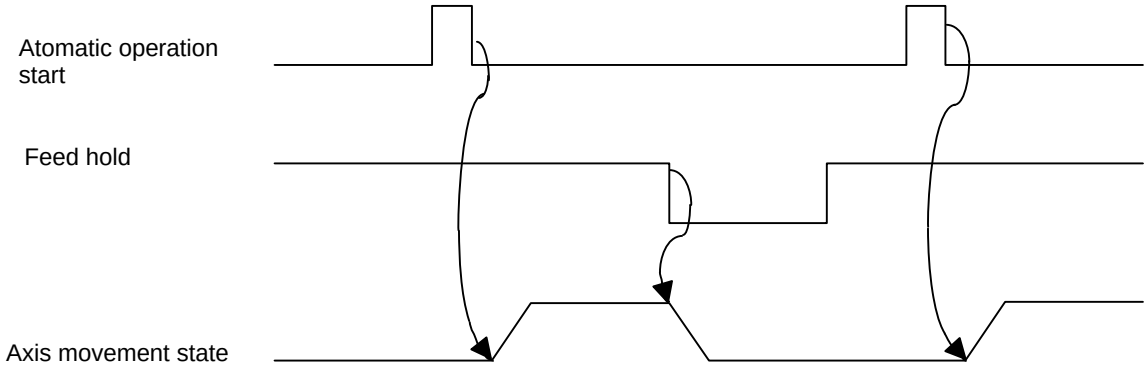
	Signal name / Target	Reset 1	Reset 2	Reset & Rewind
1	G command modals	Retained	Initialized	Initialized
2	Tool compensation data	Retained	Canceled (no operations)	Canceled
3	Memory indexing	Executed	Not executed	Executed
4	Errors/alarms	Reset	Reset	Reset
5	M, S and T code outputs	Retained	Retained	Retained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

11.3.7 Feed Hold

M system : O **L system : O**

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

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



11.3.8 Search & Start

M system : O **L system : O**

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.
The machining program No. to be searched are designated by PLC program.

11.4 Interrupt Operation

11.4.1 Manual Interruption

M system : O **L system : O**

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

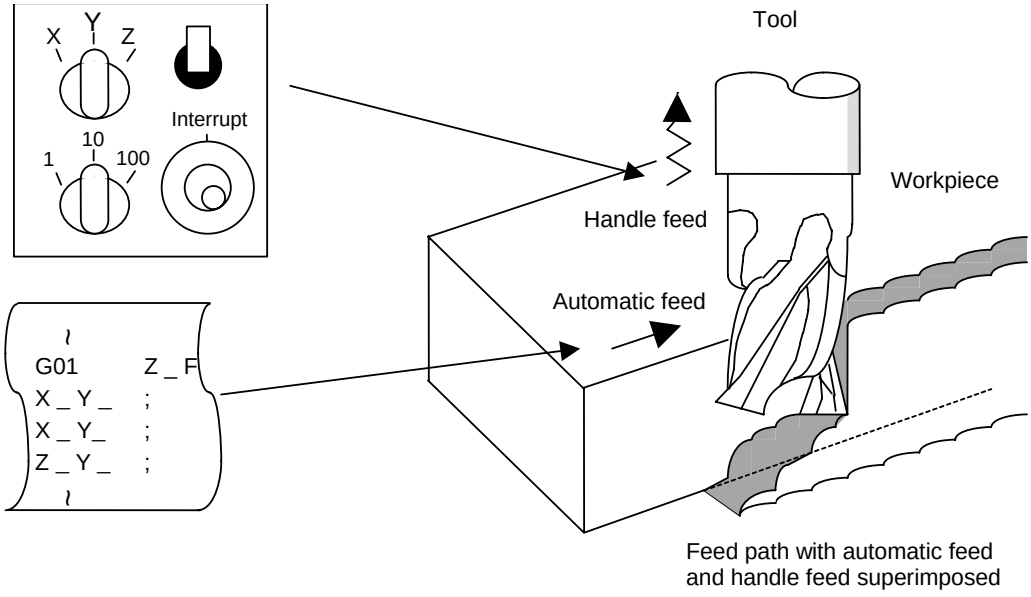
- System which initiates the interrupt by switching from the automatic mode to manual mode
- System which initiates the interrupt by selecting the manual mode at the same time as the automatic mode
  (Refer to "11.4.9 Simultaneous Operation of Manual and Automatic Modes".)

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

11.4.2 Automatic Operation Handle Interruption

M system : O **L system : O**

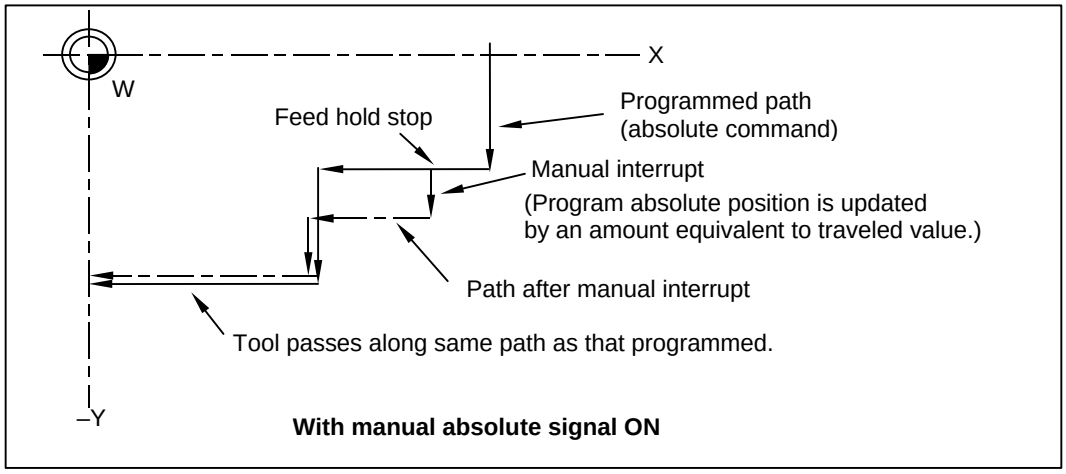
The handle command can interrupt and be superimposed onto a command without suspending automatic operation and the machine can be moved by rotating the manual pulse generator during automatic operation. If the spindle load is greatly exceeded when cutting a workpiece as per the machining program due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



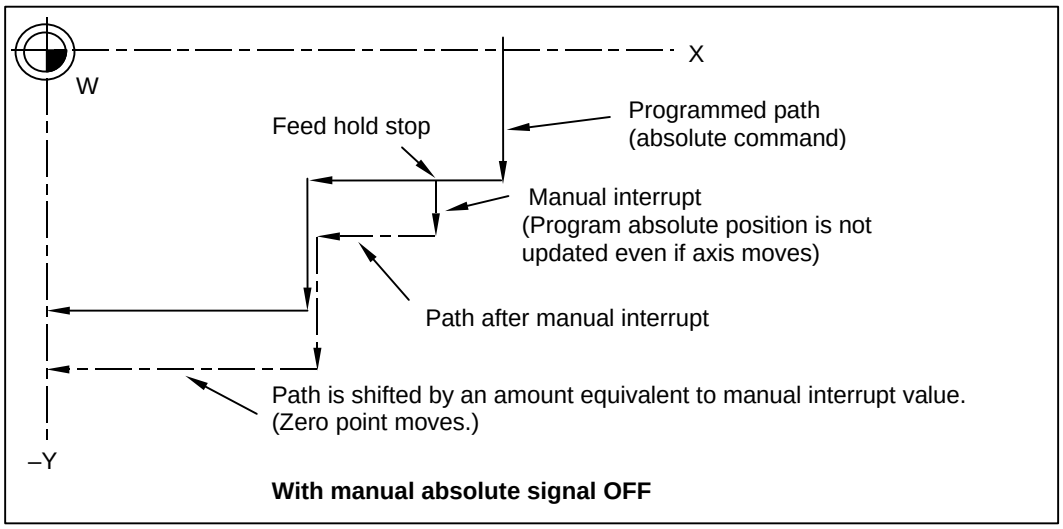
11.4.3 Manual Absolute Switch

M system : O **L system : O**

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.



With manual absolute signal ON

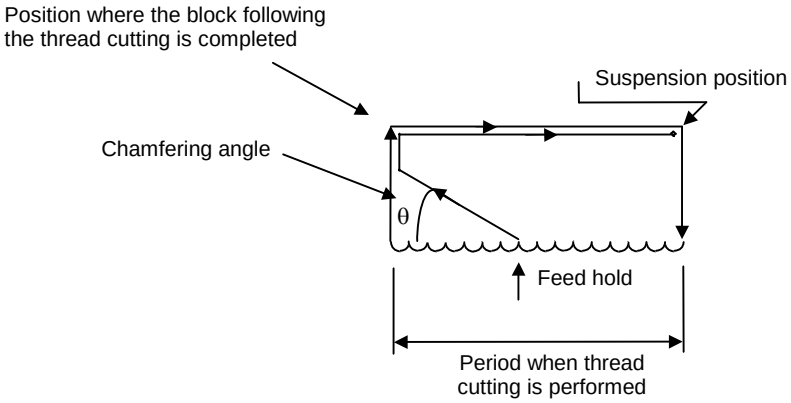


With manual absolute signal OFF

11.4.4 Thread Cutting Cycle Retract

M system : - **L system : Δ**

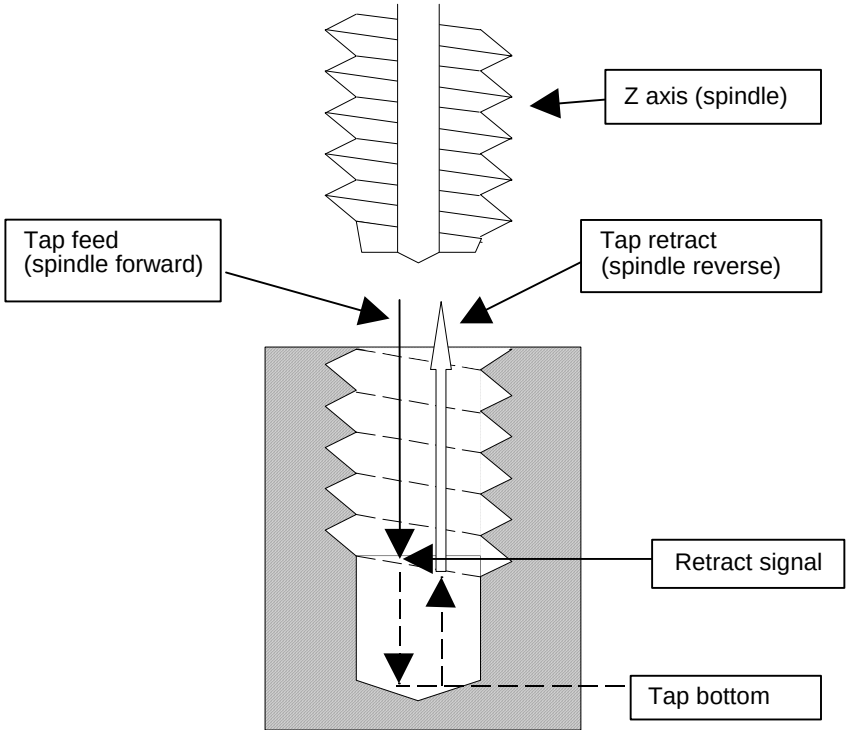
This function suspends the thread cutting cycle if a feed hold signal has been input during thread cutting in a thread cutting cycle.
If a feed hold signal is input during chamfering or thread cutting without chamfering, operation stops at the position where the block following the thread cutting is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



11.4.5 Tapping Retract

M system : O **L system : O**

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

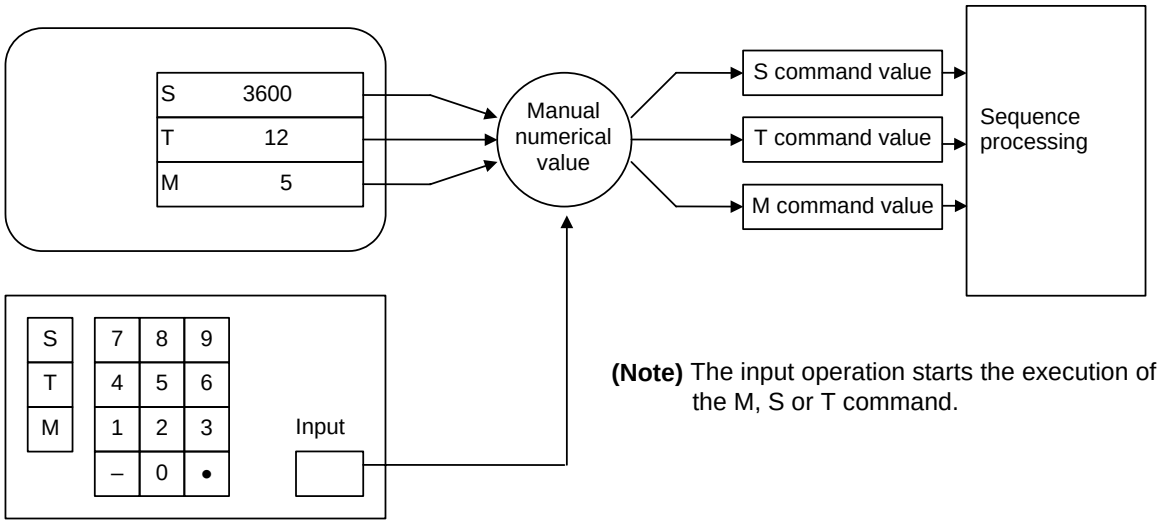


This function can be used by an interruption initiated by reset or emergency stop.
A return is made to the initial point by tap retract.

11.4.6 Manual Numerical Value Command

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 offset amount setting and display screen, therefore at the manual tool length measurement, the tools can be called successively to the spindle and measured very simply without having to change the screen page.



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

11.4.8 MDI Interruption

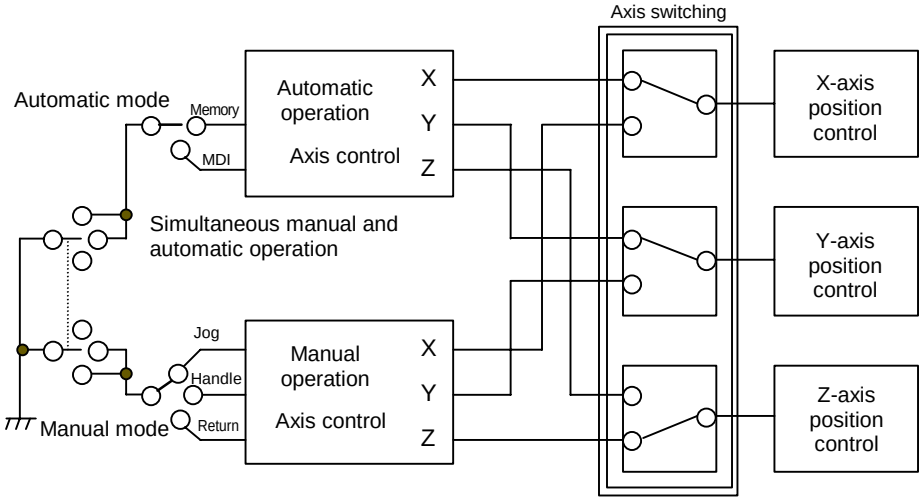
M system : ○ **L system : ○**

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

M system : O **L system : O**

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



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

12. Program Support Functions

12.1 Machining Method Support Functions

12.1.1 Program

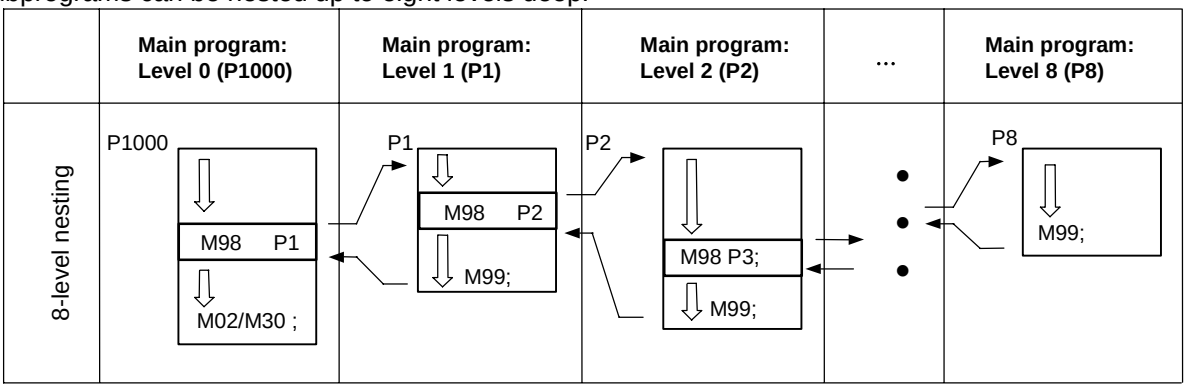
12.1.1.1 Subprogram Control

M system : O 8 layers L system : O 8 layers

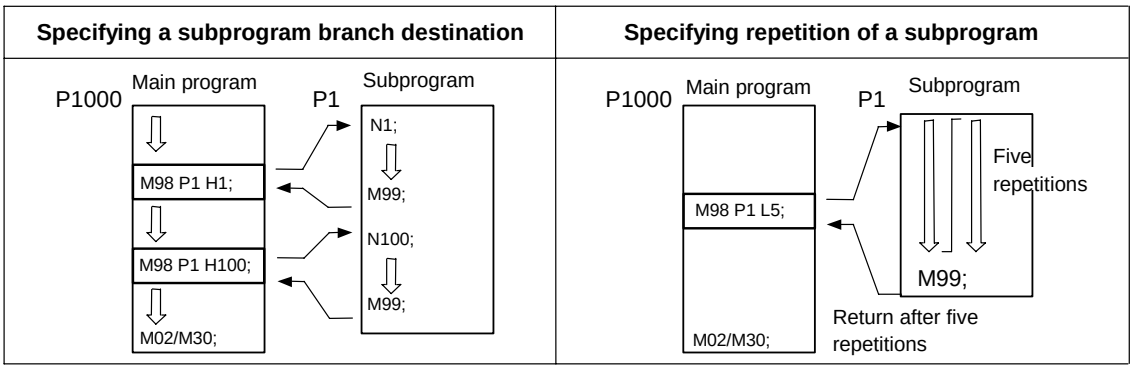
When the same pattern is repeated during machining, the machining pattern is registered as one subprogram and the subprogram is called from the main program as required, thereby realizing the same machining easily. Efficient use of program can be made. The call is designated with the program number and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

M98 Pp1 Hh1 Ll1 ;
M98 : Call command
Pp1 : Subprogram number
Hh1 : Sequence number
Ll1 : Number of repetitions
(Branch to subprogram)
Op1 (Subprogram)
:
Nh1
:
M99 ; (Return to main program)

Subprograms can be nested up to eight levels deep.



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



12.1.2 Macro Program

12.1.2.1 User Macro

M system : Δ 4 layers L system : Δ 4 layers

(1) Macro commands (1) ; G65 to G67
In order to carry through one integrated function, a group of control and arithmetic instructions can be used and registered as a macro program. Furthermore, subprograms with a high degree of expandability can be configured by setting these macro programs as types which are capable of conducting control and arithmetic operations using variable commands.

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

The program formats are given below.

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

The macro program is called immediately by this command.

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

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

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

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

The following macro command functions are available.

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

(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

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.)

(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

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

The correspondence between the Mm code which performs macro call and the program number for the macro to be called is set by a parameter. Up to 10 M codes from M00 to M95 can be entered. Select codes to be entered which are not the codes basically required by the machine and which are not M codes M0, M1, M2, M30 and M96 through M99.

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

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

12.1.2.3 Macro Interruption

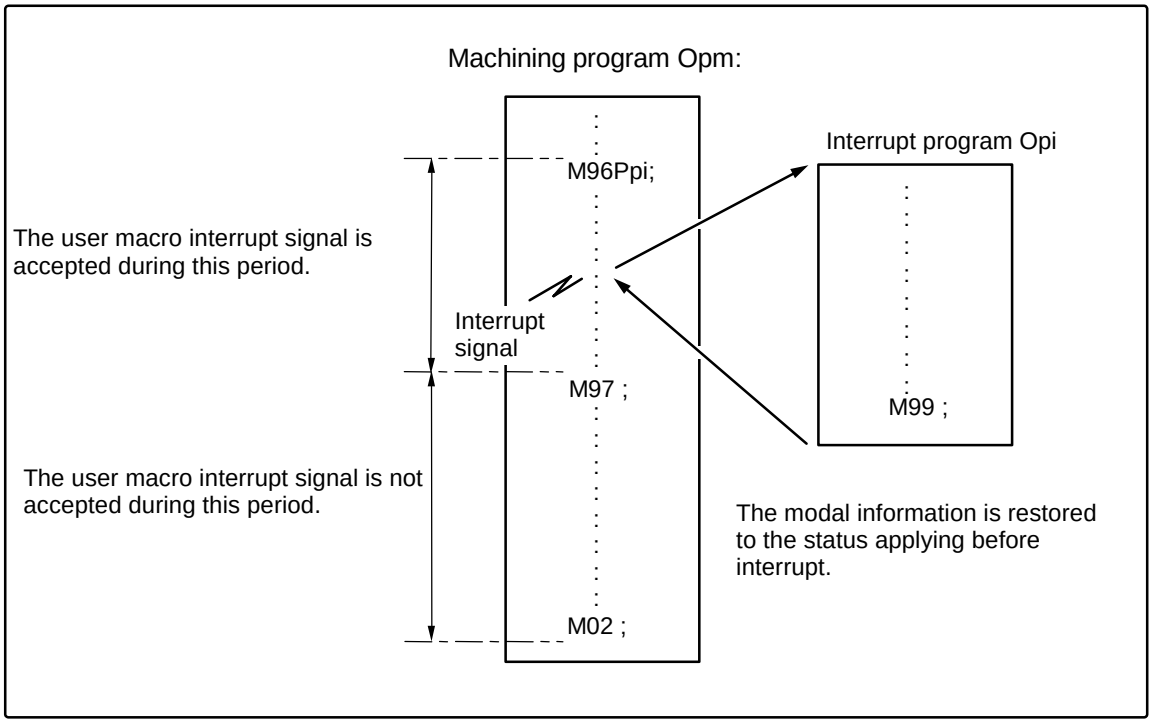
M system : Δ **L system : Δ**

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

[Interrupt type 1] The block being executed is immediately interrupted, and the interrupt program is run immediately.
[Interrupt type 2] After the block being executed is complete, the interrupt program is executed.

The command format is given below.

M96 P__ H__ ;	**User macro interrupt valid**
M97 ;	**User macro interrupt invalid**
P	: Interrupt program No.
H	: Interrupt sequence No.



12.1.2.4 Variable Command

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

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

Variable set option			Number		Function
Common variables			Variables common to all part systems	Variables for each part system	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	
	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	
		400 + 100 sets	#500 to #899	#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) Variable names can be set for #500 to #519.

Variable expressions

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

Variable definition
Variable = expression

(Note 4) Variables cannot be used with addresses "O" and "N".

12.1.2.4.1 100 Sets

M system : ○ **L system : ○**

12.1.2.4.2 200 Sets

M system : Δ **L system : Δ**

12.1.2.4.3 300 Sets

M system : Δ **L system : Δ**

12.1.2.4.4 600 Sets

M system : Δ **L system : Δ**

12.1.2.4.5 (50+50 × Number of Part Systems) Sets

M system : ○ **L system : ○**

12.1.2.4.6 (100+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.4.7 (200+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.4.8 (400+100 × Number of Part Systems) Sets

M system : Δ **L system : Δ**

12.1.2.101 N Code Macro

M system : Δ **L system : Δ**

This function calls the macro program by pre-registered N code.
The N No. and the macro program are registered by the parameter setting, and can be registered up to eight.
Argument (P, N, L, G) which cannot be used by a usual calling macro can be used.
In addition, the argument G can be used up to four.

(1) Macro call by N code

Format

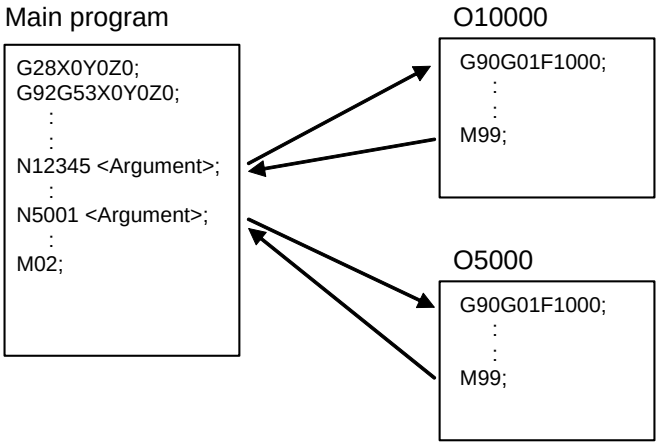
N***	**<Argument> ;**
N*****	: N code for performing macro call

(a) The macro is called by N code. (The calling is same as G65.)
(b) The called N No. is registered by the parameter setting. Wild-card (,) can be used for N No. registration.

Example for setting

	<Code>	<Program No.>		
N[01]	12345	10000	…	N No. : 12345, program No. : 10000
N[02]	5,,,	5000	…	To N No. 5000 to 5999, for program No. : 5000

(c) N code call diverges to the macro as the arguments entire address data in same block, and returns to the head of the next block.
(d) The macro subprogram can be called in up to four levels using N code macro call.



(Note 1) When prohibiting the display and edit of the macro program, set the macro program No. to O9000 to O9999, and change properly the parameter setting of edit lock C and the program display lock.

12.1.3 Fixed Cycle

List of fixed cycles

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

12.1.3.1 Fixed Cycle for Drilling

M system : O **L system : O**

(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.

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

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

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

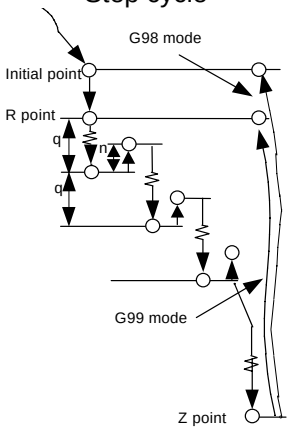
The basic program format for the fixed cycle commands is shown below.

G81 Xx1 Yy1 Zz1 Rr1 Qq1 Pp1 Ll1 Ff1 ;

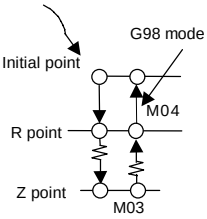
G81	:	Hole drilling mode
Xx1, Yy1	:	Hole position data; X-axis, Y-axis hole drilling position command (rapid traverse) (incremental/absolute)
Zz1	:	Hole machining data; Hole bottom position designation (incremental/absolute)
Rr1	:	Hole machining data; Hole R point designation (incremental/absolute)
Qq1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

For details on the synchronous tapping cycle, refer to the section "4.5.3 Synchronous Tapping".

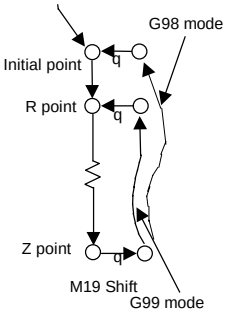
G73
Step cycle



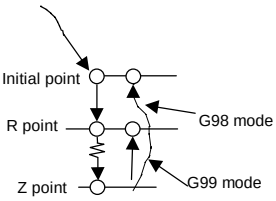
G74
Reverse tapping cycle



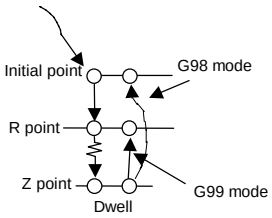
G76
Fine boring cycle



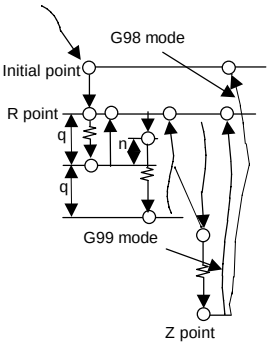
G81
Drilling, spot drilling cycle



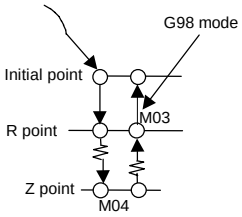
G82
Drilling, counterboring cycle



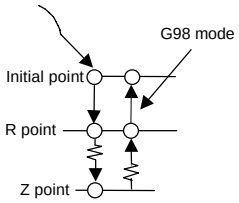
G83
Deep hole drilling cycle



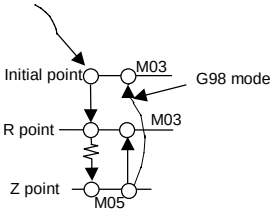
G84
Tapping cycle



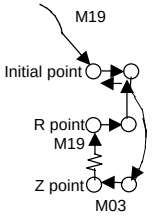
G85
Boring cycle



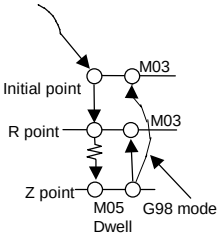
G86
Boring cycle



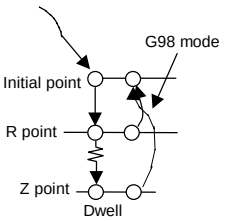
G87
Back boring cycle



G88
Boring cycle



G89
Boring cycle



(2) L system; G83 to G89, G80

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

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

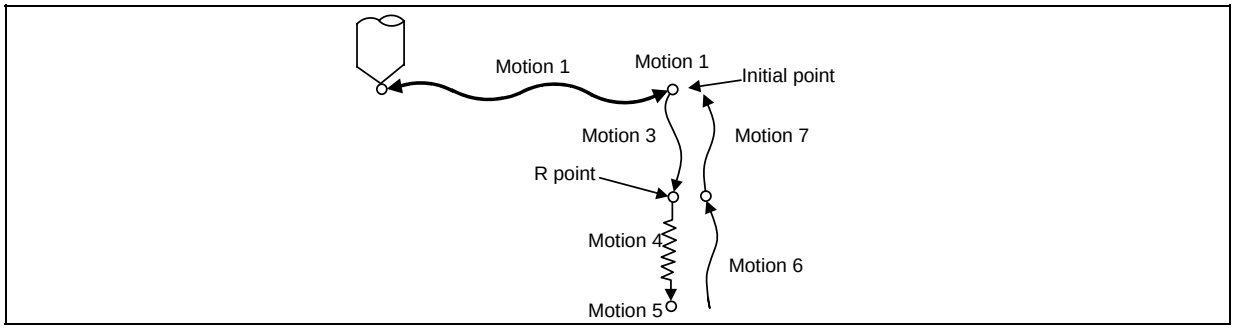
The fixed cycle mode is canceled when a G command of the G80 or G01 group is specified. Data is also cleared simultaneously.

Command format

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

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

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



Motion 1 : Rapid positioning up to the initial point of X (Z) and C axes.
If the "positioning axis in-position width" is designated, the in-position check is conducted upon completion of the block.
Motion 2 : Output if the C axis clamp M code is given.
Motion 3 : Rapid positioning up to the R point.
Motion 4 : Hole machining at cutting feed.
If the "drilling axis in-position width" is designated, the in-position check is conducted upon completion of the block. However, in the case of deep-hole drilling cycles 1 and 2, the in-position check is not conducted with the drilling of any holes except the last one. The in-position check is conducted at the commanded hole bottom position (last hole drilling).
Motion 5 : Motion at the hole bottom position. It varies depending on the fixed cycle mode. Spindle CCW (M04), spindle CW (M03), dwell, etc., are included.
Motion 6: Return to the R point.
Motion 7: Return to the initial point at rapid traverse feed.
(Operations 6 and 5 may be conducted as a single operation depending on the fixed cycle mode.)

(Note 1) With a synchronous tap command, the in-position check is conducted in accordance with the parameters.

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

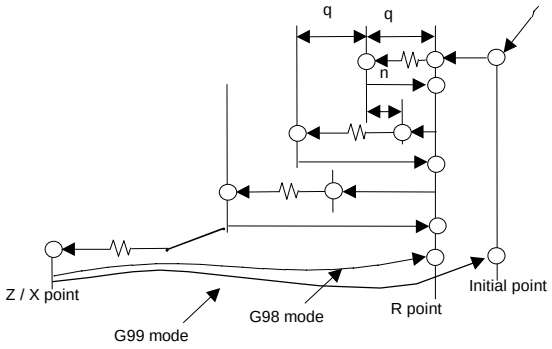
G98: Initial level return
G99: R point level return

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

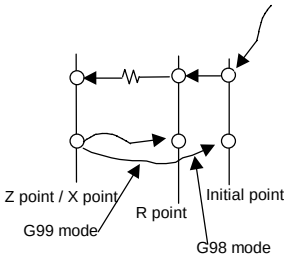
Deep-hole drilling cycle (G83, G87)

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

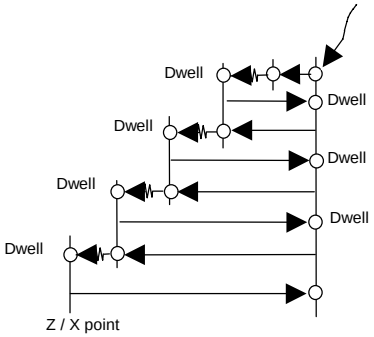
When Q command is given



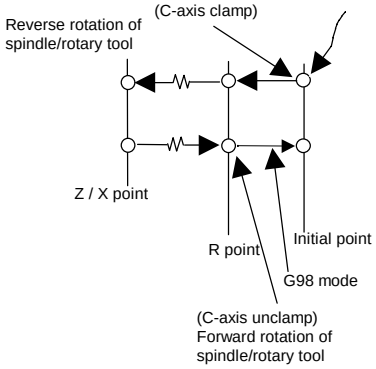
When Q command is not given



G83.2
Deep-hole drilling cycle

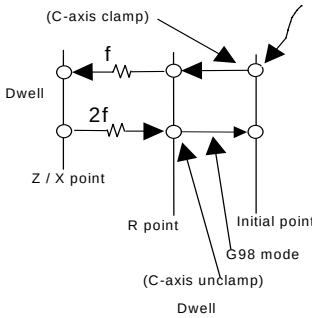


G84/88
Tapping cycle



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

G85/89
Boring cycle



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

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

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

12.1.3.3 Special Fixed Cycle

M system : Δ **L system : -**

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

The special fixed cycles are as follows:

G code	Function
G34	Bolt hole circle
G35	Line at angle
G36	Arc
G37.1	Grid

(1) Bolt hole circle (G34)

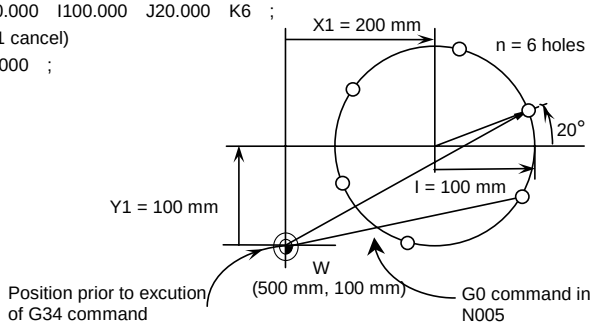
The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius R whose center is the coordinates designated by X and Y, and it drills "n" number of holes at "n" equal intervals along the circumference of that circle. The drilling data for the standard fixed 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 input increment and is provided using a positive number.
Jθ	:	Angle θ at point to be drilled initially; the counterclockwise direction is taken to be positive.
Kn	:	Number "n" of holes to be drilled; any number of holes from 1 through 9999 can be designated; 0 cannot be assigned. When 0 has been designated, the alarm will occur. A positive number provides positioning in the counterclockwise direction; a negative number provides positioning in the clockwise direction.

(Example)

With 0.001mm least input increment

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



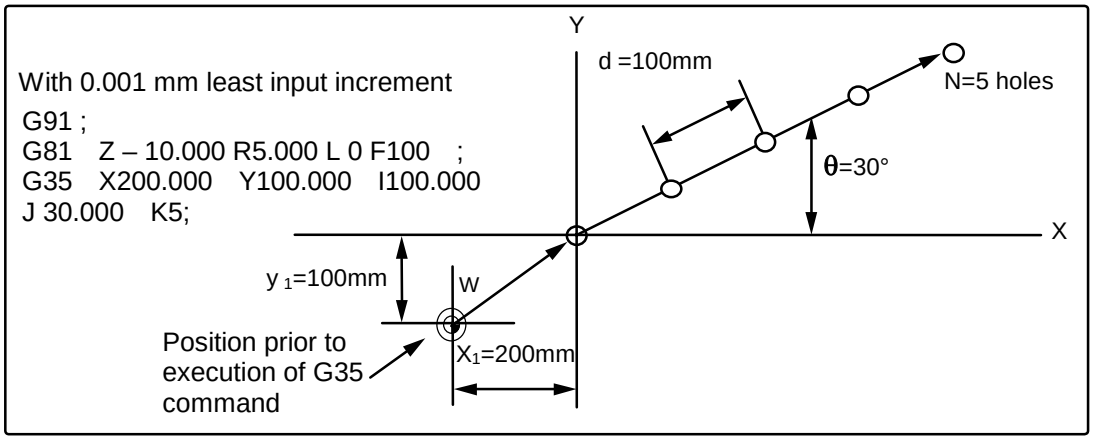
As shown in the figure, the tool is positioned above the final hole upon completion of the G34 command. This means that when it is to be moved to the next position, it will be necessary to calculate the coordinates in order to issue the command or commands with incremental values, and so it is convenient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

(2) Line at angle (G35)

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

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

(Example)



(3) Arc (G36)

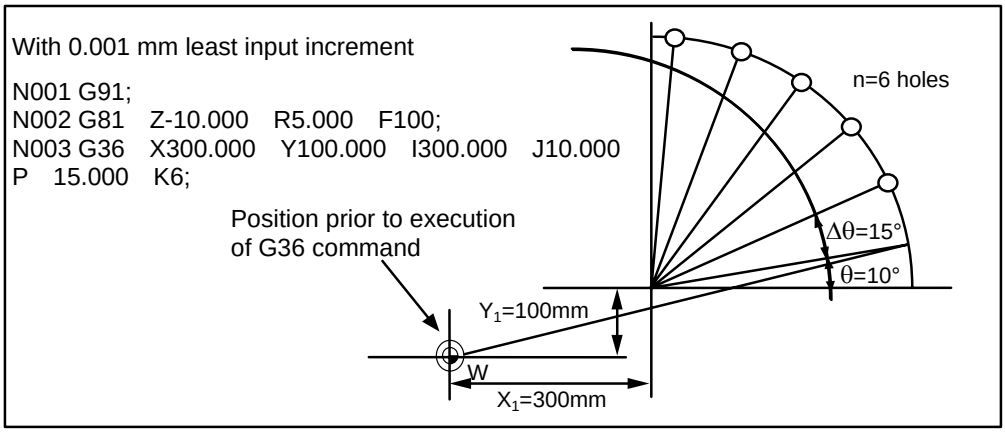
The tool starts at the point forming angle θ with the X axis on the circumference of a circle with radius "r" whose center is the coordinates designated by X and Y, and it drills "n" number of holes aligned at angle interval $\Delta\theta$. As with the bolt hole circle function, the drilling operation 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$\Delta\theta$ Kn ;		
Xx, Yy	:	Center coordinates of arc; they are affected by the G90/G91 commands.
Ir	:	Radius "r" of arc; it is based on the least input increment and is provided with a positive number.
Jθ	:	Angle θ at the point to be drilled initially; the counterclockwise direction is taken to be positive.
P$\Delta\theta$	:	Angle interval $\Delta\theta$; when it is positive, the tool drills in the counterclockwise direction and when it is negative, it drills in the clockwise direction.
Kn	:	Number "n" of holes to be drilled; any number of holes from 1 through 9999 can be assigned.

(Example)

With 0.001 mm least input increment

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

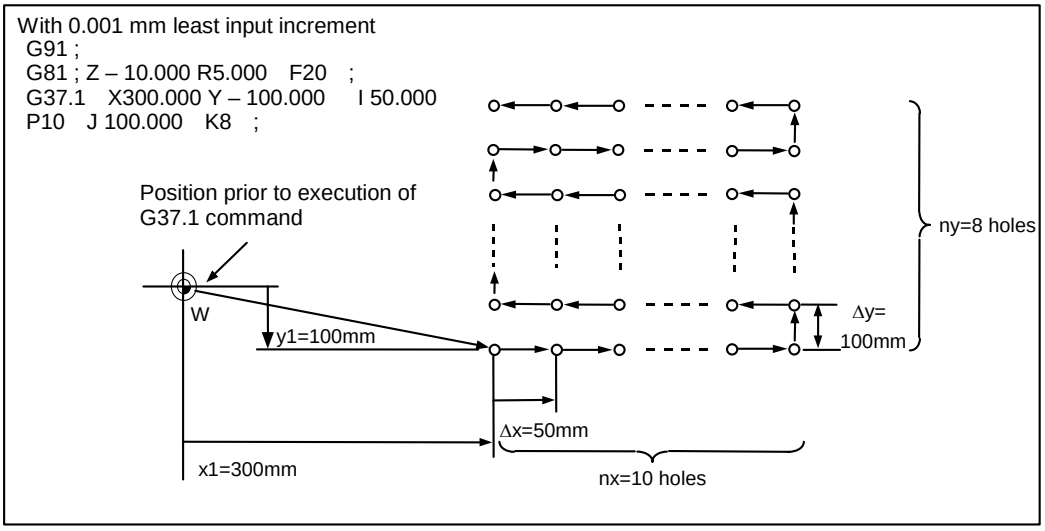


(4) Grid (G37.1)

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

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

(Example)



12.1.3.4 Fixed Cycle for Turning Machining

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:

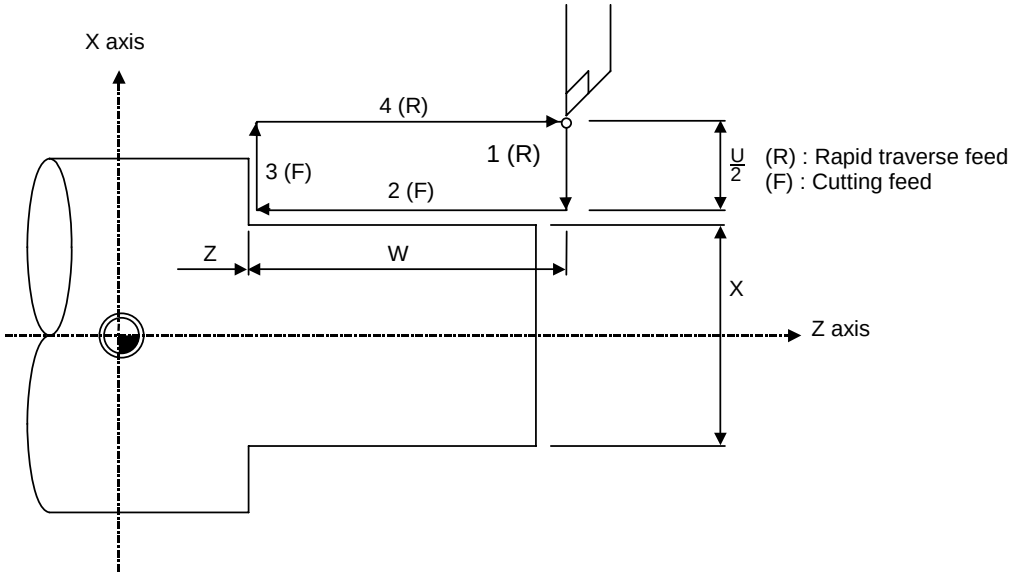
G00, G01, G02, G03
G09
G10, G11
G27, G28, G29, G30
G31
G33, G34
G37
G92
G52, G53
G65

(1) Longitudinal cutting cycle (G77)

(a) Longitudinal cutting

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

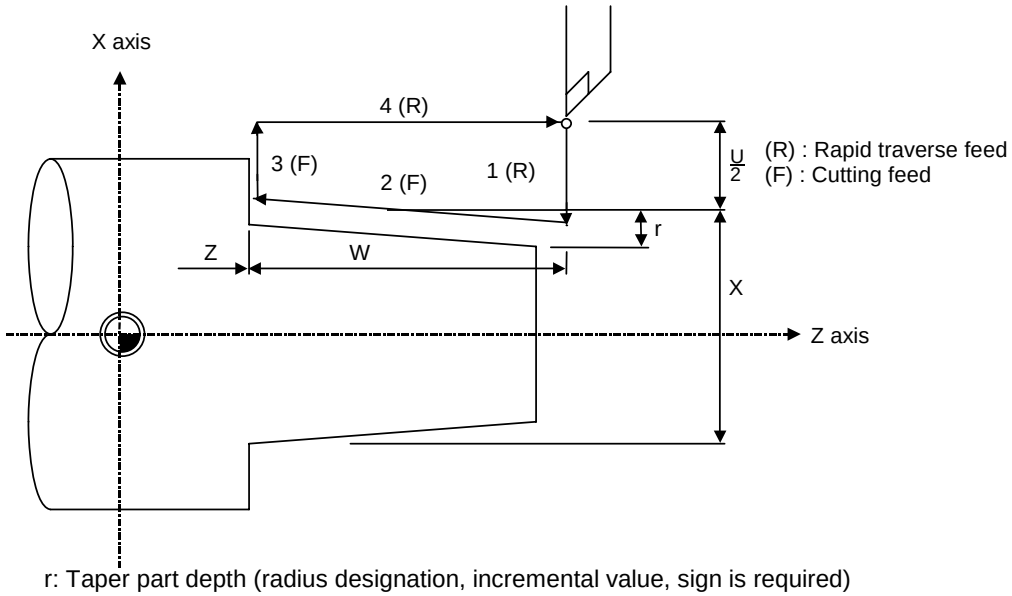
G77 X/U_ Z/W_ F_ ;



(b) Taper cutting

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

G77 X/U_ Z/W_ R_ F_ ;



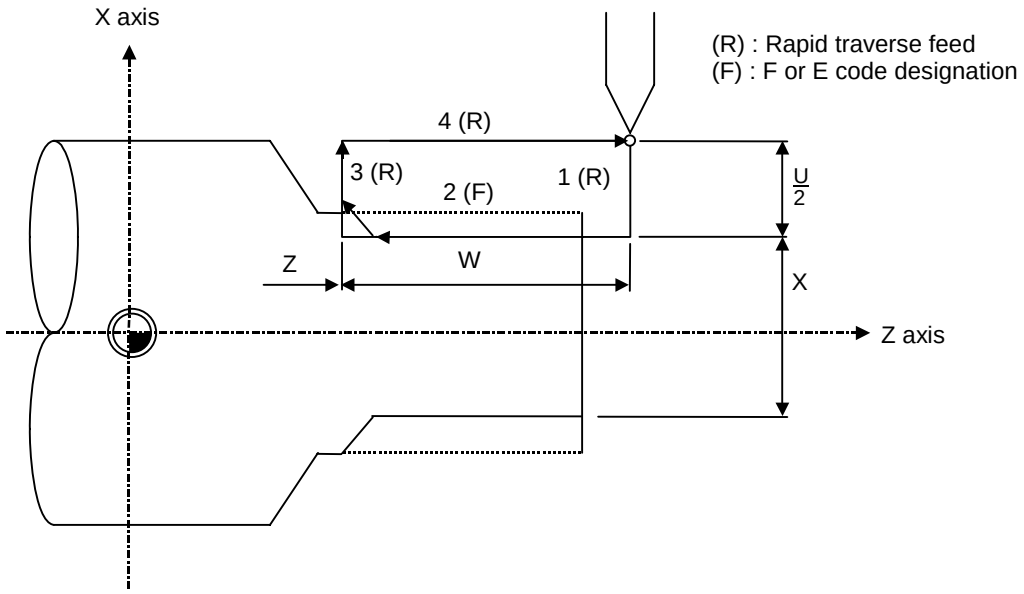
r: Taper part depth (radius designation, incremental value, sign is required)

(2) Thread cutting cycle (G78)

(a) Straight thread cutting

Straight thread cutting can be performed by the following block:

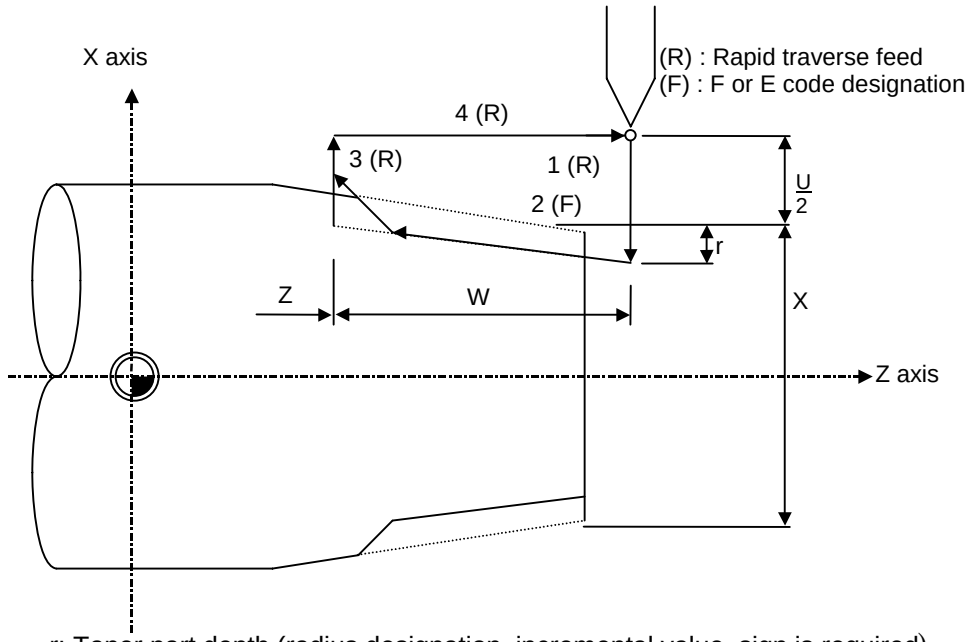
G78 X/U_ Z/W_ F/E_ ;



(b) Taper thread cutting

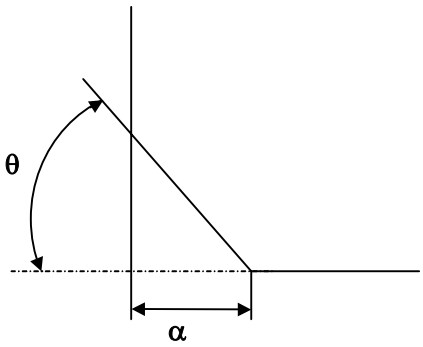
Taper thread cutting can be performed by the following block:

G78 X/U_ Z/W_ R_ F/E_ ;



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

Chamfering



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

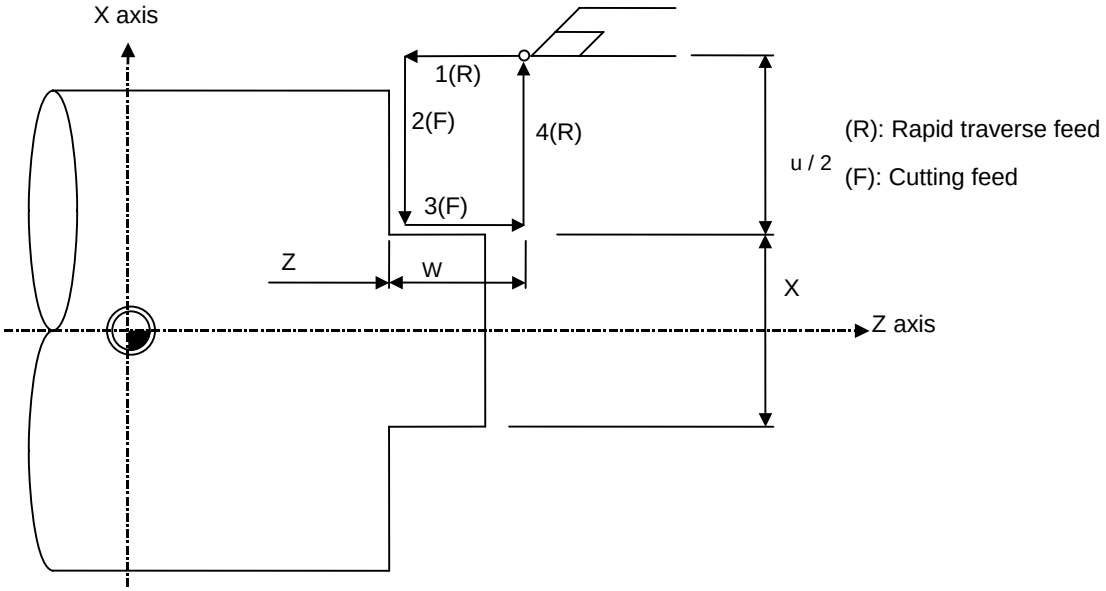
θ : Thread cutting-up angle
The thread cutting-up angle can be set in a given parameter in 1° steps in the range of 0 to 89°.

(3) Face cutting cycle (G79)

(a) Straight cutting

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

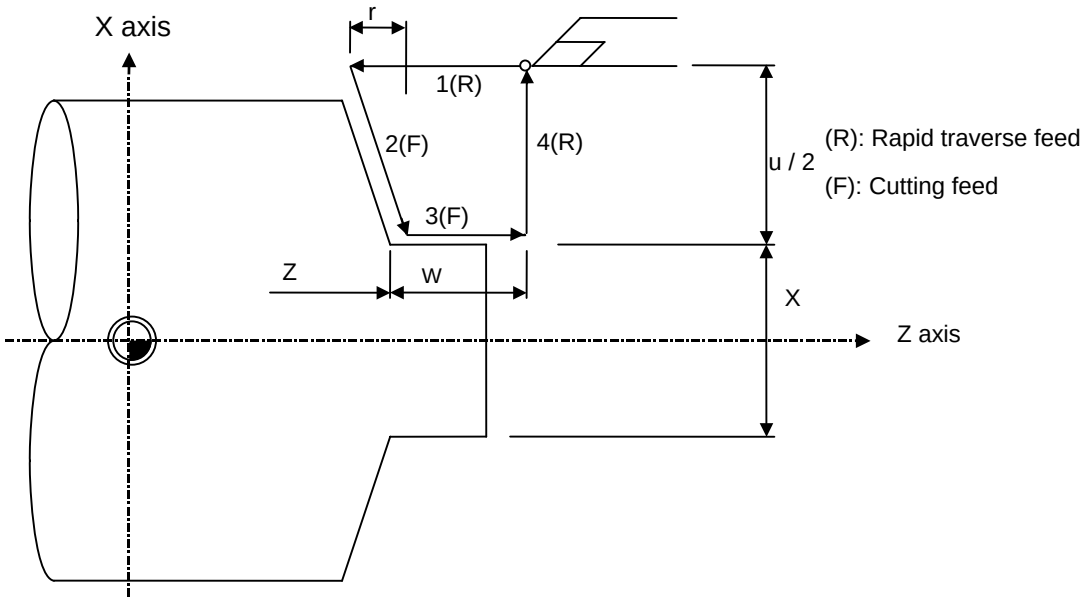
G79 X/U_ Z/W_ F_ ;



(b) Taper cutting

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

G79 X/U_ Z/W_ R_ F_ ;



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

12.1.3.5 Compound Type Fixed Cycle for Turning Machining

M system : - **L system : ○**

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)

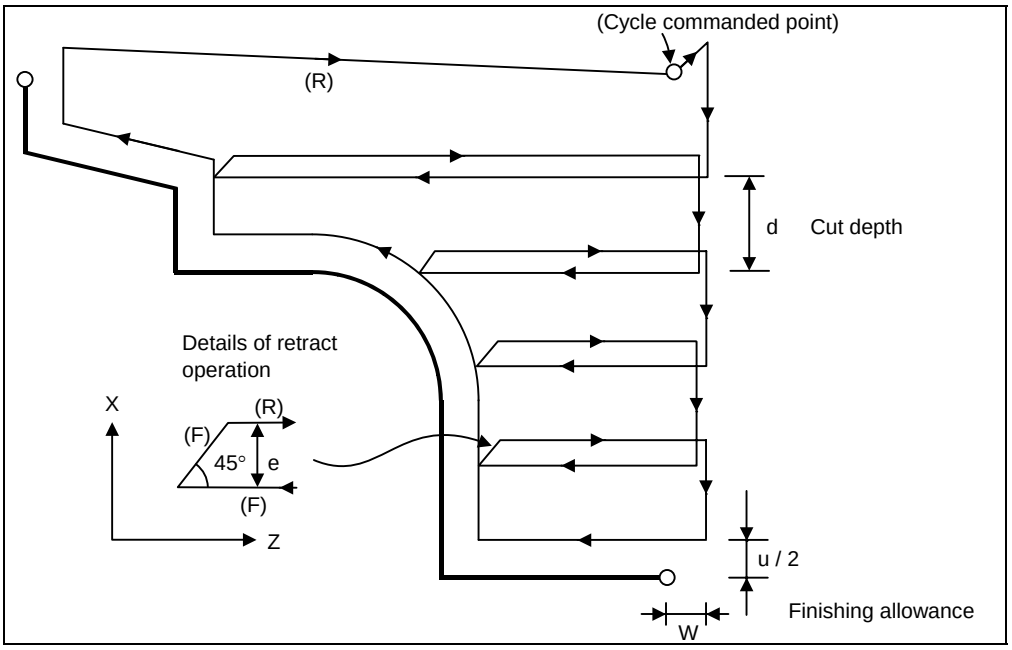
The finish shape program is called, and straight rough cutting is performed while intermediate path is being calculated automatically.

The machining program is commanded as follows.

G71 Ud Re ;
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

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

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



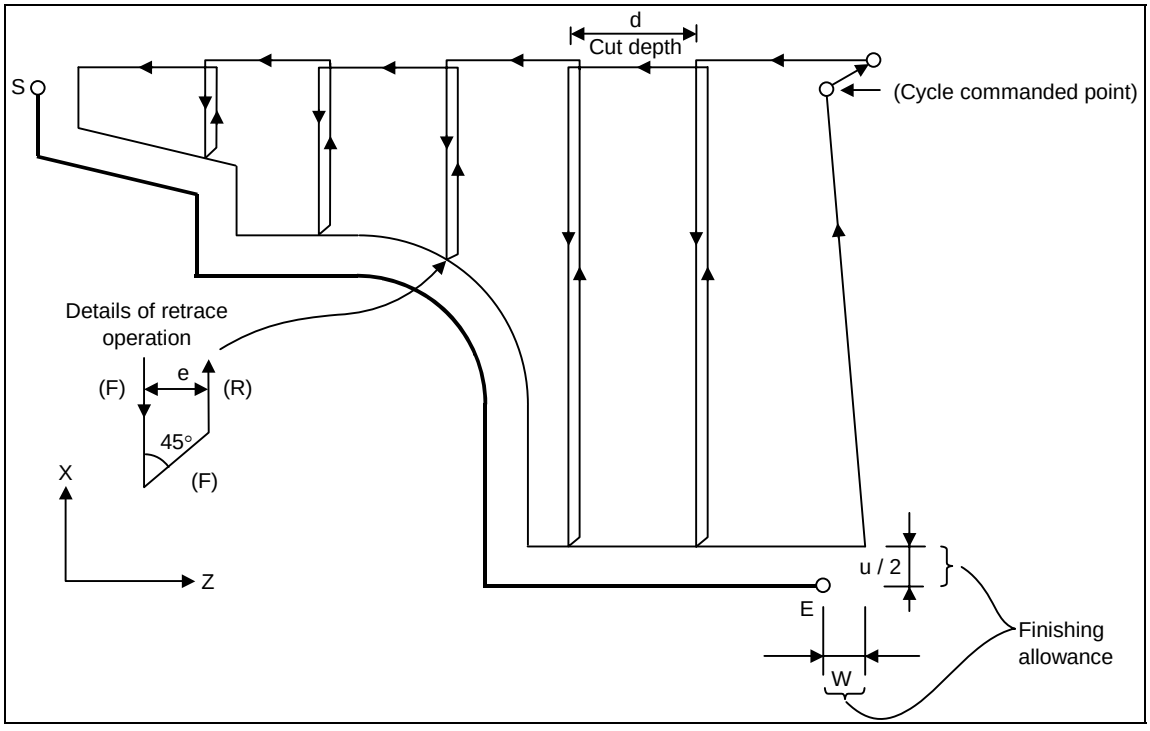
(2) Face rough cutting cycle (G72)

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

The machining program is commanded as follows.

G72 Wd Re ;
G72 Aa Pp Qq Uu Ww Ff Ss Tt ;

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



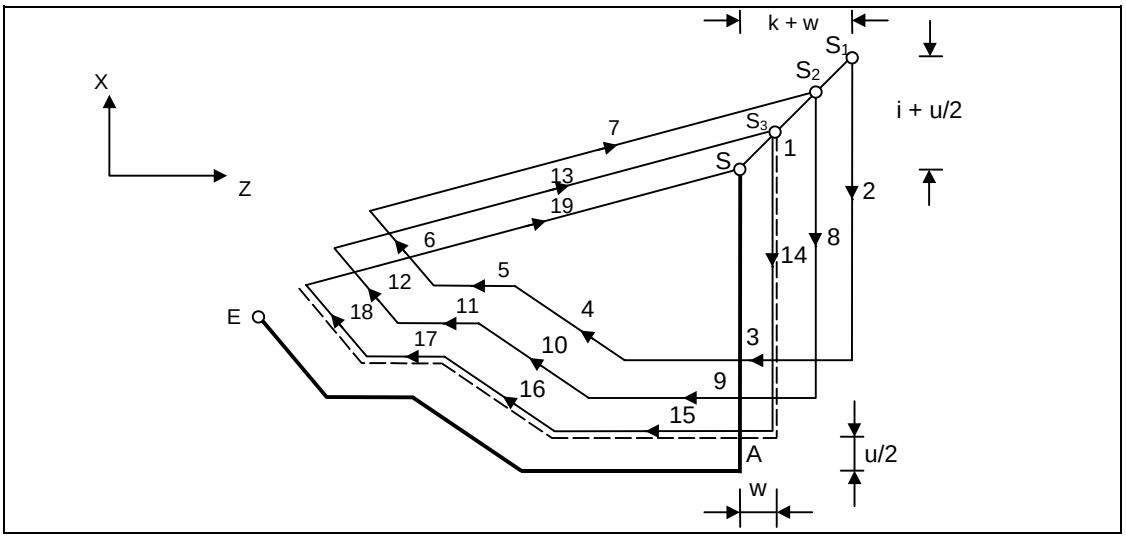
(3) Molding material in rough cutting cycle (G73)

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



(4) Finish cycle (G70)

After rough cutting is performed by using G71 to G73, finish turning can be performed by using the G70 command.

The machining program is commanded as follows.

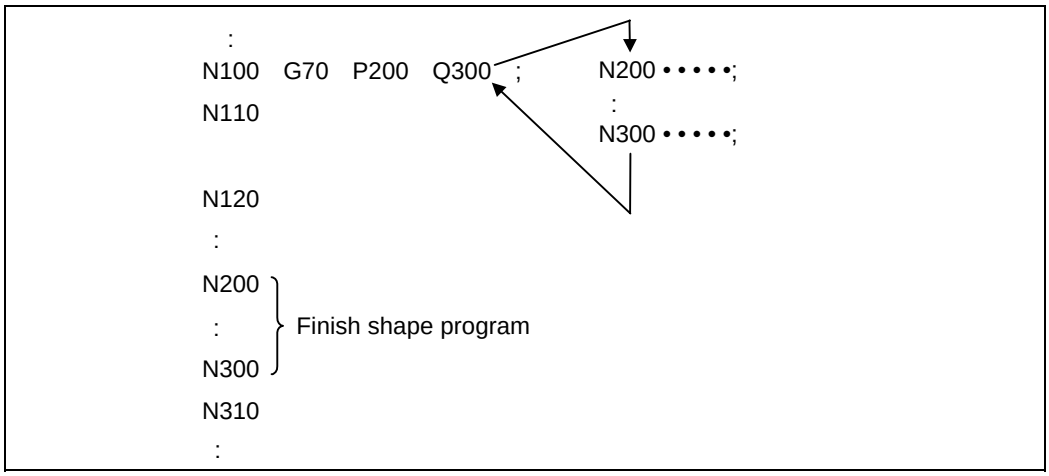
G70 A_ P_ Q_ ;	
A	: Finish shape program number. (If it is omitted, the program being executed is assumed to be designated.)
P	: Finish shape start sequence number. (If it is omitted, the program top is assumed to be designated.)
Q	: Finish shape end sequence number. (If it is omitted, the program end is assumed to be designated.) However, if M99 precedes the Q command, up to M99.

(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.

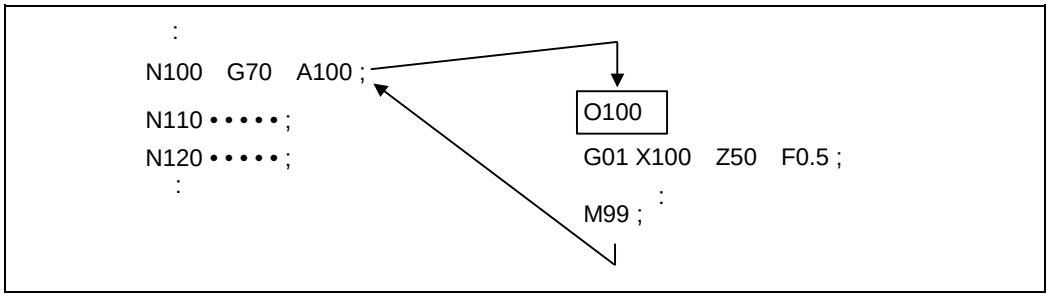
(b) The memory address of the finish shape program executed by G71 to G72 is not stored. Whenever G70 is executed, a program search is made.

(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.

(Example 1) Sequence No. designation



(Example 2) Program No. designation



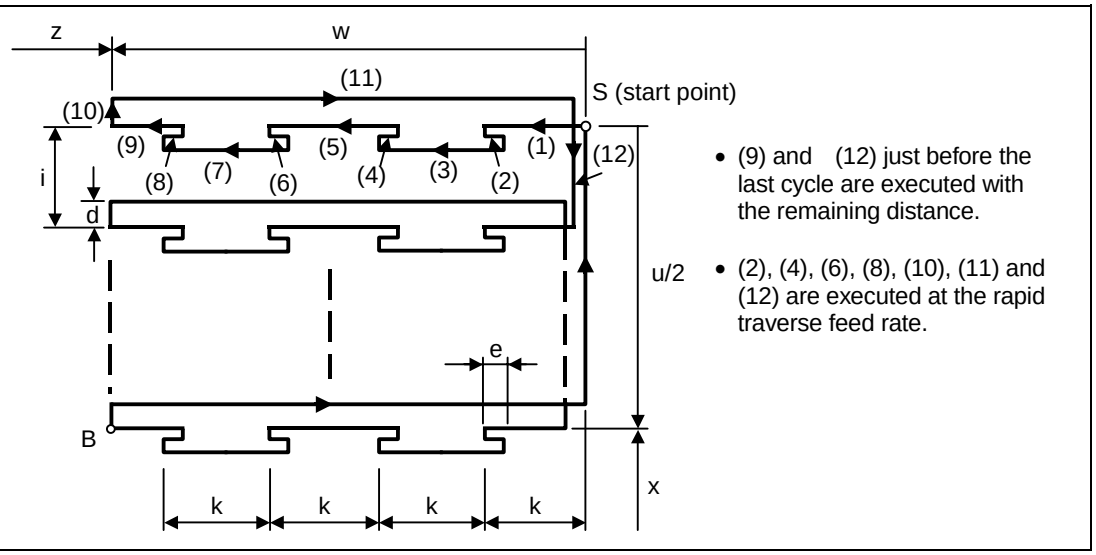
In either example 1 or 2, after the N100 cycle is executed, the N110 block is executed.

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

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

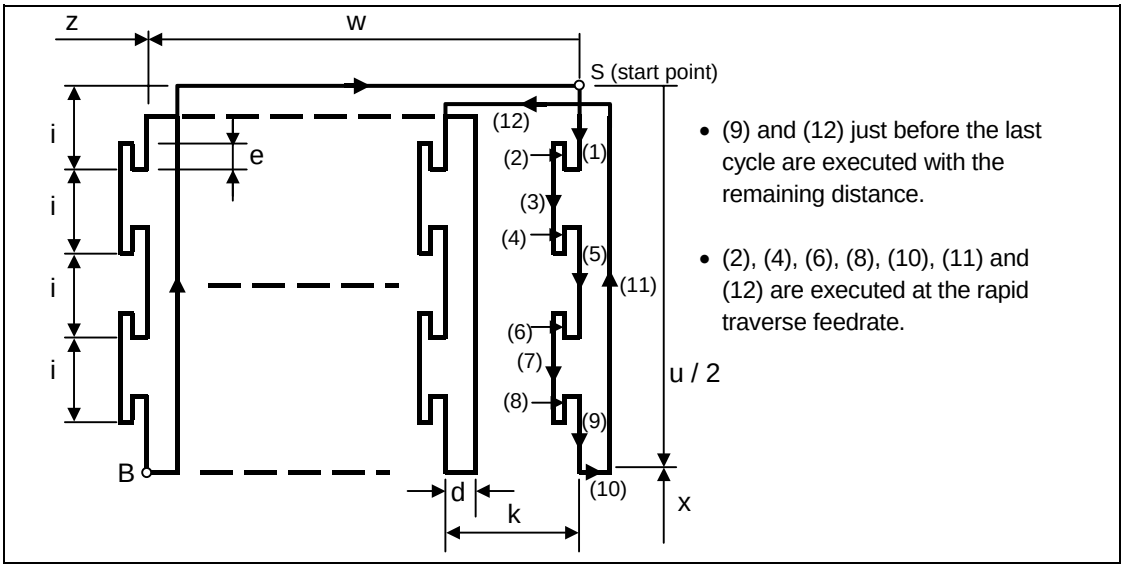


- (9) and (12) just before the last cycle are executed with the remaining distance.
- (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse feed rate.

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



(7) Multiple repetitive thread cutting cycle (G76)

When the thread cutting start and end points are commanded, cut at any desired angle can be made by automatically cutting so that the cut section area (cutting torque) per time becomes constant in the G76 fixed cycle.

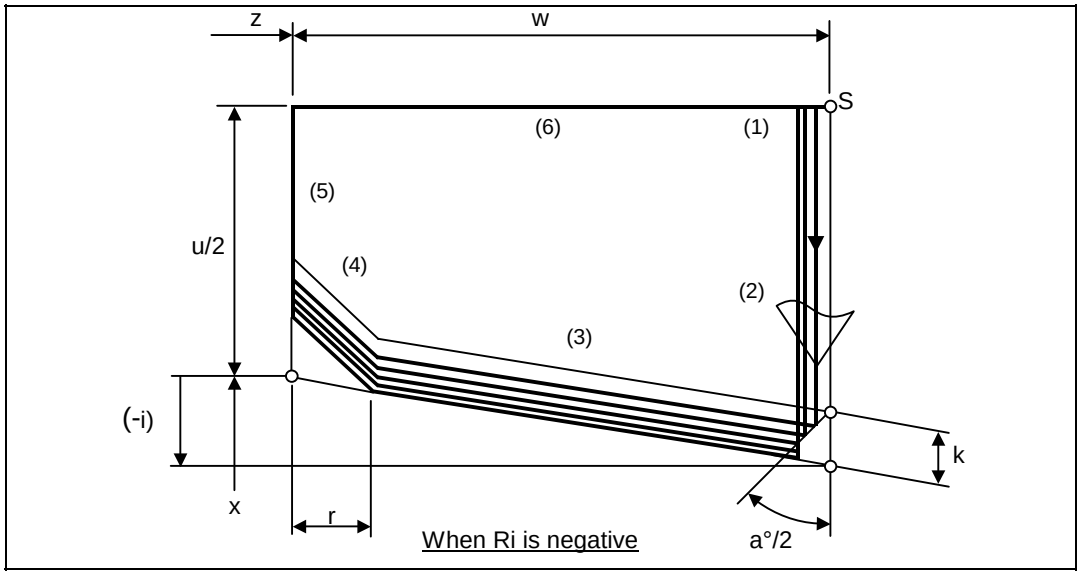
Various longitudinal threads can be cut by considering the thread cutting end point coordinate and taper height constituent command value.

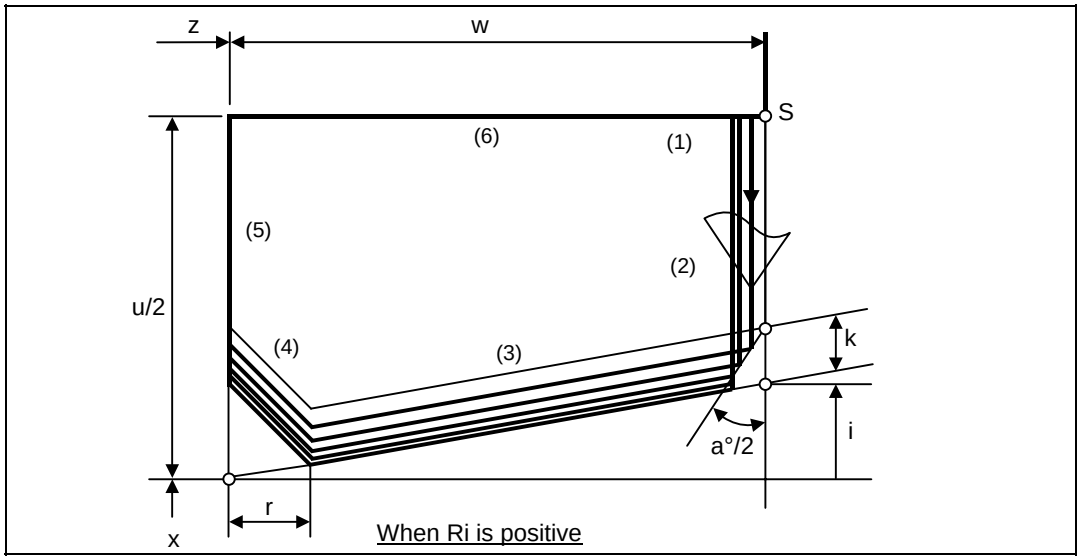
Command Format

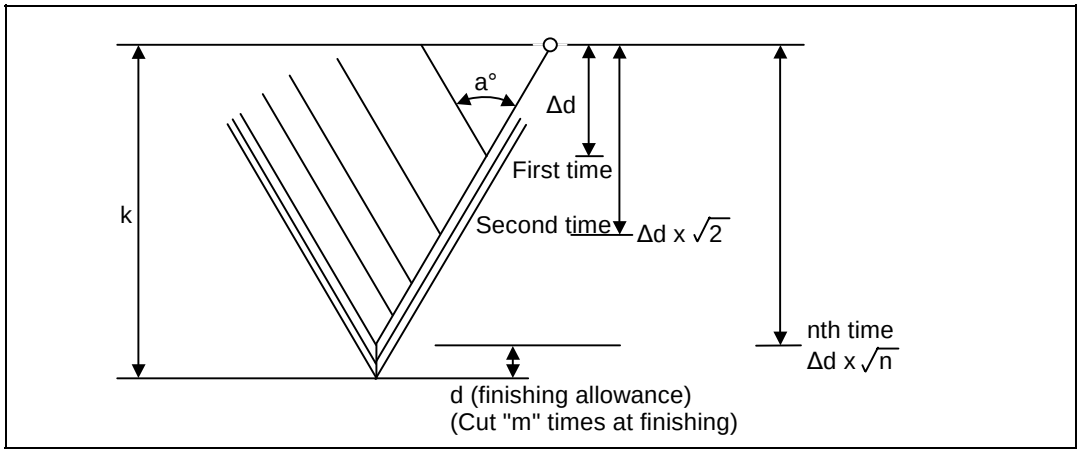
G76	**Pmra**	**Rd ;**
G76	**X/U Z/W Ri Pk QΔd Fl ;**	
m	:	Cut count at finishing 01 to 99 (modal)
r	:	Chamfering amount 00 to 99 (modal). Set in 0.1-lead increments.
a	:	Nose angle (included angle of thread) 00 to 99 (modal) Set in 1-degree increments.
d	:	Finishing allowance (modal)
X/U	:	X axis end point coordinate of thread part. Designate the X coordinate of the end point in the thread part in an absolute or incremental value.
Z/W	:	Z axis end point coordinate of thread part. Designate the Z coordinate of the end point in the thread part in an absolute or incremental value.
i	:	Taper height constituent in thread part (radius value). When i = 0 is set, straight screw is made.
k	:	Thread height. Designate the thread height in a positive radius value.
Δd	:	Cut depth. Designate the first cut depth in a positive radius value.
l	:	Thread lead

Configuration of one cycle

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







12.1.4 Mirror Image

12.1.4.3 Mirror Image by G Code

M system : ○ **L system : -**

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

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

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

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

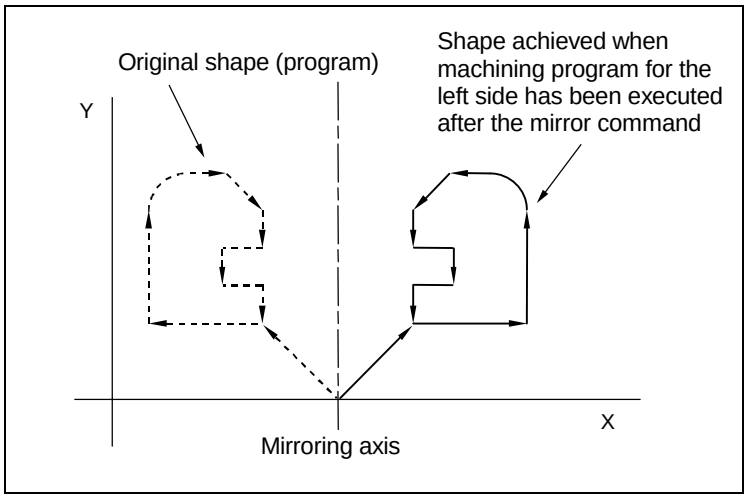
With the local coordinate system, the mirror image is applied with the mirror positioned respectively at x1, y1 and z1.

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

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

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

In the case of G51.1 Xx1



12.1.4.4 Mirror Image for Facing Tool Posts

M system : - **L system : Δ**

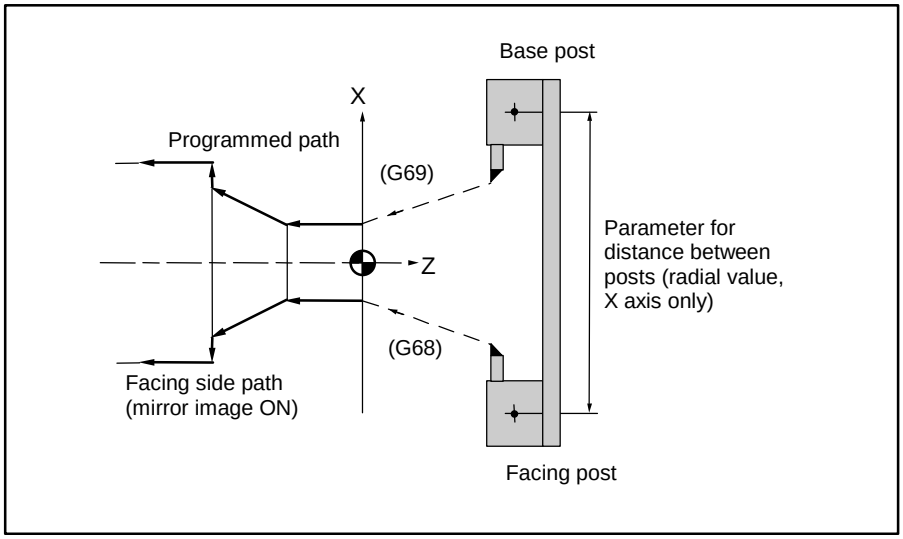
With machines in which the base tool post and facing tool post are integrated, this function enables the programs prepared for cutting at the base side to be executed by the tools on the facing side.
The distance between the two posts is set beforehand with the parameter.

The command format is given below.

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

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

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



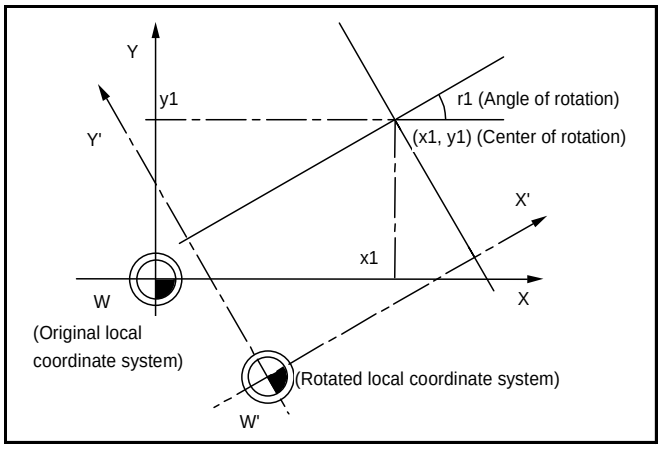
12.1.5 Coordinate System Operation

12.1.5.1 Coordinate Rotation by Program

M system : Δ **L system : -**

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

G68 Xx1 Yy1 Rr1 ;	Coordinate rotation ON
G69 ;	Coordinate rotation cancel
G68	: Coordinate rotation command
Xx1, Yy1	: Rotation center coordinates
Rr1:	: Angle of rotation



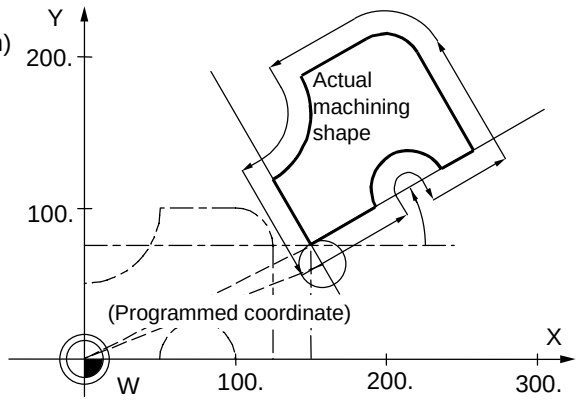
(1) Angle of rotation "r1" can be set in least input increment from –360° to 360°.
(2) The coordinates are rotated counterclockwise by an amount equivalent to the angle which is designated by angle of rotation "r1".
(3) The counter is indicated as the point on the coordinate system prior to rotation.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

(Example)

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

Sub program
(Shape programmed with original coordinate system)

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



12.1.6 Dimension Input

12.1.6.1 Corner Chamfering/Corner R

M system : Δ **L system : Δ**

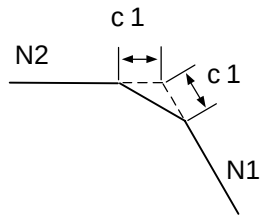
This function executes corner processing by automatically inserting a straight line or arc in the commanded amount between two consecutive movement blocks (G01/G02/G03).
The corner command is executed by assigning the ",C" or ",R" command for the block at whose end point the corner is inserted.

(1) Corner chamfering / Corner R I
When ",C" or ",R" is commanded for linear interpolation, corner chamfering or corner R can be inserted between linear blocks.

- **Corner chamfering**

Example:

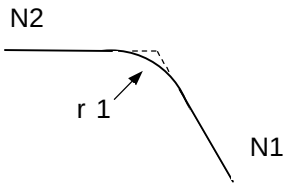
```
N1 G01 Xx1 Zz1,  Cc1  ;
N2 Zz2  ;
```



- **Corner R**

Example:

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



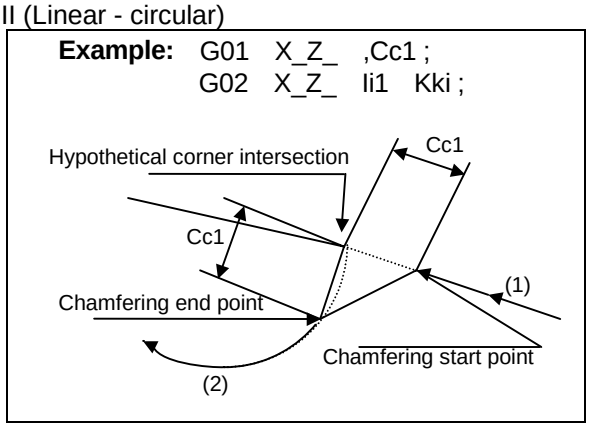
(Note 1) If a corner chamfering or corner R command is issued specifying a length longer than the N1 or N2 block, a program error occurs.

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

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

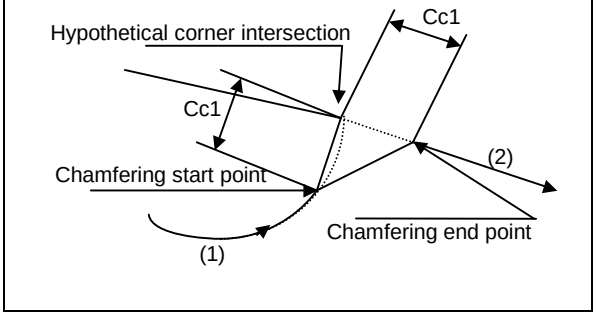
(a) Corner chamfering II (Linear - circular)

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



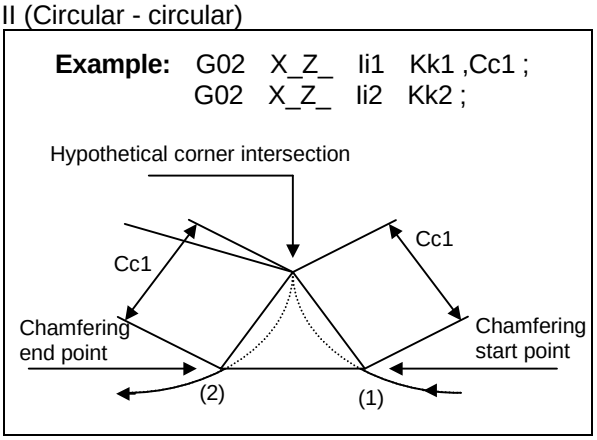
(b) Corner chamfering II (Circular - linear)

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

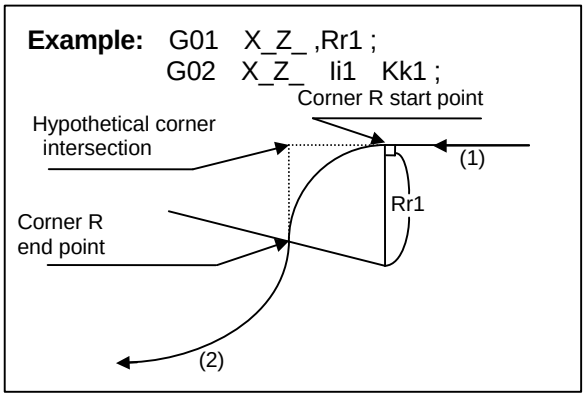


(c) Corner chamfering II (Circular - circular)

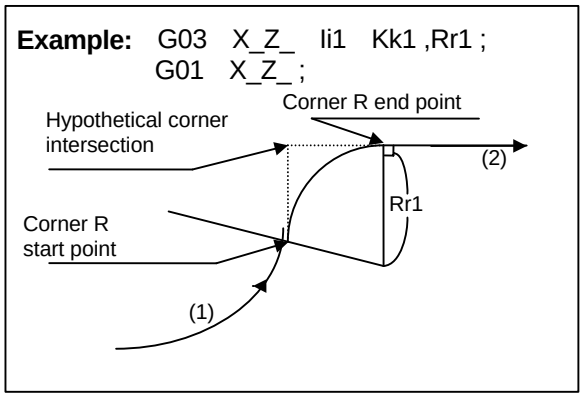
Example: G02 X_Z_ Ii1 Kk1 ,Cc1 ;
G02 X_Z_ Ii2 Kk2 ;



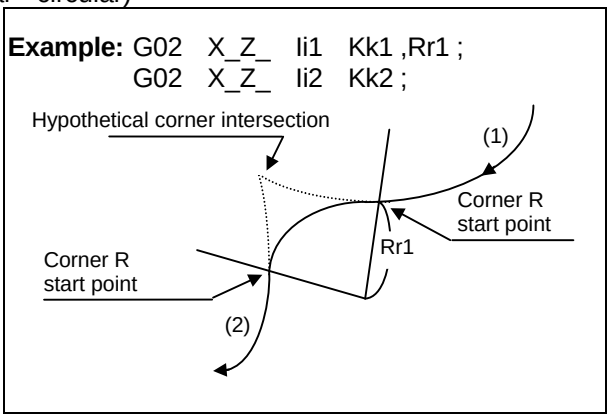
(d) Corner R II (Linear - circular)



(e) Corner R II (Circular - linear)



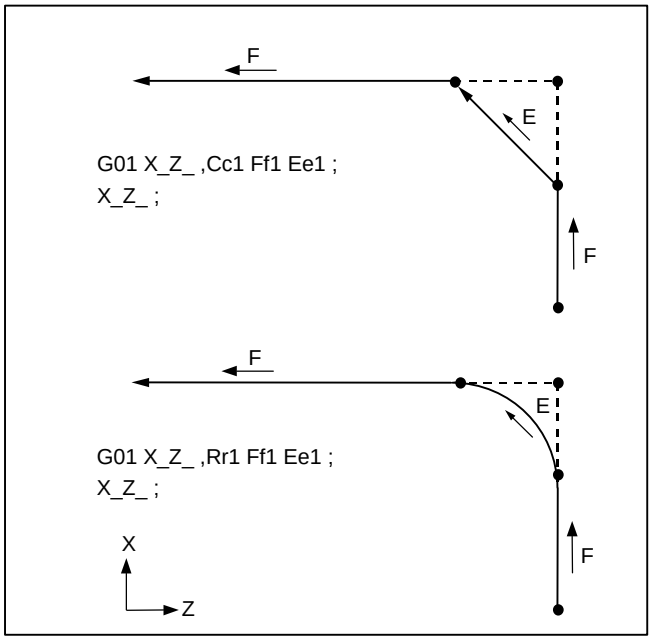
(f) Corner R II (Circular - circular)



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

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

(Example)



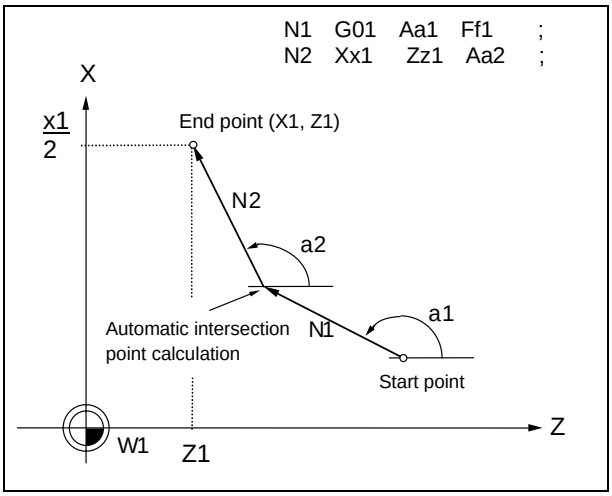
An E command is a modal and remains effective for feeding in next corner chamfering or corner R.
An E command has two separate modals: synchronous and asynchronous feed rate modals. The effective feed rate is determined by synchronous (G95) or asynchronous (G94) mode.
If an E command is specified in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

12.1.6.3 Geometric Command

M system : - **L system : O**

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

Example



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

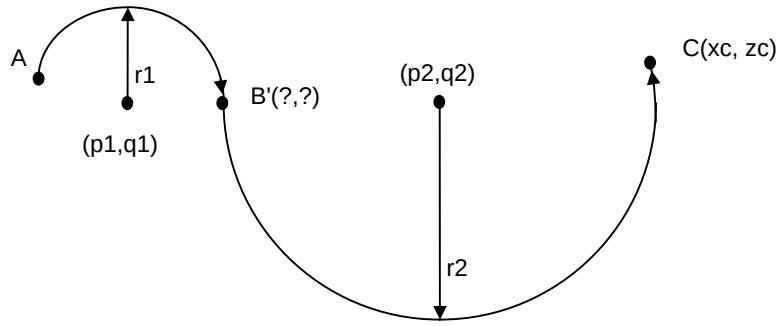
(Note 1) This function cannot be used when using the A axis or 2nd miscellaneous function A.

(1) Automatic calculation of two-arc contact

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

Example

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



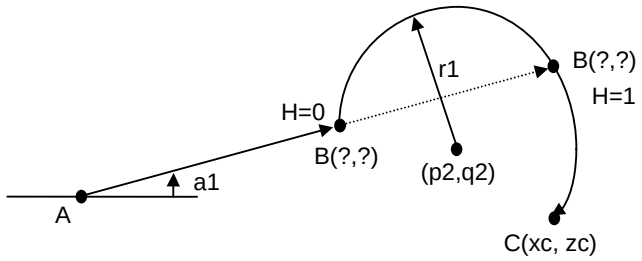
I and K are circular center coordinate incremental values; distances from the start point in the first block or distances from the end point in the second block. P and Q commands (X, Z absolute center coordinates of circular arc) can be given instead of I and K commands.

(2) Automatic calculation of linear-arc intersection

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

Example

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



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

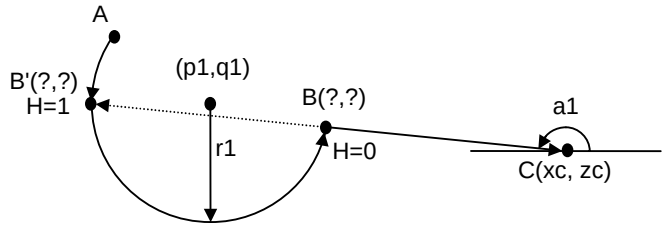
The p2 and q2 can be commanded instead of Ii2 and Kk2.
The linear - arc contact is automatically calculated by designating R instead of I and K (P, Q).

(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   Ff2   ;
```



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

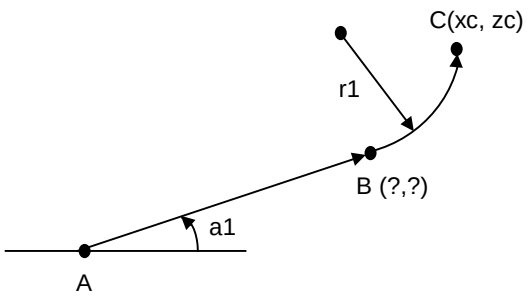
The p2 and q2 can be commanded instead of Ii2 and Kk2.
The arc - linear contact is automatically calculated by designating R instead of I and K (P, Q).

(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   Ff2   ;
```



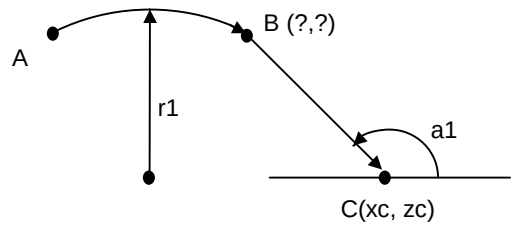
The linear - arc intersection is automatically calculated by designating R instead of P and Q (I, K).

(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  Ff2  ;
```



The arc - linear intersection is automatically calculated by designating R instead of P and Q (I, K).

12.1.7 Axis Control

12.1.7.3 Circular Cutting

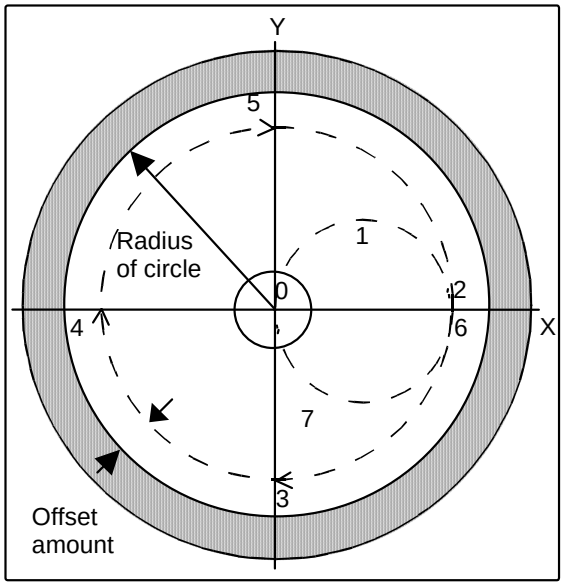
M system : Δ **L system : -**

In circular cutting, a system of cutting steps are performed: first, the tool departs from the center of the circle, and by cutting along the inside circumference of the circle, it draws a complete circle, then it returns to the center of the circle. The position at which G12 or G13 has been programmed serves as the center of the circle.

G code	Function
G12	CW (clockwise)
G13	CCW (counterclockwise)

The program format is given below.

G12/13 Ii Dd Ff ;	
G12/13	: Circular cutting command
Ii	: Radius of complete circle
Dd	: Compensation number
Ff	: Feed rate



When the G12 command is used
(path of tool center)
0 → 1 → 2 → 3 → 4 → 5 → 6 → 7 → 0
When the G13 command is used
(path of tool center)
0 → 7 → 6 → 5 → 4 → 3 → 2 → 1 → 0

(Notes)

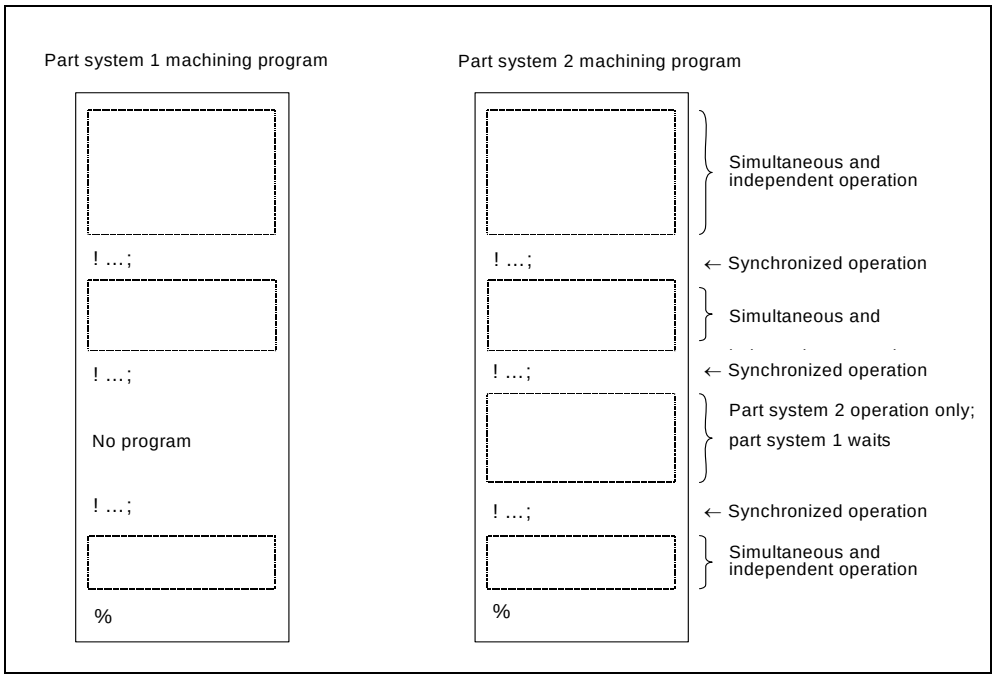
- Circular cutting is undertaken on the plane which has been currently selected (G17, G18 or G19).
- The (+) and (–) signs for the compensation amount denote reduction and expansion respectively.

12.1.8 Multi-part System Control

12.1.8.1 Timing Synchronization Between Part Systems

M system : O **L system : O**

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

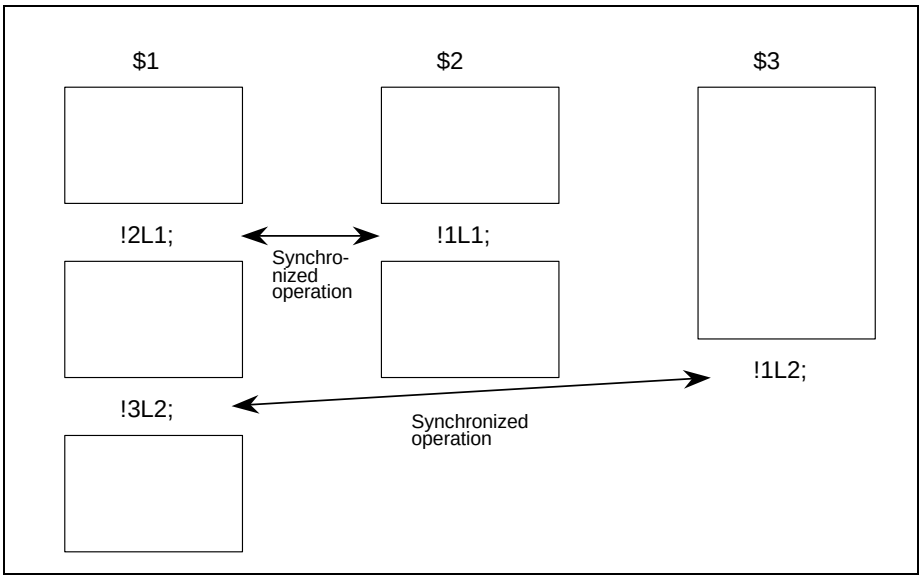


Command format

(1) Command for synchronizing with part system n

!nL1 ;

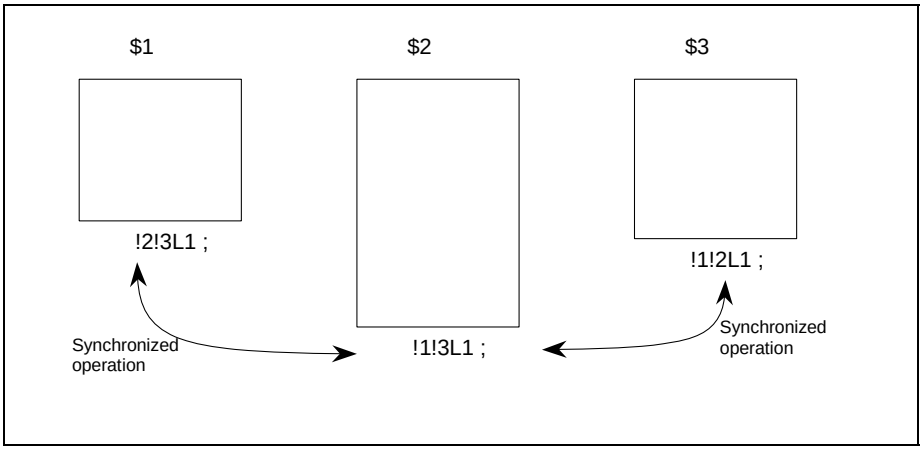
n : Part system number
1 : Synchronizing number 01 to 9999



(2) Command for synchronizing among three part systems

!n!m…L1 ;

n, m : Part system number n ≠ m
1 : Synchronizing number 01 to 9999



12.1.8.2 Start Point Designation Timing Synchronization

M system : ○ **L system : ○**

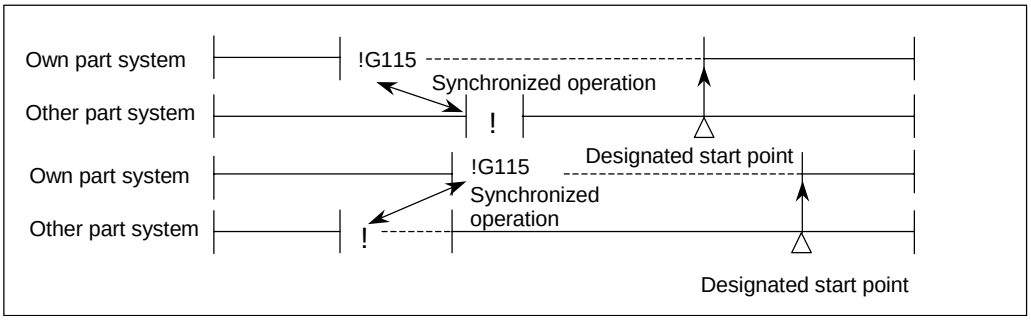
The synchronizing point can be placed in the middle of the block by designating the start point.

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

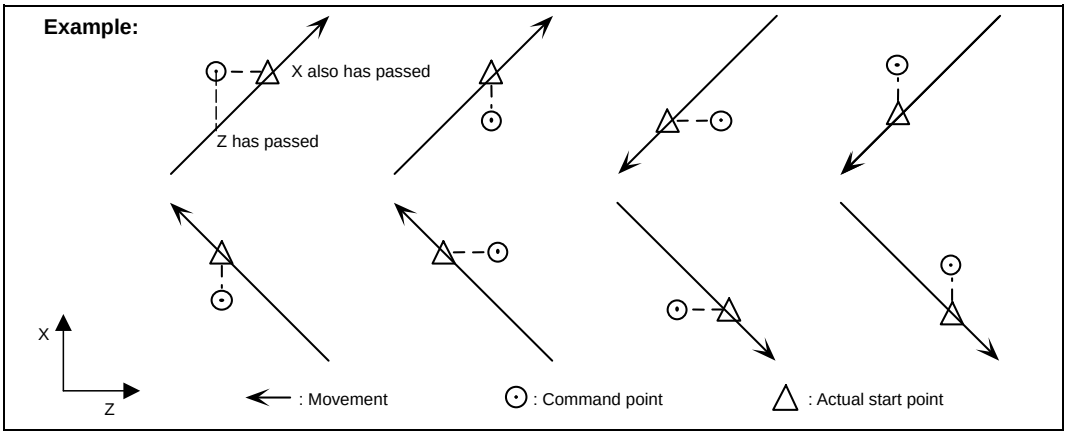
Command format

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

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



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

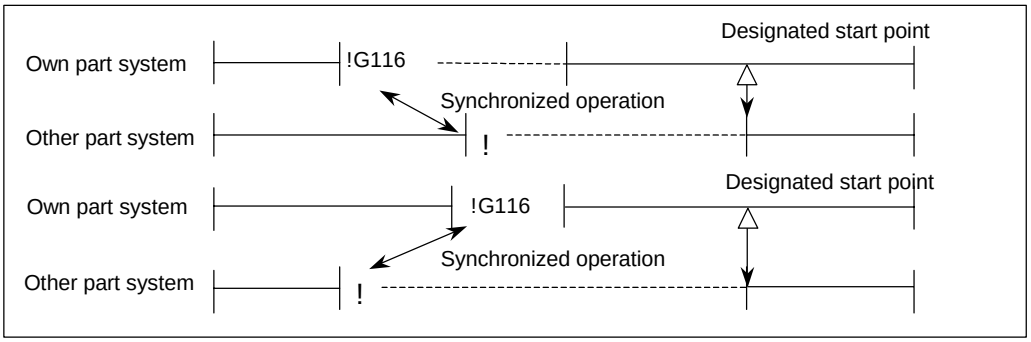


(2) Start point designation synchronization Type 2 (G116)

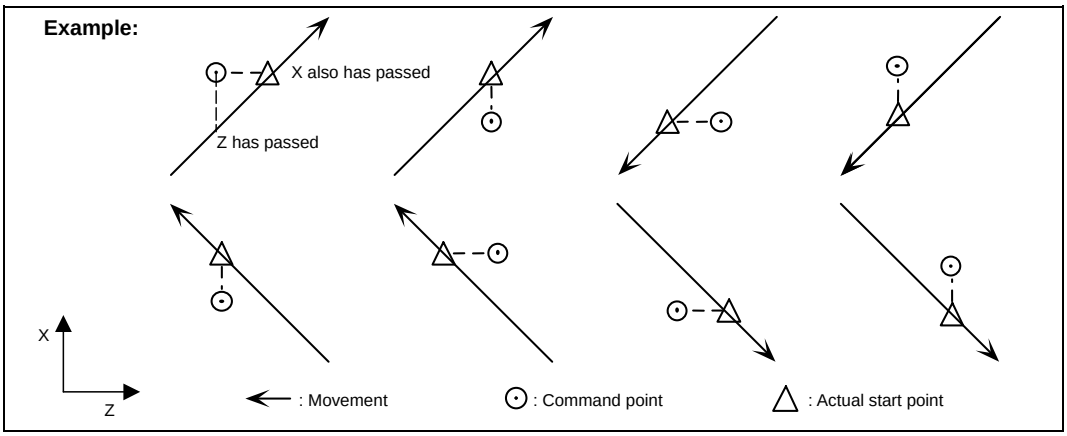
Command format

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

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



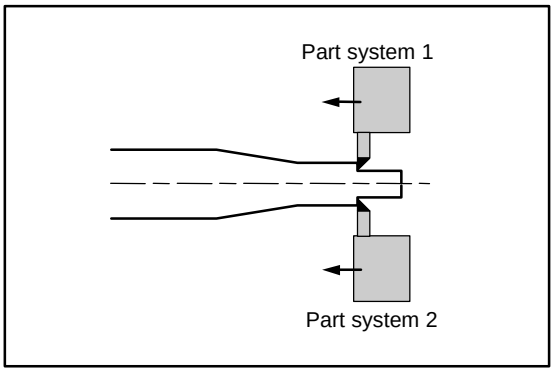
(c) When the start point designated by G116 is not on the next block movement path of the own part system, the other part system starts once the own part system has reached all of the start point axis coordinates.



12.1.8.6 Balance Cut

M system : - **L system : O**

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



The command format is given below.

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

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

Part system 1 program

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

Part system 2 program

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

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

12.1.8.8 2-part System Synchronous Thread Cutting

M system : - **L system : O**

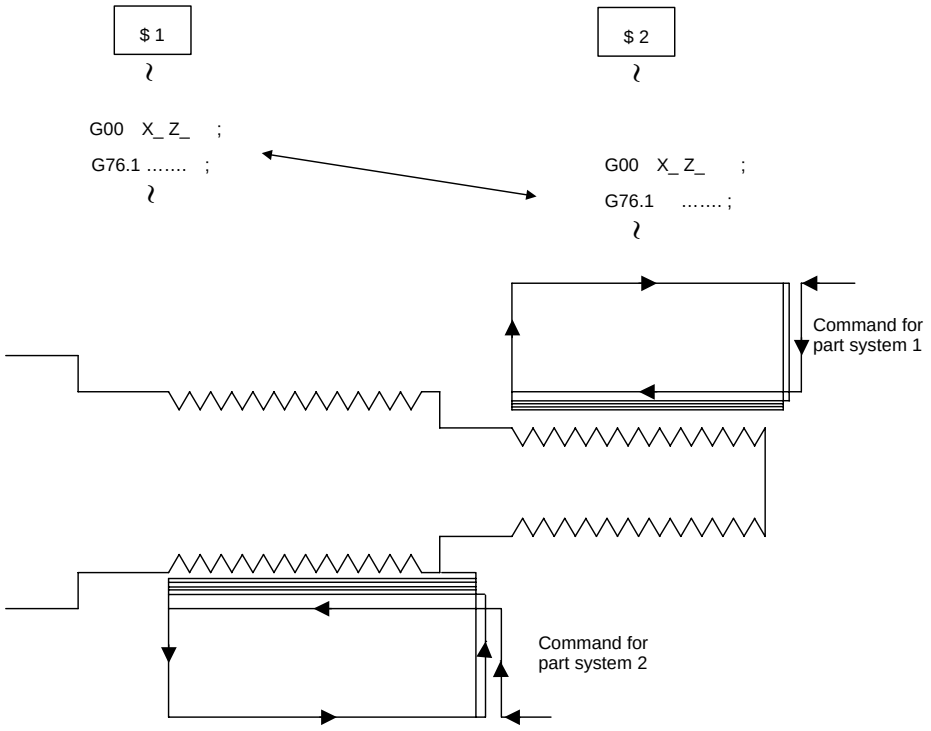
The 2-part system synchronous thread cutting cycle is the function which performs synchronous thread cutting for the same spindle by part systems 1 and 2.
The 2-part system synchronous thread cutting cycle is "2-part system synchronous thread cutting cycle I" (G76.1) for synchronous thread cutting of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

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

Command format

G76. 1 X/U_ Z/W_ Ri Pk Q∆d Fl ;	
X/U	: X axis end point coordinate of screw .... Designate the X coordinate of the end point at screw in an absolute or incremental value.
Z/W	: Z axis end point coordinate of screw .... Designate the Z coordinate of the end point at screw in an absolute or incremental value.
i	: Height constituent of taper at screw (radius value) ... When i is 0, a straight screw is generated.
k	: Screw thread height .... Designate the thread height in a positive radius value.
∆d	: Cut depth .... Designate the first cut depth in a positive radius value.
l	: Thread lead

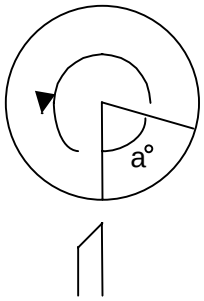
If G76.1 command is given in part system 1 or 2, a wait is made until G76.1 command is given in the other part system.
Once the G76.1 command exists in both part systems, the thread cutting cycle is started.



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

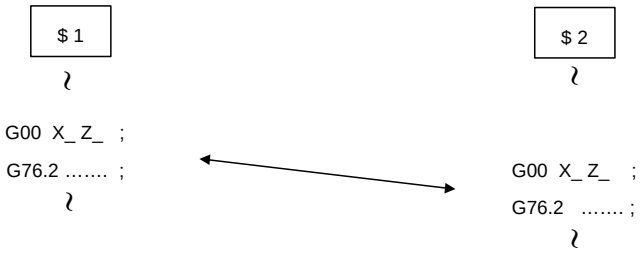
Command format

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

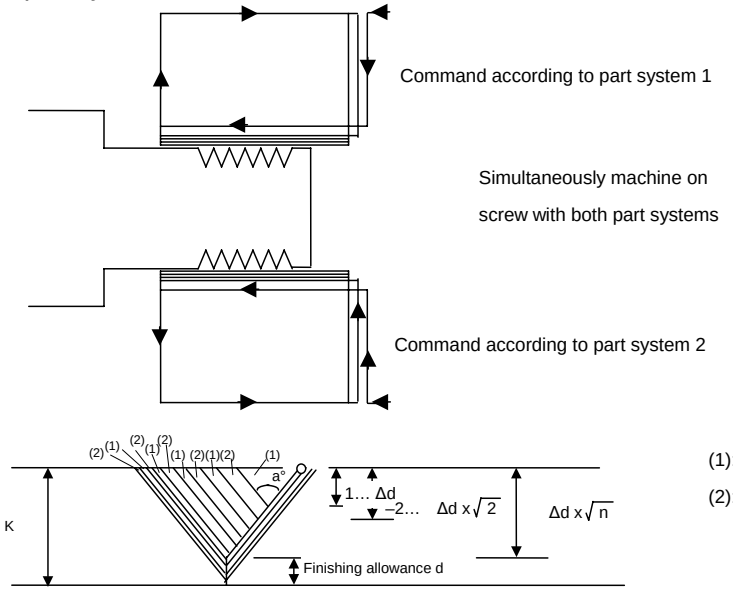


Thread cutting command waits for 1-revolution synchronizing signal of the spindle encoder and starts moving. The start point can be delayed by thread cutting start angle.

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



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



12.1.9 Data Input by Program

12.1.9.1 Parameter Input by Program

M system : Δ **L system : Δ**

The parameters set from the setting and display unit can be changed using the machining programs. The format used for the data setting is shown below.

G10 L50 ; ..... Data setting command

P Major classification No.	(A Axis No.)	N Data No.	H□	Bit type data ;	Parameter settings in data setting mode. There is no A axis No. for axis common data.
P Major classification No.	A Axis No.	N Data No.	D	Byte type data ;	
P Major classification No.	A Axis No.	N Data No.	S	Word type data ;	
P Major classification No.	A Axis No.	N Data No.	L	2-word type data ;	

G11 ; ….. Data setting mode cancel (data setting completed)

The following types of data formats can be used according to the type of parameter (axis-common and axis-independent) and data type.

With axis-common data

Axis-common bit-type parameter	P_____ N_____ H□_____ ;
Axis-common byte-type parameter	P_____ N_____ D _____ ;
Axis-common word-type parameter	P_____ N_____ S _____ ;
Axis-common 2-word-type parameter	P_____ N_____ L _____ ;

With axis-independent data

Axis-independent bit-type parameter	P_____ A _____ N_____ H□_____ ;
Axis-independent byte-type parameter	P_____ A _____ N_____ D _____ ;
Axis-independent word-type parameter	P_____ A _____ N_____ S _____ ;
Axis-independent 2-word-type parameter	P_____ A _____ N_____ L _____ ;

(Note 1) The order of addresses in a block must be as shown above.
(Note 2) For a bit type parameter, the data type will be H□ (□ is a value between 0 and 7).
(Note 3) The axis number is set in the following manner: 1st axis is "1", 2nd axis is "2", and so forth. When using the multi-part system, the 1st axis in each part system is set as "1", the 2nd axis is set as "2", and so forth.
(Note 4) Command G10L50 and G11 in independent blocks. A program error will occur if not commanded in independent blocks.

Depending on the G90/G91 modal status when the G10 command is assigned, the data is used to overwrite the existing data or added.

12.1.9.2 Compensation Data Input by Program

M system : Δ **L system : Δ**

(1) Workpiece coordinate system offset input

The value of the workpiece coordinate systems selected by the G54 to G59 commands can be set or changed by program commands.

G code			Function
G10	L2	P0	External workpiece coordinate system setting
G10	L2	P1	Workpiece coordinate system 1 setting (G54)
G10	L2	P2	Workpiece coordinate system 2 setting (G55)
G10	L2	P3	Workpiece coordinate system 3 setting (G56)
G10	L2	P4	Workpiece coordinate system 4 setting (G57)
G10	L2	P5	Workpiece coordinate system 5 setting (G58)
G10	L2	P6	Workpiece coordinate system 6 setting (G59)

The format for the workpiece coordinate system setting commands is shown below.

G10 L2 Pp1 Xx1 Yy1 Zz1 ;

G10 L2 : Parameter change command
Pp1 : Workpiece coordinate No.
Xx1, Yy1, Zz1 : Settings

(Note 1) L2 can be omitted. Omitting Pp1 results in a program error. [M system]

(2) Tool offset input

The tool offset amounts, which have been set from the setting and display unit, can be input by program commands.

The command format differs between the [M system] and the [L system]. The respective command format must be set by a parameter.

[M system]

Tool offset memory type I

G code	Function
G10 L10	Tool length shape offset amount

Tool offset memory type II

G code	Function
G10 L10	Tool length shape offset amount
G10 L11	Tool length wear offset amount
G10 L12	Tool radius shape offset amount
G10 L13	Tool radius wear offset amount

The tool offset input format is as follows.

G10 L10(L11/L12/L13) Pp1 Rr1 ;	
G10	L10(L11/L12/L13) : Command for setting offset amount
Pp1	: Offset No.
Rr1	: Offset amount

(Note 1) When L11(L12/L13) has been omitted, the tool length shape offset amount is set. Omitting Pp1 results in a program error.

[L system]

G code	Function
G10 L10	Tool length offset amount
G10 L11	Tool wear offset amount

The tool offset input format is as follows.

G10 L10(L11) Pp1 Xx1 Zz1 Rr1 Qq1 ;	
G10	L10(L11) : Command for setting offset amount
Pp1	: Offset No.
Xx1	: X axis offset amount
Zz1	: Z axis offset amount
Rr1	: Nose R compensation amount
Qq1	: Hypothetical tool nose point

12.1.10 Machining Modal

12.1.10.1 Tapping Mode

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) [M system]

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

12.1.10.2 Cutting Mode

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]

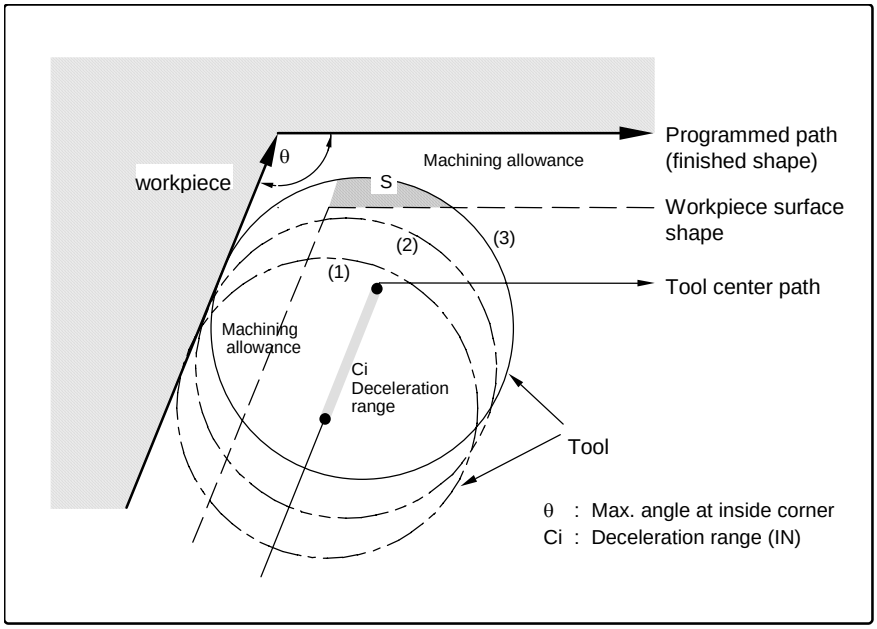
The machine is in the cutting mode status when its power is turned on.

12.2 Machining Accuracy Support Functions

12.2.1 Automatic Corner Override

M system : O **L system : O**

To prevent machining surface distortion due to the increase in the cutting load during cutting of corners, this function automatically applies an override on the cutting feed rate so that the cutting amount is not increased for a set time at the corner.
Automatic corner override is valid only during tool radius compensation.
The automatic corner override mode is set to ON by the G62 command and it is canceled by any of the G commands below.
G40 Tool radius compensation cancel
G61 Exact stop check mode
G63 Tapping mode
G64 Cutting mode
G61.1 High-accuracy control mode [M system]



Operation

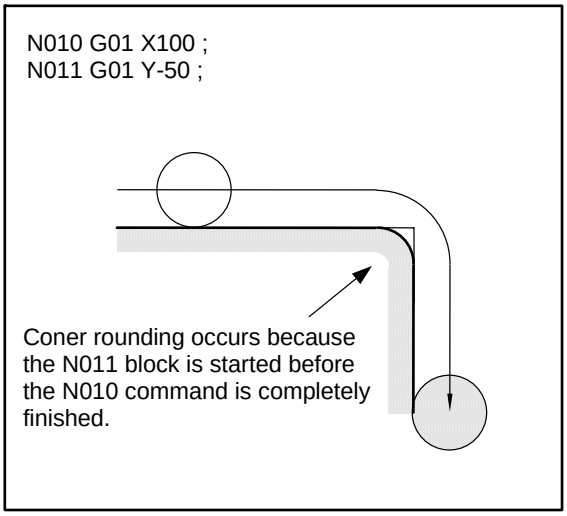
(a) When automatic corner override is not to be applied :
When the tool moves in the order of (1) → (2) → (3) in the figure above, the machining allowance at (3) is larger than that at (2) by an amount equivalent to the area of shaded section S and so the tool load increases.

(b) When automatic corner override is to be applied :
When the inside corner angle θ in the figure above is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range Ci.

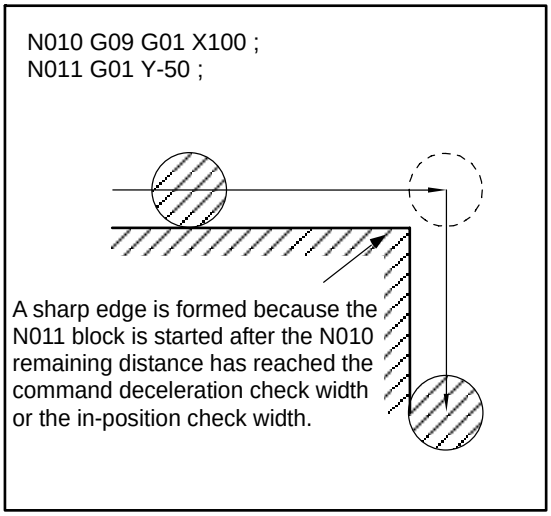
12.2.2 Deceleration Check

The deceleration check function leads the machine to decelerate and stop at the join between one block and another before executing the next block to alleviate the machine shock and to prevent the corner roundness that occurs when the feed rate of the control axis changes suddenly.

Without deceleration check



With deceleration check

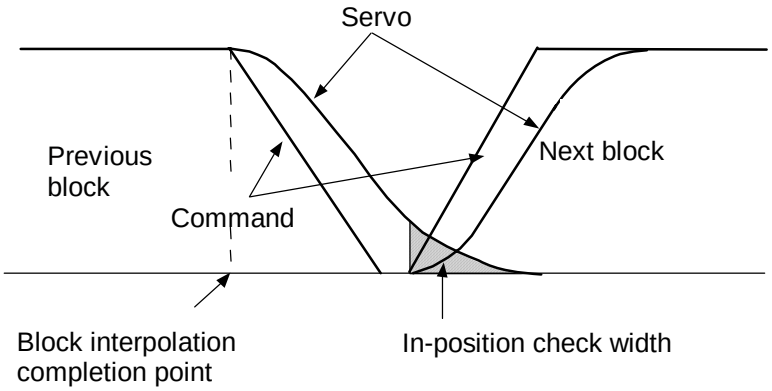


The conditions for executing deceleration check are described below.

(1) Deceleration check in the rapid traverse mode
In the rapid traverse mode, the deceleration check is always performed when block movement is completed before executing the next block.

(2) Deceleration check in the cutting feed mode
In the cutting feed mode, the deceleration check is performed at the end of block when any of the conditions below is applicable before executing the next block.
(a) When G61 (exact stop check mode) is selected.
(b) When the G09 (exact stop check) is issued in the same block.
(c) when the error detect switch (external signal) is ON.

(3) Deceleration check system
Deceleration check is a system that executes the next block only after the command deceleration check is executed as shown below, and it has been confirmed that the position error amount, including the servo system, is less than the in-position check width (designated with parameter or with ",I" in same block).



12.2.2.1 Exact Stop Check Mode

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 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

M system : ○ **L system : ○**

A deceleration check is performed when the G09 (exact stop check) command has been designated in the same block.
The G09 command is issued in the same block as the cutting command. It is an unmodal command.
Refer to "12.2.2 Deceleration Check" for details on the deceleration check.

12.2.2.3 Error Detection

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

M system : O **L system : O**

This command is used to designate the in-position width, which is valid when a linear interpolation command is assigned, from the machining program. The in-position width designated with a linear interpolation command is valid only in cases when the deceleration check is performed, such as:

- When the error detect switch is ON.
- When the G09 (exact stop check) command has been designated in the same block.
- When the G61 (exact stop check mode) command has been selected.

G01 X_ Z_ F_ ,I_;	
X_,Z_	: Linear interpolation coordinates of axes
F_	: Feed rate
,I_	: In-position width

This command is used to designate the in-position width, which is valid when a positioning command is assigned, from the machining program.

G00 X_ Z_ ,I_;	
X_,Z_	: Positioning coordinates of axes
,I_	: In-position width

In-position check operation

After it has been verified that the position error between the block in which the positioning command (G00: rapid traverse) is designated and the block in which the deceleration check is performed by the linear interpolation command (G01) is less than the in-position width of this command, the execution of the next block is commenced.

12.3 High-speed and High-accuracy Functions

12.3.5 High-Accuracy Control 1

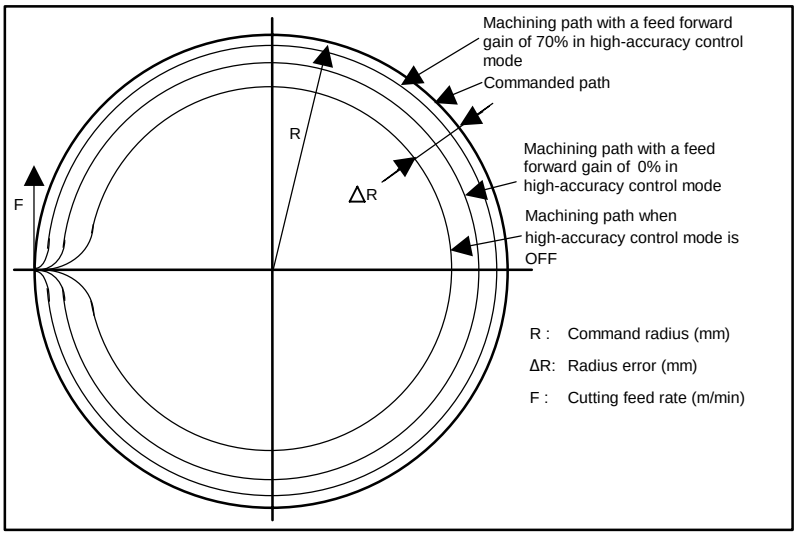
M system : Δ **L system : -**

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

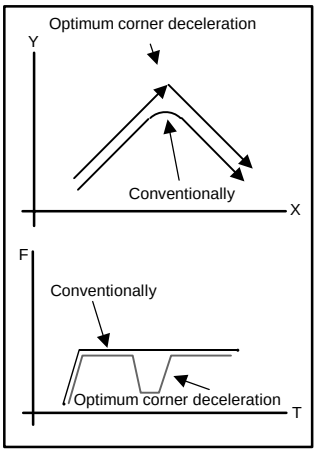
The high-accuracy control is commanded with ;

G61.1	High-accuracy control ON

Effects in G02/G03 circular interpolation



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



(1) Acceleration / deceleration before interpolation

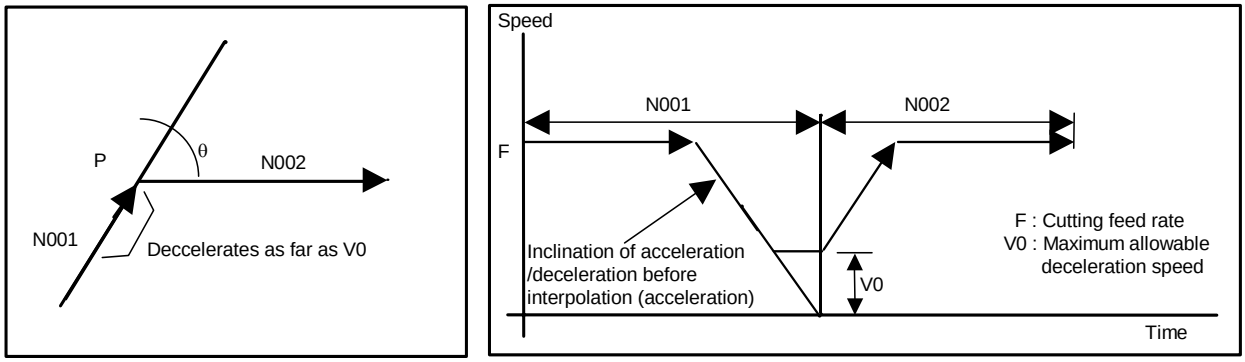
By accelerating /decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved.
With the arc commands, the radius reduction error can be significantly minimized.
Furthermore, since constant inclination acceleration/deceleration is performed, the time taken for positioning at microscopically small distances in the G00 command is reduced.

(Note 1) Whether acceleration/deceleration before interpolation in the rapid traverse command (G00) is to be performed always or not can be selected using a parameter setting independently from the high-accuracy control assignment.

(2) Optimum corner deceleration

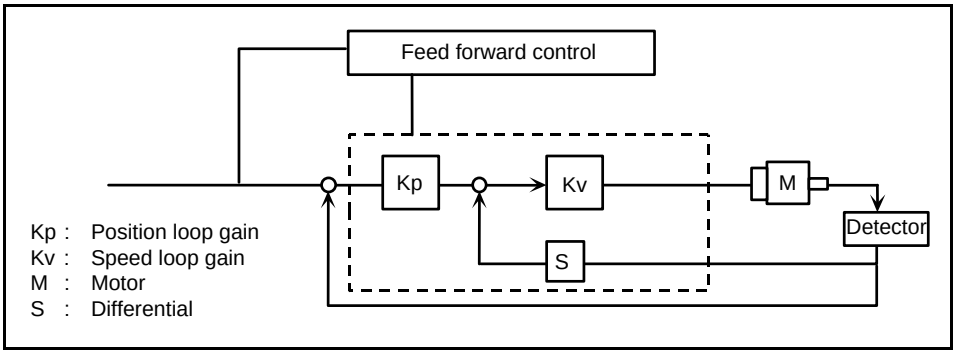
By determining the command vector in the machining program and thereby performing corner deceleration, it is possible to machine workpiece with a high-edge accuracy. The figure below shows the pattern of the deceleration speed at the corners.
(Optimum corner deceleration is a function of high-accuracy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

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



(3) Feed forward control

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



13. Machine Accuracy Compensation

13.1 Static Accuracy Compensation

13.1.1 Backlash Compensation

M system : O **L system : O**

This function compensates for the error (backlash) produced when the direction of the machine system is reversed.
The backlash compensation can be set in the cutting feed mode or rapid traverse mode.
The amount of backlash compensation can be set separately for each axis. It is set using a number of pulses in increments of one-half of the least input unit. The output follows the output unit system. The "output unit system" is the unit system of the machine system (ball screw unit system).
The amount of compensation for each axis ranges from 0 to ±9999 (pulses).

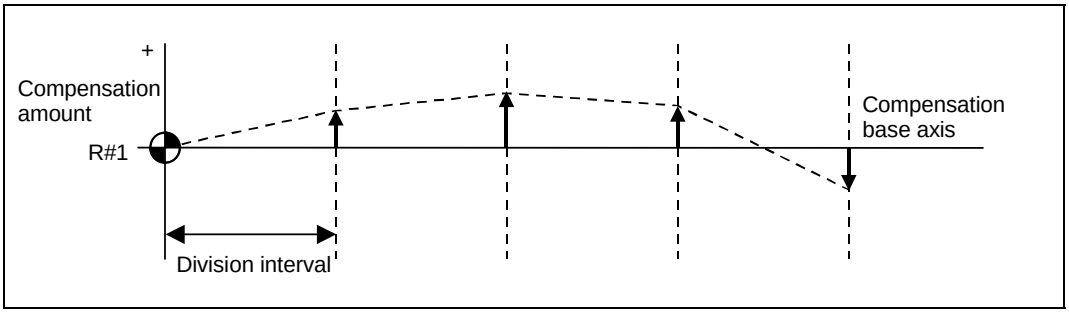
13.1.2 Memory-type Pitch Error Compensation

M system : Δ **L system : Δ**

The machine accuracy can be improved by compensating for the errors in the screw pitch intervals among the mechanical errors (production errors, wear, etc.) of the feed screws.
The compensation positions and amounts are stored in the memory by setting them beforehand for each axis, and this means that there is no need to attach dogs to the machine.
The compensation points are divided into the desired equal intervals.

1. Division intervals of compensation points : 1 to 9999999 (μm)
2. Number of compensation points : 1024
3. Compensation amount : -128 to 127 (output unit)
4. No. of compensated axes : 10 axes (including number of axes for relative position error compensation)

(1) The compensation position is set for the compensation axis whose reference point serves as the zero (0) point. Thus, memory-type pitch error compensation is not performed if return to reference point is not made for the compensation base axis or compensation execution axis after the controller power is turned ON and the servo is turned ON.
(2) When the compensation base axis is a rotary axis, select the dividing intervals so that one rotation can be divided.



(3) As shown in the figure above, highly individualized compensation control is exercised using the minimum output units with linear approximation for the compensation intervals between the compensation points.

(Note 1) Compensation points 1,024 is a total including the points for memory-type relative position error compensation.
(Note 2) A scale of 0 to 99-fold is applied on the compensation amount.

13.1.3 Memory-type Relative Position Error Compensation

M system : Δ **L system : Δ**

Machine accuracy can be improved by compensating a relative error between machine axes, such as a production error or time aging.
The compensation base axis and compensation execution axis are set by using parameters.
The compensation points are divided at any desired equal intervals.

1. Compensation point dividing intervals	: 1 to 9999999 (μm)
2. Number of compensation points	: 1024
3. Compensation amount	: -128 to 127 (output unit)
4. No. of compensated axe	: 10 axes (including number of axes for memory type pitch error compensation.)

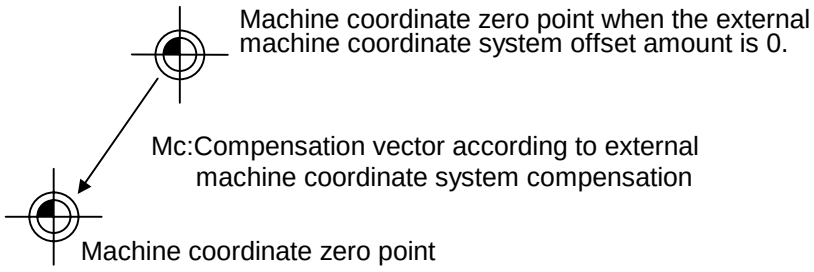
(1) The compensation position is set for the compensation axis whose reference point serves as the zero (0) point. Thus, memory-type relative position error compensation is not performed if return to reference point is not made for the compensation base axis or compensation execution axis after the controller power is turned ON and the servo is turned ON.
(2) When the compensation base axis is a rotary axis, select the dividing intervals so that one rotation can be divided.
(3) Since all coordinate systems of compensation execution axes are shifted or displaced by the compensation amount when the relative position error compensation is made, the stroke check point and machine coordinate system are also shifted or displaced.

(Note 1) Compensation points 1024 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

M system : Δ **L system : Δ**

The coordinate system can be shifted by inputting a compensation amount from the PLC. This compensation amount will not appear on the counter (all counters including machine position). If the machine's displacement value caused by heat is input for example, this can be used for thermal displacement compensation.



13.1.5 Circular Error Radius Compensation

M system : Δ **L system : Δ**

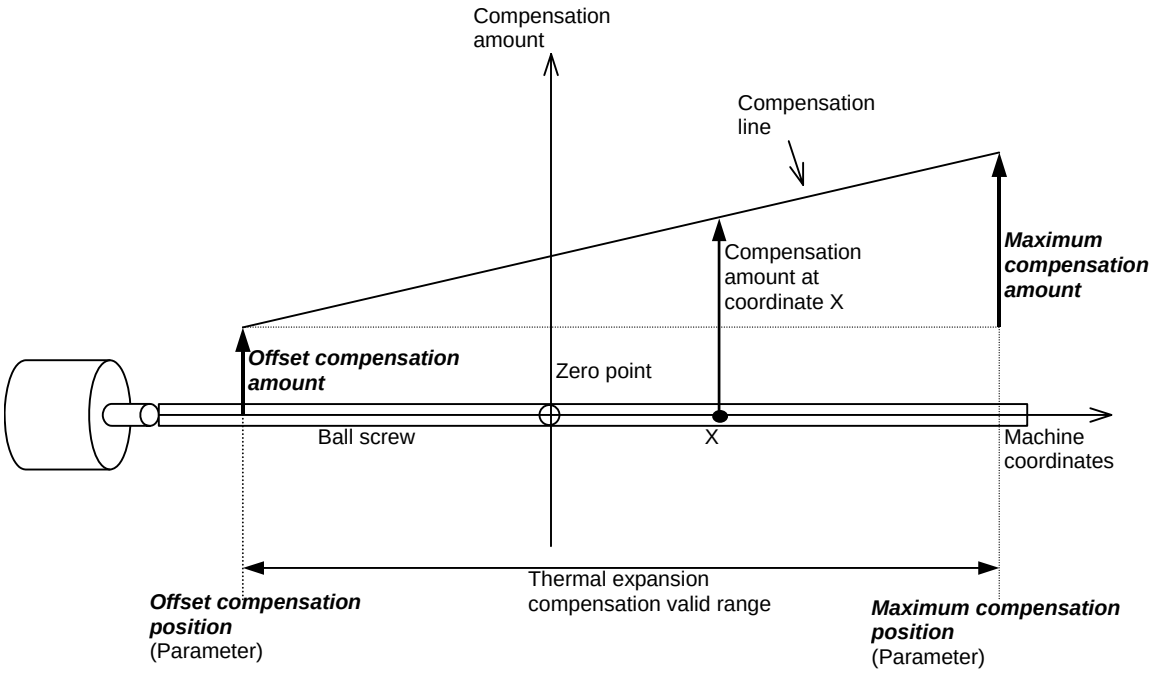
With commands designated during arc cutting, this function compensates for movement toward the inside of the arcs caused by a factor such as servo delay.

13.1.6 Ball Screw Thermal Expansion Compensation

M system : Δ **L system : Δ**

(1) Outline

The feed error caused by the thermal expansion of the ball screw is set from the PLC, and compensated.
The compensation amount depends on the offset compensation amount and maximum compensation amount.
The compensation amount based on the offset compensation amount is set as the maximum compensation amount.
The offset compensation amount and maximum compensation amount are set beforehand in the parameters.



(2) Compensation operation

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

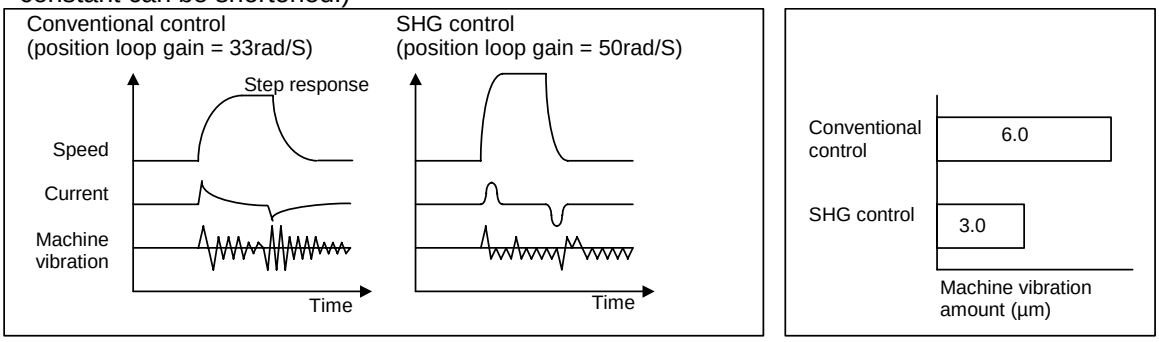
13.2 Dynamic Accuracy Compensation

13.2.1 Smooth High-gain (SHG) Control

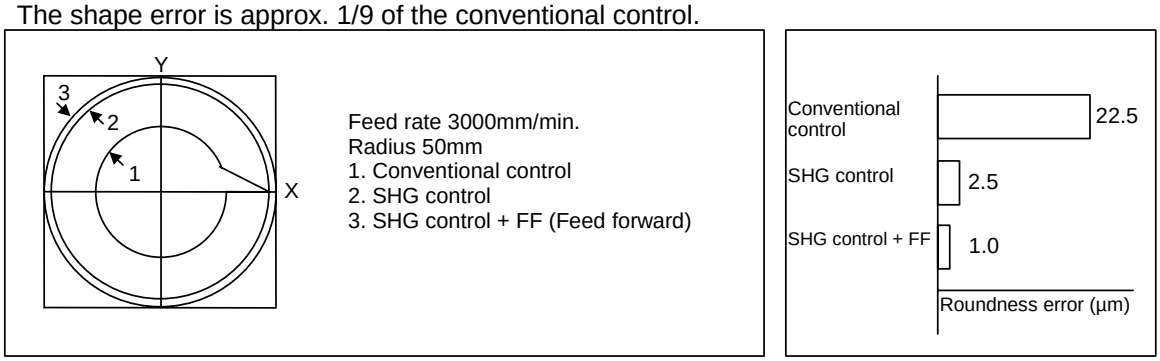
M system : ○ **L system : ○**

This is a high-response and stable position control method using the servo system (MDS-□-V□/SVJ3). This SHG control realizes an approximately three-fold position loop gain equally compared to the conventional control method.
The features of the SHG control are as follows.

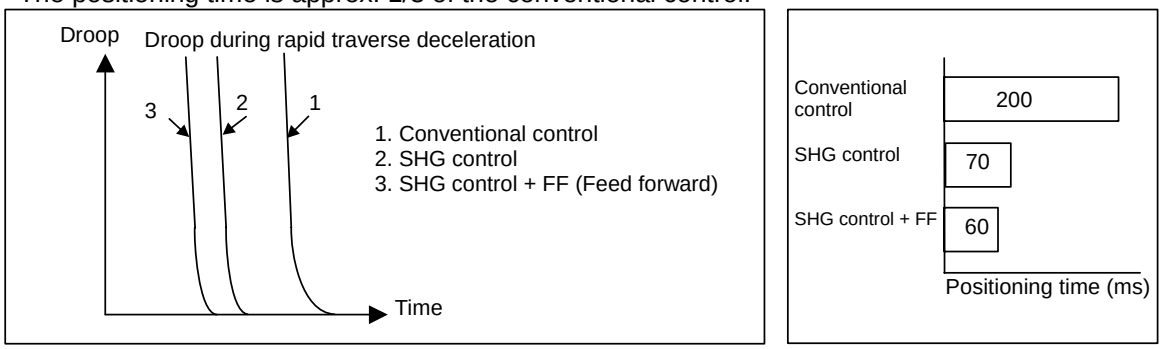
(1) The acceleration/deceleration becomes smoother, and the mechanical vibration can be suppressed (approx. 1/2) during acceleration/deceleration. (In other words, the acceleration/ deceleration time constant can be shortened.)



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



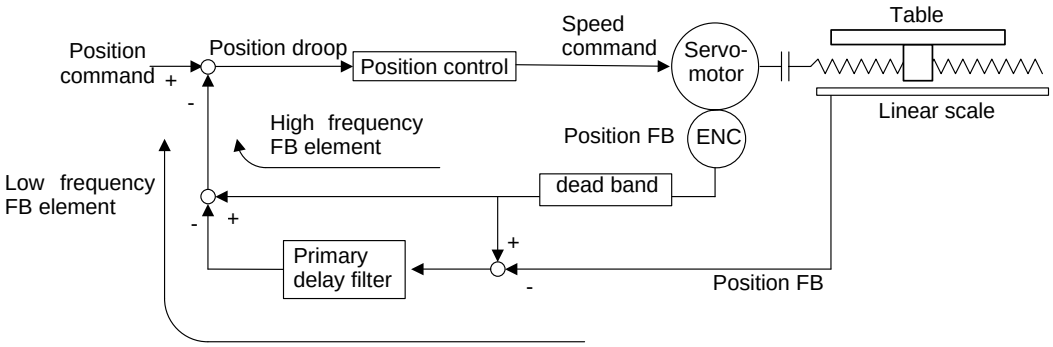
(3) The positioning time is approx. 1/3 of the conventional control.



13.2.2 Dual Feedback

M system : O **L system : O**

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.



Dual feedback control

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

M system : O **L system : O**

This function compensates the error in the protrusion shape caused by lost motion at the arc quadrant changeover section during circular cutting.

14. Automation Support Functions

14.1 Measurement

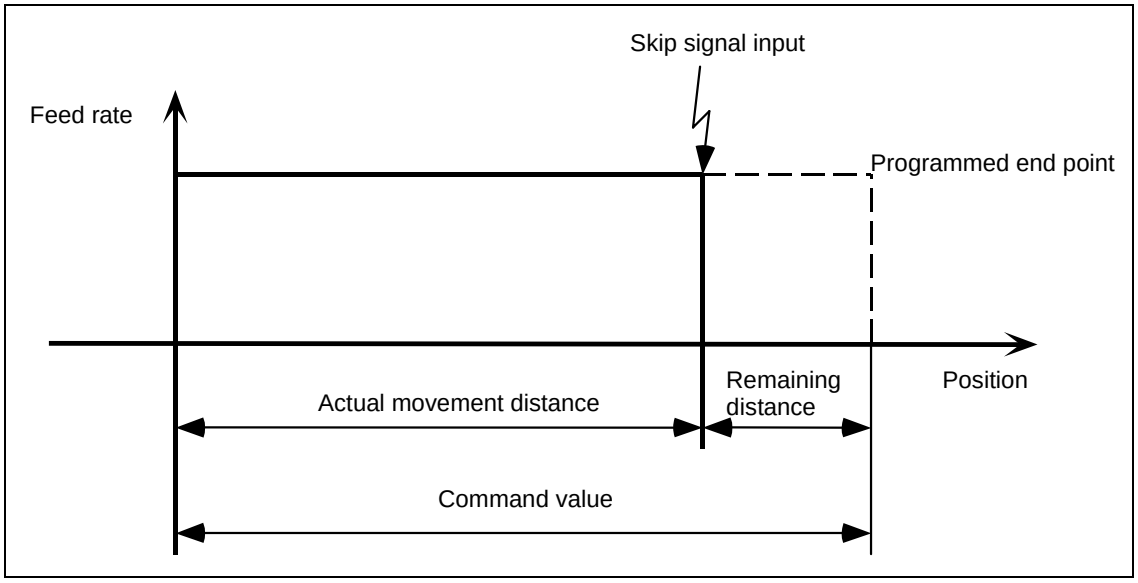
14.1.1 Skip

14.1.1.1 Skip

M system : Δ **L system : Δ**

When the external skip signal is input during linear interpolation with the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the commands in the next block are executed.

G31 Xx1 Yy1 Zz1 Ff1 ;	
G31	: Measurement command
Xx1, Yy1, Zz1	: Command values
Ff1	: Feed rate



When the G31 command is issued, acceleration/deceleration is accomplished in steps (time constant = 0).

There are two types of skip feed rate.
(1) Feed rate based on program command when F command is present in program
(2) Feed rate based on parameter setting when F command is not present in program

(Note 1) The approximate coasting distance up to feed stop based on the detection delay in the skip signal input is calculated as below.

$$\delta \fallingdotseq \frac{F}{60} \times (Tp + t)$$

δ : Coasting distance (mm)
F : G31 rate (mm/min)
Tp : Position loop time constant (s) = (position loop gain)$^{-1}$
T : Response delay time of 0.0035 (s)

(Note 2) Skipping during machine lock is not valid.

14.1.1.2 Multiple-step Skip

M system : Δ **L system : Δ**

(1) G31.n method

This function realizes skipping by designating a combination of skip signals for each skip command (G31.1, G31.2, G31.3).
The combination of the skip signals 1, 2, 3 and 4 are designated with parameters for each G code (G31.1, 31.2, 31.3), and the skip operation is executed when all signals in the combination are input.

G31.n Xx1 Yy1 Zz1 Ff1 ;	
G31.n	: Skip command (n=1, 2, 3)
Xx1, Yy1, Zz1	: Command format axis coordinate word and target coordinates
Ff1	: Feed rate (mm/min)

(2) G31Pn method

As with the G31.n method, the valid skip signal is designated and skip is executed. However, the method of designating the valid skip signal differs.
The skip signals that can be used are 1 to 4. Which is to be used is designated with P in the program. Refer to Table 1 for the relation of the P values and valid signals.
Skip can be executed on dwell, allowing the remaining dwell time to be canceled and the next block executed under the skip conditions (to distinguish external skip signals 1 to 4) set with the parameters during the dwell command (G04).

G31 Xx1 Yy1 Zz1 Pp1 Ff1 ;	
G31	: Skip command
Xx1, Yy1, Zz1	: Command format axis coordinate word and target coordinates
Pp1	: Skip signal command
Ff1	: Feed rate (mm/min)

(a) Specify the skip rate in command feedrate F. However, F modal is not updated.
(b) Specify skip signal command in skip signal command P. Specify the P value in the range of 1 to 15. If it exceeds the specified range, a program error occurs.
(c) When the skip signals are commanded in combination, the skip operation takes place with OR result of those signals.

Table 1 Valid skip signals

Skip signal command P	Valid skip signal			
	4	3	2	1
1				○
2			○	
3			○	○
4		○		
5		○		○
6		○	○	
7		○	○	○
8	○			
:	:	:	:	:
13	○	○		○
14	○	○	○	
15	○	○	○	○

14.1.2 Automatic Tool Length Measurement

M system : Δ **L system : Δ**

(1) Automatic Tool Length Measurement (M system)

This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset amount for that tool. If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current compensation amount is further corrected.

If the compensation amount at this time is one type, the compensation amount is automatically corrected; if there is a distinction between the tool length compensation amount and wear compensation amount, the wear amount is automatically corrected.

G37 Z_R_D_F_ ;

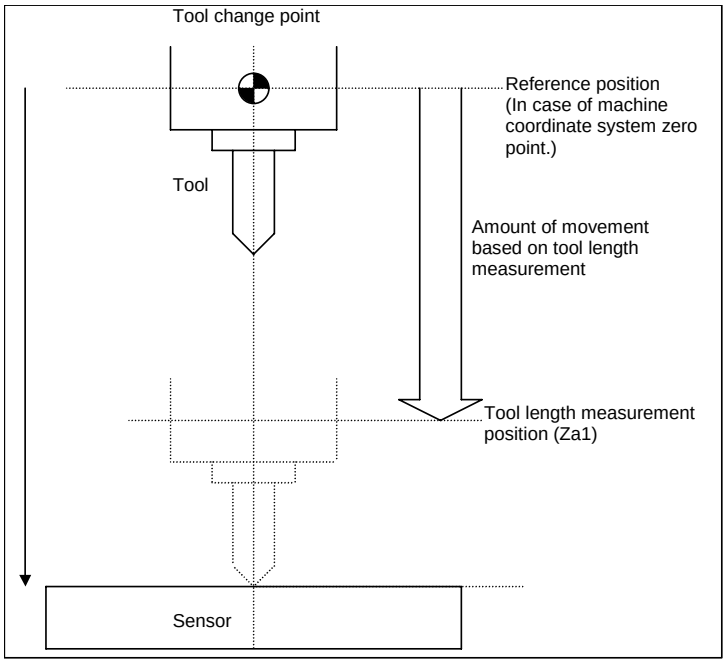
Z : Measurement axis address and measurement position coordinate. ... X, Y, Z, α (where α is an optional axis)

R : The distance between the point at which tool movement is to start at the measurement speed and the measurement position.

D : The range in which the tool is to stop.

F : The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.



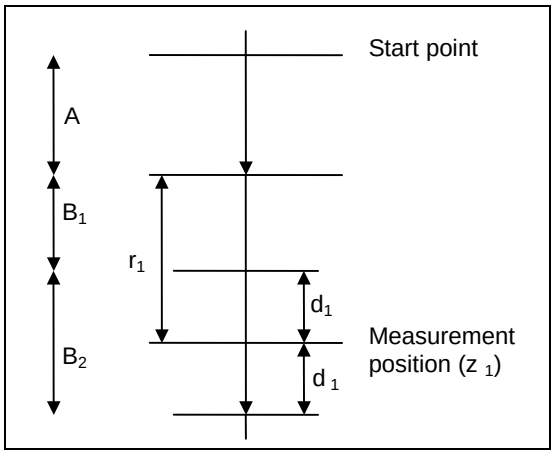
At this time, the tool length offset amount has a minus ("-") value.

Example of program

```
G28  Z0 ;
T01 ;
M06  T02 ;
G43  G00  Z0  H01 ;
G37  Z–300.  R10.D2.F10 ;
  .
  .
```

In this case, the distance (H01 = Za1 – z0) from the tool T01 tip to the top of the measurement sensor is calculated as the tool length offset amount which is then registered in the tool offset table.

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



Area A	: Moves with rapid traverse feed rate.
Areas B1, B2	: Moves with the measurement speed (f_1 or parameter setting)

If a sensor signal is input in area B_1, an error will occur.
If a sensor signal is not input in the area B_2, an error will occur.

(2) Automatic tool length measurement (L system)

This function moves the tool in the direction of the tool measurement position by the commanded value between the measurement start position to the measurement position, it stops the tool as soon as it contacts the sensor and calculates the difference between the coordinates when the tool has stopped and commanded coordinates. It registers this difference as the tool length offset amount for that tool. If compensation has already been applied to the tool, it is moved in the direction of the measurement position with the compensation still applied, and when the measurement and calculation results are such that a further compensation amount is to be provided, the current wear compensation amount is further corrected.

G37 α_R_D_F_ ;

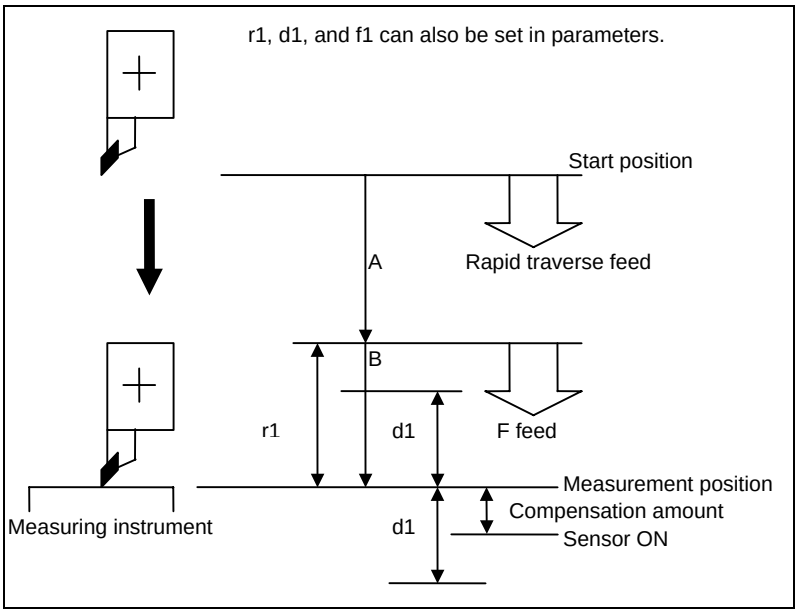
α : Measurement axis address and measurement position coordinate. ... X, Z

R : The distance between the point at which tool movement is to start at the measurement speed and the measurement position. (Always a radial value: incremental value)

D : The range in which the tool is to stop. (Always a radial value: incremental value)

F : The measurement rate.

When R_, D_ and F_ have been omitted, the values set in the parameters are used.



When the tool moves from the start position to the measurement position specified in G37 x1 (z1), it passes through the A area at rapid traverse. Then, it moves at the measurement rate set in F command or parameter from the position specified in r1. If the measurement position arrival signal (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 d1, an error occurs.

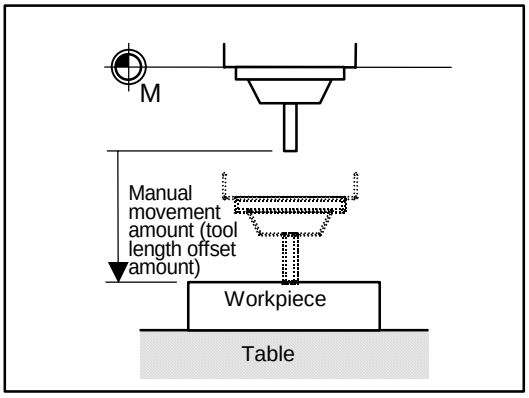
(Note 1) The measurement position arrival signal (sensor signal) is also used as the skip signal.
(Note 2) This is valid for the G code lists 2 and 3.

14.1.3 Manual Tool Length Measurement 1

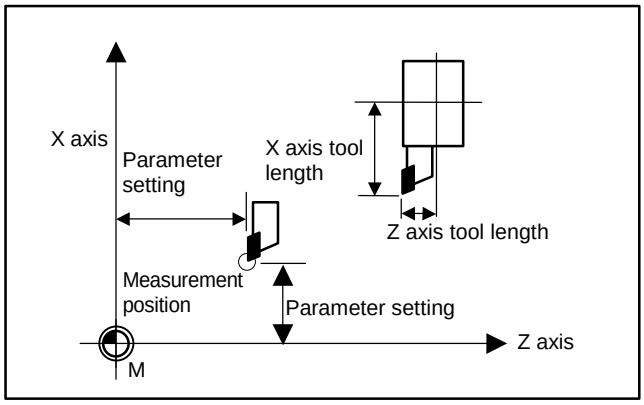
M system : Δ **L system : Δ**

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 point, this function enables the distance from the tool tip to the measurement position (top of workpiece) to be measured and registered as the tool length offset amount.



(2) Manual tool length measurement I
[L system]
A measurement position (machine coordinates) to match the tool nose on the machine is preset and the tool nose is set to the measurement position by manual feed, then the operation key is pressed, thereby automatically calculating the tool offset amount and setting it as the tool length offset amount.



Measurement method

(a) Preset the machine coordinates of the measurement position in a given parameter as the measurement basic value.
(b) Select a tool whose tool length offset amount is to be measured.
(c) Set the tool nose to the measurement position by manual feed.
(d) Press the input key. The tool length offset amount is calculated and displayed on the setting area.
Tool length offset amount = machine coordinates - measurement basic value
(e) Again press the input key to store the value in the memory as the tool length offset amount of the tool.

14.2 Tool Life Management

14.2.1 Tool Life Management

14.2.1.1 Tool Life Management I

M system : Δ **L system : Δ**

(1) M system

For the tool mounted on the spindle, 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.

(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

M system : Δ **L system : Δ**

(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 a tool selection (T) command, then outputs data of such usable spare tool. The spare tool can be selected in two ways: the tools are selected in order they were registered in the group or the tool whose remaining life is the longest of all in the group is selected.

(2) L system

The life of each tool (time and frequency) is controlled, and when the life is reached, a spare tool that is the same type is selected from the group where the tool belongs and used.

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

14.2.2 Number of Tool Life Management Sets

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

80 sets

M system : - **L system : Δ**

100 sets

M system : Δ **L system : -**

14.3 Others

14.3.1 Programmable Current Limitation

M system : ○ **L system : ○**

This function allows the current limit value of the servo axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc.
The commanded current limit value is designated with a ratio of the limit current to the rated current. The current limit value can also be set from the window function and setting and display unit.
The validity of the current limit can be selected with the external signal input.
However, the current limit value of the PLC axis cannot be rewritten.

G10 L14 X dn ;	
L14	: Current limit value setting (+ side/– side)
X	: Axis address
dn	: Current limit value 1% to 300%

(1) If the current limit is reached when the current limit is valid, the current limit reached signal is output.
(2) The following two modes can be used with external signals as the operation after the current limit is reached.
- Normal mode
  The movement command is executed in the current state.
  During automatic operation, the movement command is executed to the end, and then the next block is moved to with the droops still accumulated.
- Interlock mode
  The movement command is blocked (internal interlock).
  During automatic operation, the operation stops at the corresponding block, and the next block is not moved to.
  During manual operation, the following same direction commands are ignored.

(3) During the current limit, the droop generated by the current limit can be canceled with external signals. (Note that the axis must not be moving.)
(4) The setting range of the current limit value is 1% to 300%. Commands that exceed this range will cause a program error.
"P35 CMD VALUE OVER" will be displayed.
(5) If a decimal point is designated with the G10 command, only the integer will be valid.
(Example) G10 L14 X10.123 ; The current limit value will be set to 10%.
(6) For the axis name "C", the current limit value cannot be set from the program (G10 command). To set from the program, set the axis address with an incremental axis name, or set the axis name to one other than "C".

15. Safety and Maintenance

15.1 Safety Switches

15.1.1 Emergency Stop

M system : ○ **L system : ○**

All operations are stopped by the emergency stop signal input and, at the same time, the drive section is stopped using the dynamic brake and the movement of the machine is stopped.
The servo ready signal is turned OFF.

15.1.2 Data Protection Key

M system : ○ **L system : ○**

With the input from the user PLC, it is possible to prohibit the setting and deletion of parameters and the editing of programs from the 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

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.

15.2.2 NC Alarm

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 warning	Description
Operation alarm	This alarm occurring due to incorrect operation by the operator during NC operation and that by machine trouble are displayed.
Servo alarm	This alarm describes errors in the servo system such as the servo drive unit, motor and encoder.
Spindle alarm	This alarm describes errors in the spindle system such as the spindle drive unit, motor and encoder.
MCP alarm	An error has occurred in the drive unit and other interfaces.
System alarm	This alarm is displayed with the register at the time when the error occurred on the screen if the system stops due to a system error.
Absolute position detection system alarm	An alarm in the absolute position detection system is displayed.
Program error	This alarm occur during automatic operation, and the cause of this alarm is mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs which conform to the specification have not been prepared.

15.2.3 Operation Stop Cause

M system : ○ **L system : ○**

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

15.2.4 Emergency Stop Cause

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

M system : ○ **L system : ○**

When overheating is detected in the control unit or the CNC CPU module, the alarm is displayed. If the system is in auto run at the time, run is continued, but it cannot be started after reset or M02/M30 run ends. (It can be started after block stop or feed hold.)
When the temperature falls below the specified temperature, the alarm is released and the overheat signal is turned OFF.

15.2.6 Battery Alarm/Warning

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)

M system : ○ **L system : ○**

When limit switches and dogs have been attached to the machine and a limit switch has kicked a dog, the movement of the machine is stopped by the signal input from the limit switch.
At the same time, the alarm output is sent to the machine.
The stroke end state is maintained and the alarm state is released by feeding the machine in the reverse direction in the manual mode to disengage the dog.

15.3.2 Stored Stroke Limit

The stored stroke limits I, II, IIB, IB and IC are handled as follows.

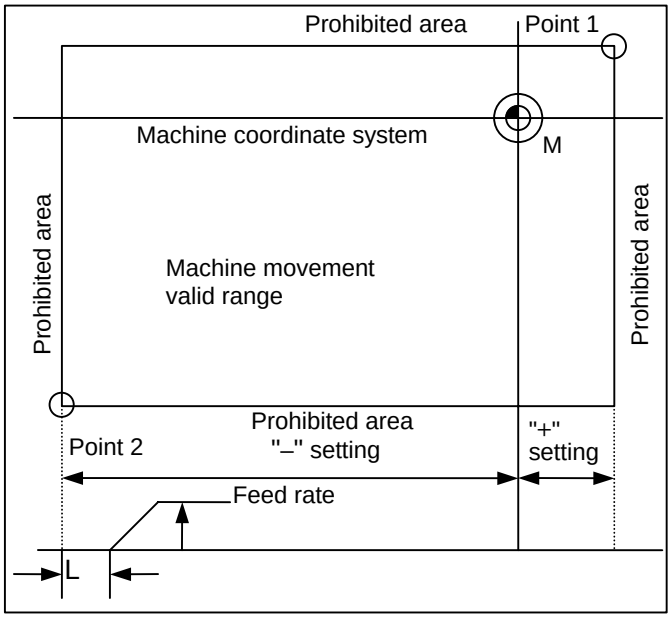
Type	Prohibited range	Explanation
I	Outside	• Set by the machine tool builder. • When used with II, the narrow range designated by the two types becomes the movement valid range. • Can be rewritten with window function.
II	Outside	• Set by the user. • The change or function of parameter can be turned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

15.3.2.1 Stored Stroke Limit I/II

M system : O **L system : O**

(1) Stored Stroke Limit I

This is the stroke limit function used by the machine maker, and the area outside the set limits is the entrance prohibited area.
The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit II function described in the following section, and the tolerable area of both functions is the movement valid range.
The setting range is -99999.999 to +99999.999mm.
The stored stroke limit I function is made valid not immediately after the controller power is turned ON but after reference point return.
The stored stroke limit I function will be invalidated if the maximum and minimum values are set to the same data.

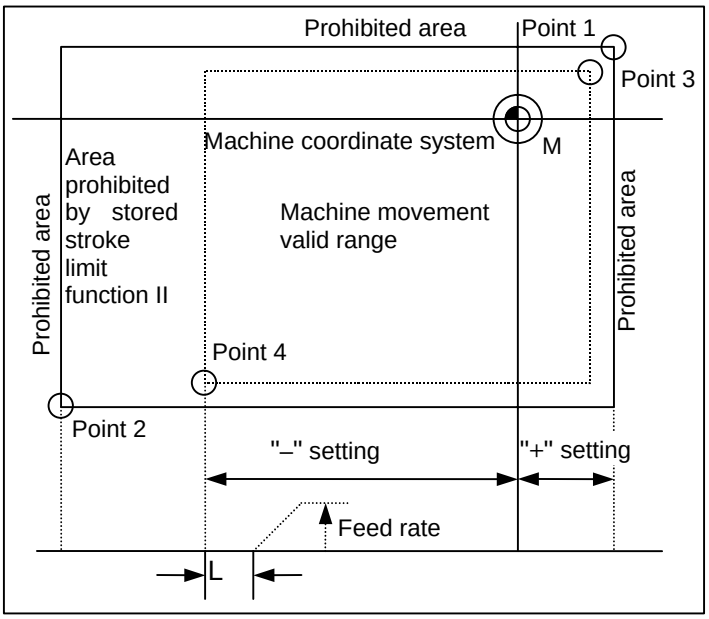


The values of points 1 and 2 are set using the coordinate values in the machine coordinate system.

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.
The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.

(2) Stored Stroke Limit II

This is the stroke limit function which can be set by the user, and the area outside the set limits is the prohibited area.
The maximum and minimum values for each axis can be set by parameters. The function itself is used together with the stored stroke limit I function described in the foregoing section, and the tolerable area of both functions is the movement valid range.
The setting range is –99999.999 to +99999.999mm.
The stored stroke limit II function will be invalidated if the maximum and minimum parameter values are set to the same data.



The values of points 3 and 4 are set with the coordinate values in the machine coordinate system.

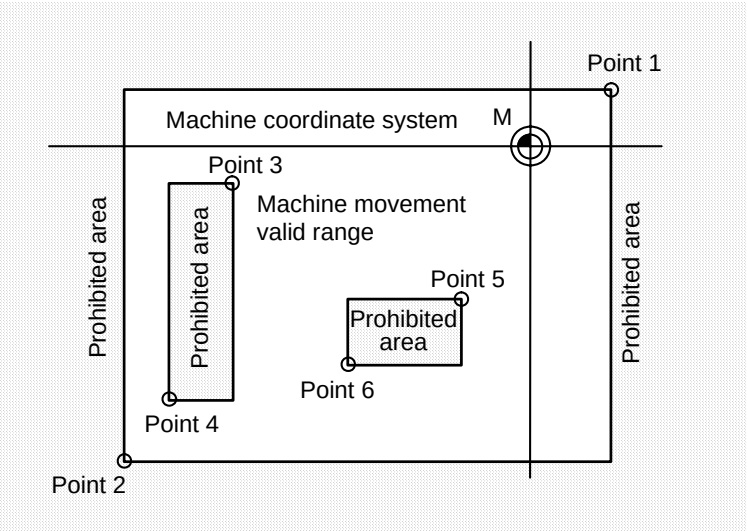
The area determined by points 1 and 2 is the prohibited area set with stored stroke limit I.

All axes will decelerate and stop if an alarm occurs even for a single axis during automatic operation. Only the axis for which the alarm occurs will decelerate and stop during manual operation. The stop position must be before the prohibited area.
The value of distance "L" between the stop position and prohibited area differs according to the feed rate and other factors.
The stored stroke limit II function can also be invalidated with the parameter settings.

15.3.2.2 Stored Stroke Limit IB

M system : Δ **L system : Δ**

Three areas where tool entry is prohibited can be set using the stored stroke limit I, stored stroke limit II, IIB and stored stroke limit IB functions.



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.

When an attempt is made to move the tool beyond the set range, an alarm is displayed, and the tool decelerates and stops. If the tool has entered into the prohibited area and an alarm has occurred, it is possible to move the tool only in the opposite direction to the direction in which the tool has just moved. This function is an option.

Precautions

- Bear in mind that the following will occur if the same data is set for the maximum and minimum value of the tool entry prohibited area:
  (1) When zero has been set for the maximum and minimum values, tool entry will be prohibited in the whole area.
  (2) If a value other than zero has been set for both the maximum and minimum values, it will be possible for the tool to move in the whole area.

15.3.2.3 Stored Stroke Limit IIB

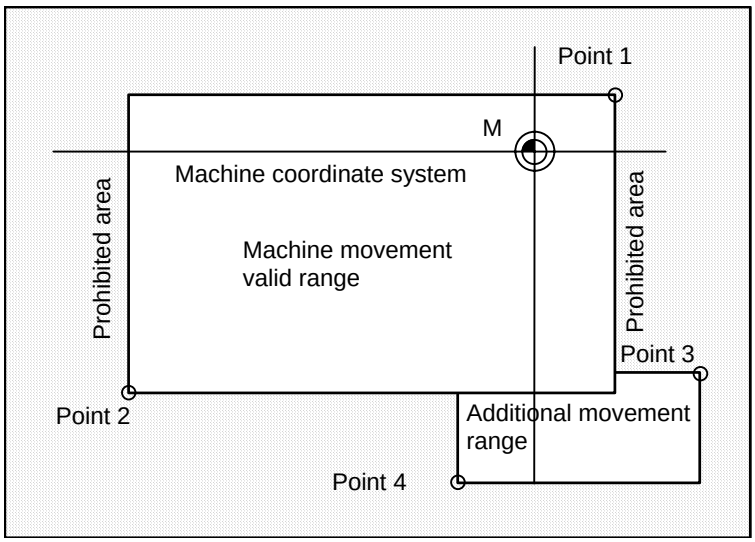
M system : Δ **L system : Δ**

A parameter is used to switch between this function and stored stroke limit II. With stored stroke limit IIB, the range inside the boundaries which have been set serves as the tool entry prohibited area.

15.3.2.4 Stored Stroke Limit IC

M system : Δ **L system : Δ**

The boundary is set for each axis with the parameters. The inside of the set boundary is the additional movement range.
This cannot be used with soft limit IB.



The 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.

15.3.4 Chuck/Tail Stock Barrier Check

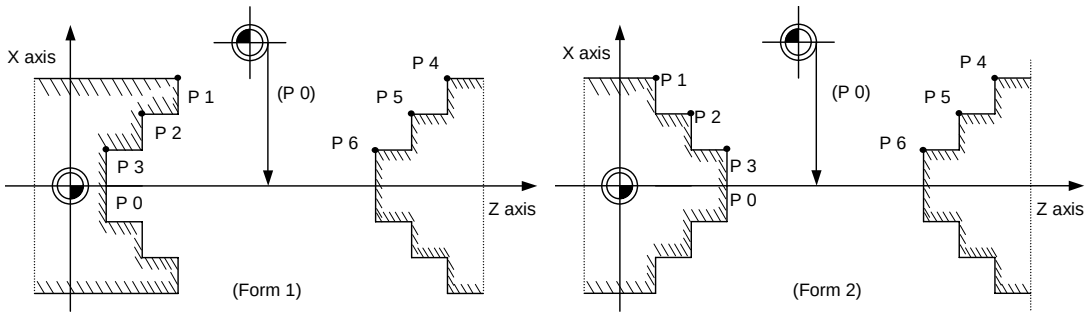
M system : - **L system : O**

By limiting the tool nose point move range, this function prevents the tool from colliding with the chuck or tail stock because of a programming error.
When a move command exceeding the area set in a given parameter is programmed, the tool is stopped at the barrier boundaries.

Program format

G22 ; ..... Barrier ON **G23 ;** ..... Barrier OFF (cancel)

(1) When the machine is about to exceed the area, the machine is stopped and an alarm is displayed. To cancel the alarm, execute reset.
(2) The function is also effective when the machine is locked.
(3) This function is valid when all axes for which a barrier has been set have completed reference point return.
(4) The chuck barrier/tail stock barrier can be set independently for part system 1 and part system 2.
(5) Chuck barrier/tail stock barrier setting



The chuck barrier and tail stock barrier are both set with the machine coordinate by inputting one set of three-point data in the parameter. Points P1, P2 and P3 are the chuck barrier, and points P4, P5 and P6 are the tail stock barrier. The X axis is set with the coordinate value (radius value) from the workpiece center, and the Z axis is set with the basic machine coordinate system coordinate.
Point P0 is the chuck barrier and tail stock barrier's basic X coordinates, and the workpiece center coordinate in the basic machine coordinate system is set.
The barrier area is assumed to be symmetrical for the Z axis, and if the X axis coordinate of barrier point P_ is minus, the sign is inverted to plus and the coordinate is converted for a check.
Set the absolute values of the X axis coordinates of the barrier points as shown below:
P1 >= P2 >= P3, P4 >= P5 >= P6
(However, this need not apply to the Z axis coordinates.)

15.3.5 Interlock

M system : O **L system : O**

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.
(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.

15.3.6 External Deceleration

M system : O **L system : O**

This function reduces the feed rate to the deceleration speed set by the parameter when the external deceleration input signal 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.

15.3.9 Door Interlock

15.3.9.1 Door Interlock I

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 units so that the motors are no longer driven.
When the door open signal has been input during automatic operation, the suspended machining can be resumed by first closing the door concerned and then initiating cycle start again.

Description of operation

When a door is open
The NC system operates as follows when the door open signal is input:

(1) It stops operations.
  1. When automatic operation was underway
    The machine is set to the feed hold mode, and all the axes decelerate and stop.
    The spindle also stops.
  2. When manual operation was underway
    All the axes decelerate and stop immediately.
    The spindle also stops.
(2) The complete standby status is established.
(3) After all the servo axes and the spindle have stopped, the ready OFF status is established.
(4) The door open enable signal is output.
  Release the door lock using this signals at the PLC.

When a door is closed
After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the door open signal is set to OFF.

(5) All the axes are set to ready ON.
(6) The door open enable signal is set to OFF.

Resuming operation

(7) When automatic operation was underway
  Press the AUTO START button.
  Operation now resumes from the block in which machining was suspended when the door open signal was input.
(8) When manual operation was underway
  Axis movement is commenced when the axis movement signals are input again.
(9) Spindle rotation
  Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again.

15.3.9.2 Door Interlock II

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 units so that the motors are no longer driven.
With the door interlock function established by the door open II signal, automatic start can be enabled even when the door open signal has been input. However, the axes will be set to the interlock status.

Description of operation

When a door is open

The NC system operates as follows when the door open II signal is input:

(1) It stops operations.
All the axes decelerate and stop.
The spindle also stops.
(2) The complete standby status is established.
(3) After all the servo axes and the spindle have stopped, the ready OFF status is established.
However, the servo ready finish signal (SA) is not set to OFF.

When a door is closed

After the PLC has confirmed that the door has been closed and locked, the NC system operates as follows when the door open signal is set to OFF.

(4) All the axes are set to ready ON.
(5) The door open enable signal is set to OFF.

Resuming operation

(6) When automatic operation was underway
The door open signal is set to OFF, and after the ready ON status has been established for all the axes, operation is resumed.
(7) When manual operation was underway
Axis movement is commenced when the axis movement signals are input again.
(8) Spindle rotation
Restore the spindle rotation by inputting the forward rotation or reverse rotation signal again

(Note 1) Concerning the handling of an analog spindle
The signals described in this section are valid in a system with serial 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

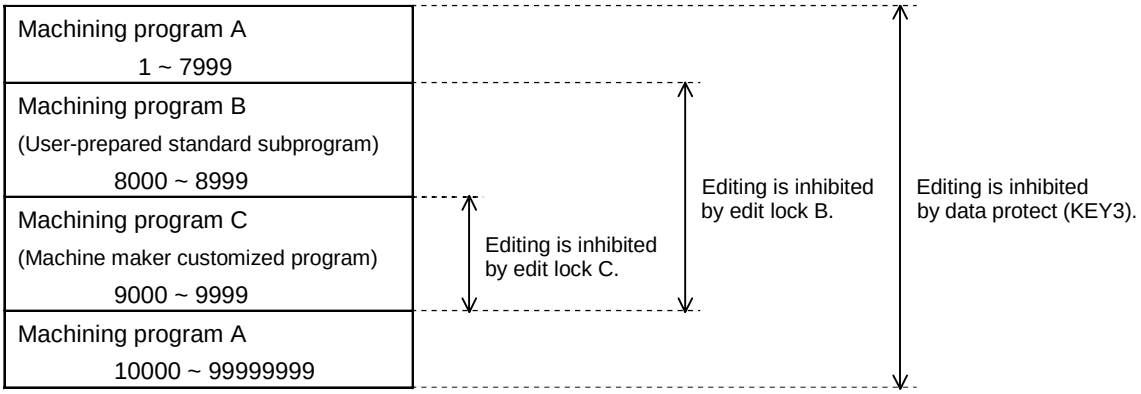
M system : O **L system : O**

This function is used to prohibit changing the setup parameter.

15.3.11 Program Protect (Edit Lock B, C)

M system : O **L system : O**

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



15.3.12 Program Display Lock

M system : O **L system : O**

This function allows the display of only a target program (label address 9000) to be invalidated for the program display in the monitor screen, etc.
The operation search of a target program can also be invalidated.
The validity of the display is selected with the parameters. The setting will be handled as follows according to the value.

0: Display and search are possible.
1: Display of the program details is prohibited.
2: Display and operation search of the program details are prohibited.

The program details are not displayed in the prohibited state, but the program number and sequence number will be displayed.

15.3.13 Safety Observation

M system : Δ **L system : Δ**

This function is composed of the following three functions.

[Dual safety circuit function]
PLC CPU and NC CPU separately control the Input/Output signal of the safety signal unit.
The state of the disagreement of the Input/Output signal of each CPU is observed by the safety signal compare. When an error is detected during observation, the main power for the drive is shut.

[Dual emergency stop function]
PLC CPU, NC CPU and drive CPU separately observe the input of emergency stop.
The main power for the drive can be shut by controlling the contactor from PLC, NC, and drive CPU respectively when the emergency stops.

[Dual speed monitor function]
NC CPU and drive CPU separately observe the following.

- Observe the open and close state signal of the safety door detected with a different circuit
- Observe that the command speed should not exceed the speed set by the parameter (safety speed).
- Observe that the motor rotation speed should not exceed the rotation speed set by parameter (safety rotation speed).

When an error is detected during observation, the main power for the drive is shut.

15.4 Maintenance and Troubleshooting

15.4.1 Operation history

M system : ○ **L system : ○**

This is a maintenance function which is useful for tracing down the history and NC operation information and analyzing trouble, etc. This information can be output as screen displays or as files.

(1) Screen display showing operation history and event occurrence times
The times/dates (year/month/day and hour/minute/second) and messages are displayed as the operation history data. The key histories, alarm histories and input/output signal change histories are displayed as the messages.
The part system information is displayed as the alarm histories.
For instance, "$1" denotes the first part system, and "$2" the second part system.
The history data containing the most recent operation history and event occurrence times (2,068 sets) are displayed on the "Operation history" screen. The most recent history data appears at the top of the screen, and the older data is displayed in sequence below.

(2) Outputting the data in the operation history memory
Information on the alarms occurring during NC operation and stop codes, signal information on the changes in the PLC interface input signals and the key histories can be output.

15.4.2 Data Sampling

M system : ○ **L system : ○**

The data sampling function can sample the NC internal data (speed output from the NC to the drive unit and feedback data from the drive unit, etc.) and output as text data.

15.4.3 NC Data Backup

M system : ○ **L system : ○**

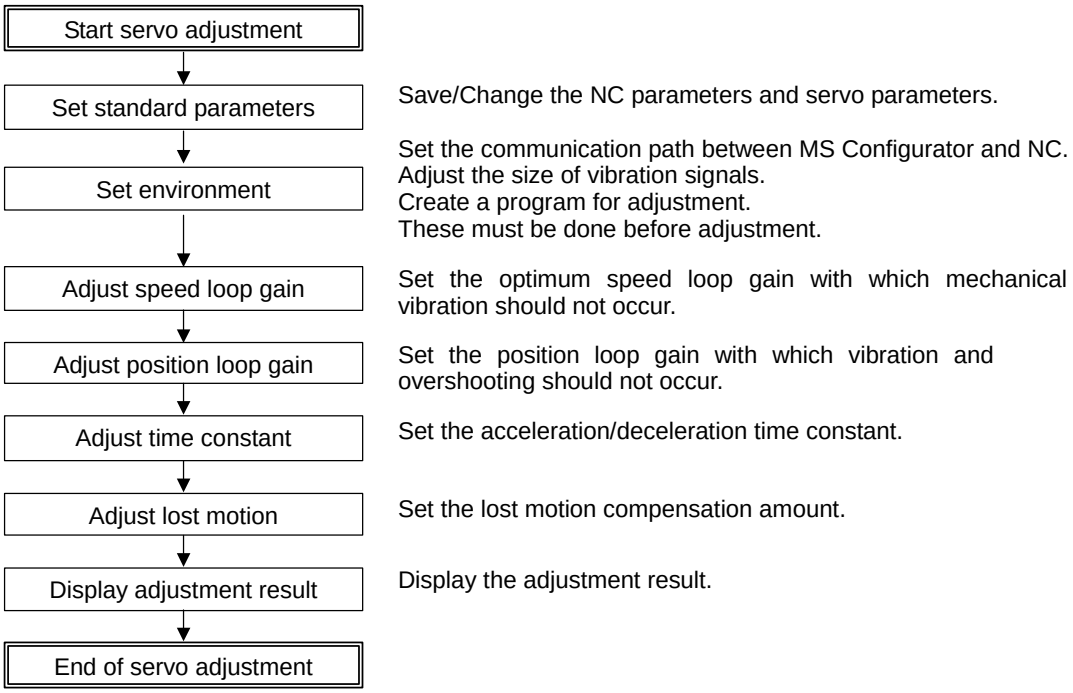
This function serves to back up the parameters and other data of the NC control unit.
The data can also be restored.

15.4.5 Servo Automatic Tuning

M system : Δ **L system : Δ**

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.

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.

(1) Environment setting

Function	Details
Communication path setting	Sets the communication path with NC.
Vibration signal setting	Adjusts the size of vibration signals.
Parameter setting	Saves and changes the servo parameters.
Program creation	Creates program for adjustment.

(2) Automatic adjustment function

Function	Details
Speed loop gain adjustment	Automatically adjusts the speed loop gain.
Position loop gain adjustment	Automatically adjusts the position loop gain.
Time constant adjustment	Automatically adjusts the acceleration/deceleration time.
Lost motion adjustment	Automatically adjusts the quadrant protrusion amount of the designated axis.
Batch adjustment	Automatically adjusts the above 4 items.

15.4.102 All Backup

M system : O **L system : O**

This function saves (backup) the screen data and each controller (PLC, NC) data to a GOT's CF card. And this function reloads (restores) that data to each device.
If this function is used, the backup is unnecessary for the MONITOR screen and each controller, and work improves.

16. Drive System

Refer to the following manuals for details on the servo and spindle system.
MDS-D series Specifications Manual (IB-1500011)
MDS-DH series Specifications Manual (IB-1500003)

16.1 Servo/Spindle

16.1.1 Feed Axis

16.1.1.1 MDS-D-V1/D-V2 (200V)

(1) Servo motor : HF□□-A48 (260 kp/rev)

M system : □ **L system : □**

(2) Servo motor : HP□□-A48 (260 kp/rev)

M system : □ **L system : □**

16.1.1.2 MDS-DH-V1/DH-V2 (400V)

(1) Servo motor : HF-H□□-A48 (260 kp/rev)

M system : □ **L system : □**

(2) Servo motor : HP-H□□-A48 (260 kp/rev)

M system : □ **L system : □**

16.1.1.3 MDS-D-SVJ3 (200V)

(1) Servo motor: HF**-A48 (260kp/rev)

M system : □ **L system : □**

(2) Servo motor: HF-KP**JW04 (260kp/rev)

M system : □ **L system : □**

16.1.2 Spindle

16.1.2.1 MDS-D-SP (200V)

M system : ☐ **L system : ☐**

16.1.2.2 MDS-DH-SP (400V)

M system : ☐ **L system : ☐**

16.1.2.3 MDS-D-SPJ3 (200V)

M system : ☐ **L system : ☐**

16.1.4 Power Supply

16.1.4.1 Power Supply : MDS-D-CV (200V)

M system : ☐ **L system : ☐**

16.1.4.2 Power Supply : MDS-DH-CV (400V)

M system : ☐ **L system : ☐**

16.1.4.3 AC Reactor for Power Supply

M system : ☐ **L system : ☐**

16.1.4.4 Ground Plate

M system : ☐ **L system : ☐**

17. Machine Support Functions

17.1 PLC

17.1.1 Built-in PLC Processing Mode

M system : ○ **L system : ○**

A built-in PLC is not usually used for MITSUBISHI CNC C70 differently from other MITSUBISHI CNC (70, 700, etc.).
The safety signal unit is used when the important signals safety observing is executed. GX Developer is used as development tool, but the instructions which can be used are limited.
Refer to the manual concerning the safety observing function for details.

17.1.2 PLC Functions

17.1.2.1 Built-in PLC Basic Function

M system : ○ **L system : ○**

As the PLC function of MITSUBISHI CNC C70, the PLC CPU of MITSUBISHI Programmable Controller MELSEC is used.
Select a PLC suitable for the control scale and the performance from several kinds of PLCs and use it.
Refer to the material of MITSUBISHI Programmable Controller MELSEC which can be used with MITSUBISHI CNC C70 for details.

17.1.2.2 NC Exclusive Instruction

M system : ○ **L system : ○**

NC exclusive instructions, which directly connected to memory in NC other than the MELSEC standard instructions, etc., are convenience instructions for NC.
These instructions can be programmed by MELSEC programming tool GX Developer as well as other standard instructions.

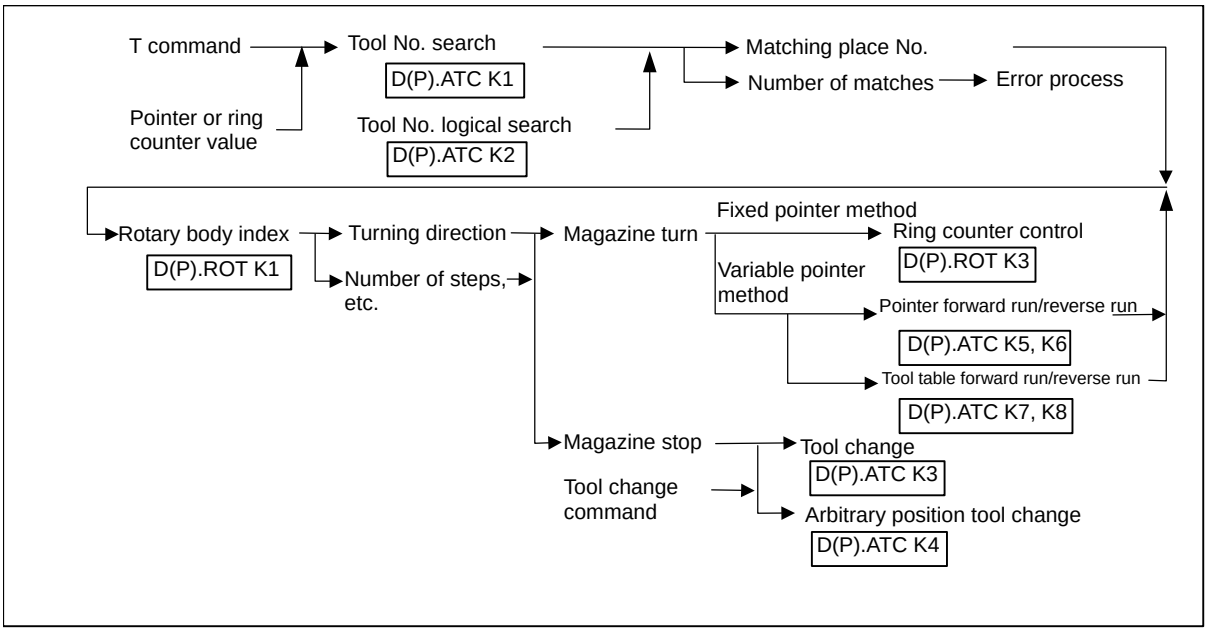
NC dedicated instructions include:

(1) ATC dedicated instruction (D(P).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 (D(P).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 D(P).ATC and D(P).ROT instructions when T is commanded or tool exchange is commanded is shown below.



17.1.4 Built-in PLC Capacity

M system : ○ **L system : ○**

This function is for the safety signal control.
Number of step is up to 1000 steps.

17.1.5 Machine Contact Input/Output I/F

M system : ○ **L system : ○**

The device is selected from the I/O unit of MITSUBISHI Programmable Controller MELSEC Q series.
Follow the manual of the I/O unit about the method of handling.

17.1.6 Ladder Monitor

M system : ○ **L system : ○**

This function enables the operating status of the sequence circuit to be checked on the MITSUBISHI Graphic Operation Terminal (GOT).
The monitor functions include the following.
(1) Circuit monitoring
(2) Batch monitor
(3) Entry monitoring

17.1.7 PLC Development

17.1.7.2 MELSEC Development Tool (GX Developer)

M system : ○ **L system : ○**

This function enables the data of the MELSEC CPU PLC programs 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.

17.1.10 GOT Connection

For the connection of MITSUBISHI Graphic Operation Terminal (GOT), refer to the material of GOT.
Only when GOT has been bus-connected with the DISPLAY interface of NC CPU or the basic base unit, the CNC exclusive use screen (CNC monitor function) can be displayed.
The size of GOT corresponds to SVGA and XGA.

17.2 Machine Construction

17.2.1 Servo OFF

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.

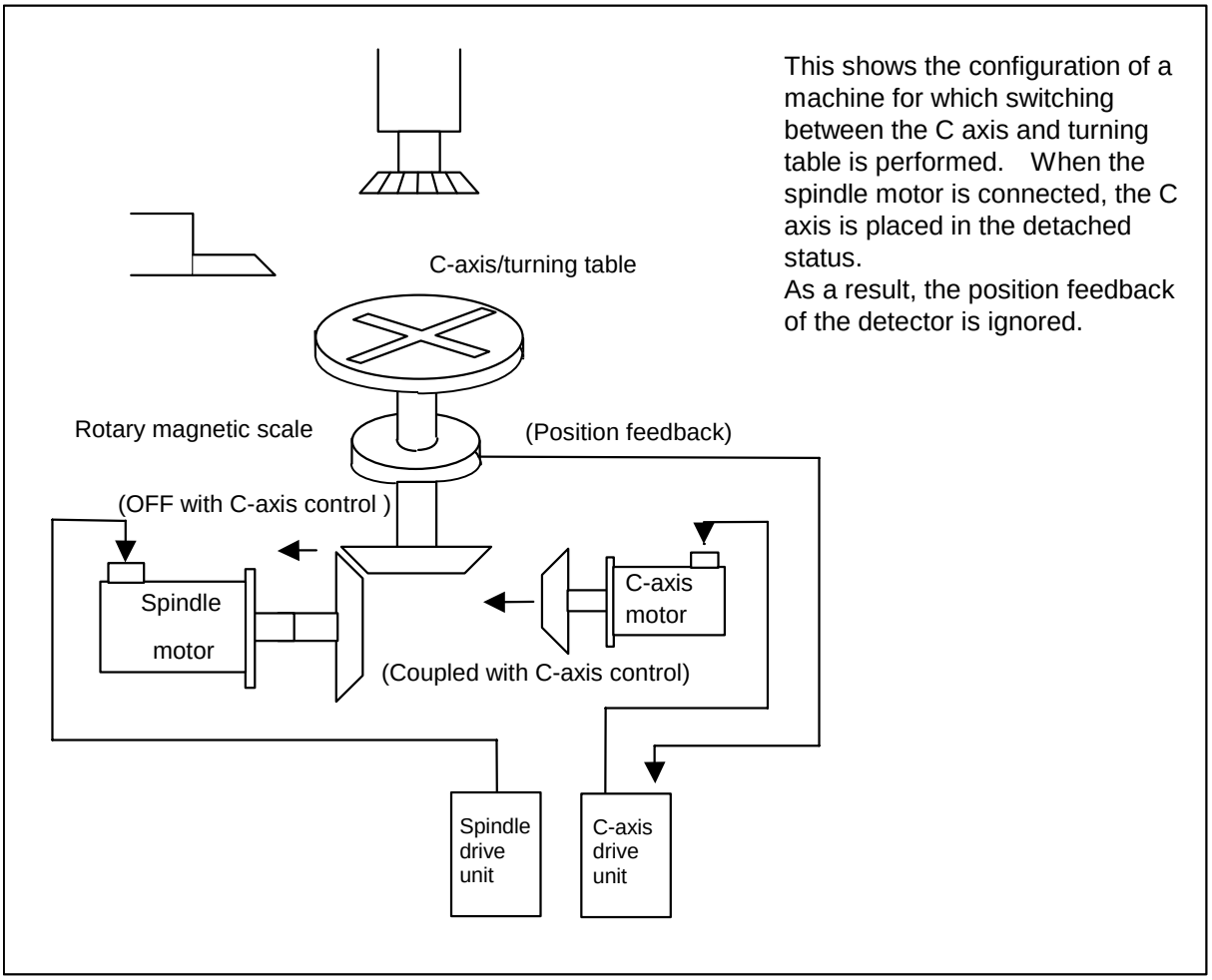
Mechanical handle

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

M system : Δ **L system : Δ**

This function enables the control axis to be freed from control. Conversely, an axis which has been freed from control can be returned to the control status.
This function enables the rotary table or attachments to be removed and replaced.
Automatic operation is disabled until the axis for which the axis detach command has been released completes its dog-type reference position return.



This shows the configuration of a machine for which switching between the C axis and turning table is performed. When the spindle motor is connected, the C axis is placed in the detached status.
As a result, the position feedback of the detector is ignored.

```
POSITION

   X    1 2 3 . 4 5 6
   Z        0 . 0 0 0 #1
   C    3 4 5 . 6 7 8 ><
```

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

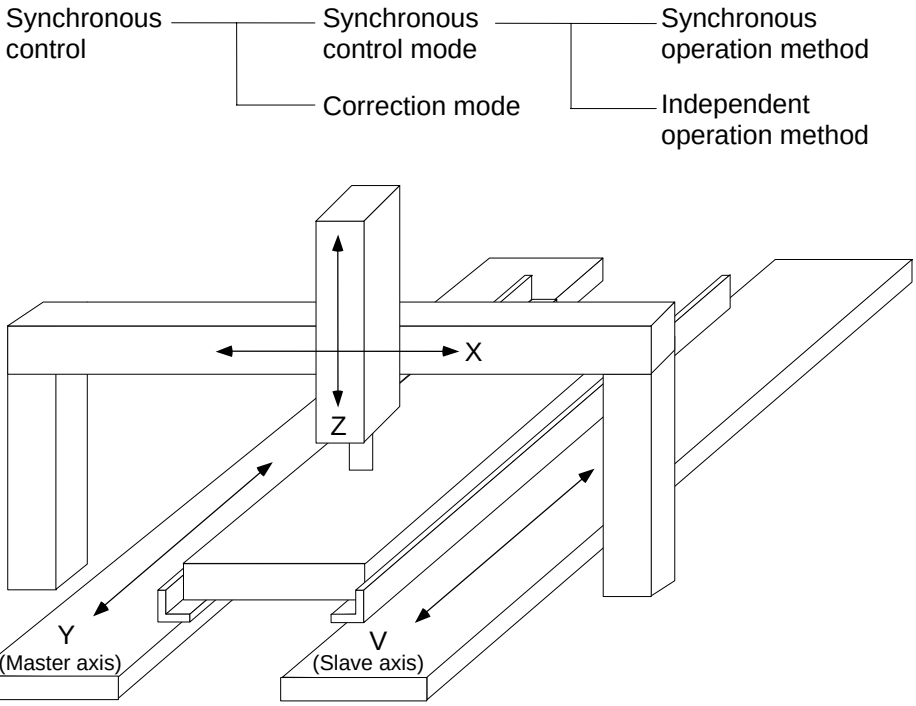
(Note) Axis detach can be executed even for the absolute position detection specifications axis, but when the axis is reinstalled, the zero point must be set.

17.2.3 Synchronous Control

M system : Δ **L system : -**

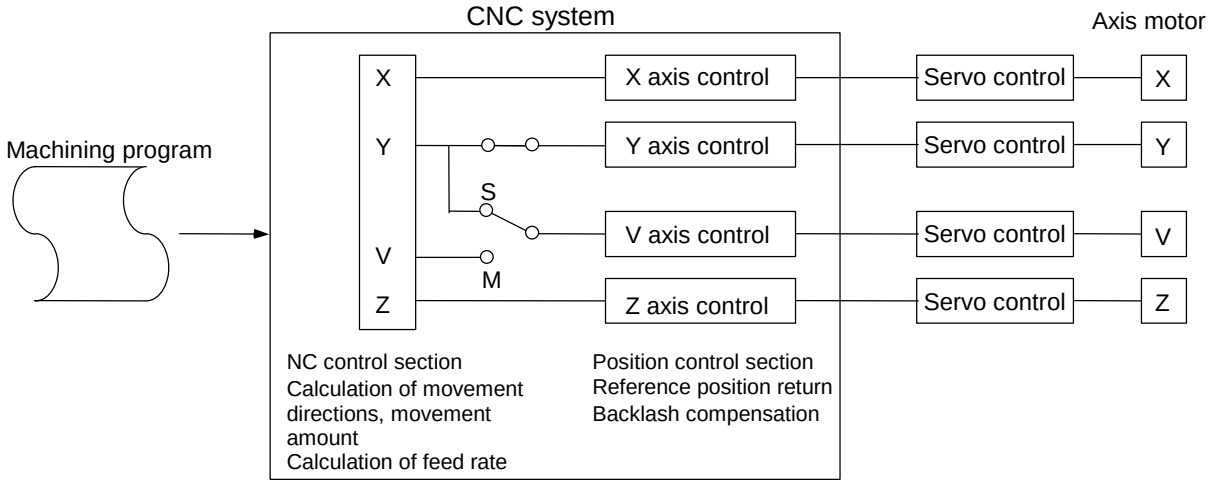
The synchronous control is a control method that both master and slave axes are controlled with the same movement command by designated the movement command for the master axis also to the slave axis. This function is assumed to be used in the large machine tool, etc. which drives one axis with two servo 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 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.



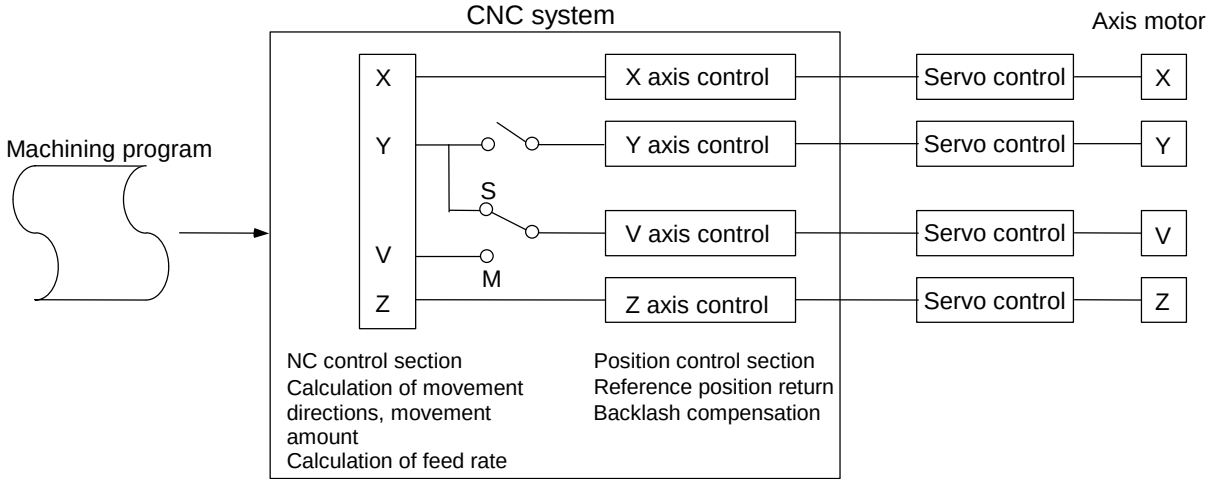
(1) Synchronous control mode
The following two operation methods are available in the synchronous control mode.

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



There is a function that checks the correlation between the positions of the master axis and slave axis at all times while the synchronous operation method is selected to stop the feed as alarm when the 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.



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".

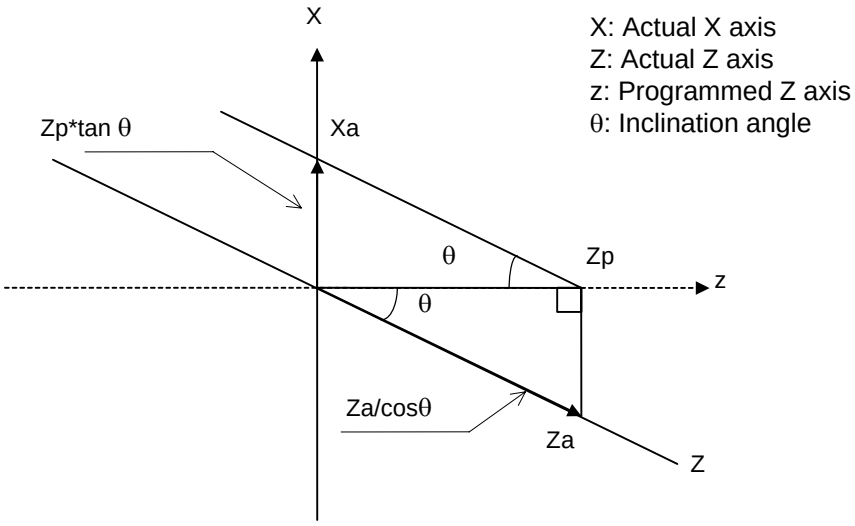
(2) Correction mode
The synchronization is temporary canceled to adjust the balance of the master and slave axes during the synchronous control mode in the machine adjustment. Each axis can be moved separately with the manual handle feed or the arbitrary feed in manual mode. If the operation mode other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

M system : - **L system : Δ**

Even when the control axes configuring that machine are mounted at an angle other than 90 degrees, this function enables it to be programmed and controlled in the same way as with an orthogonal axis.
The inclination angle is set using a parameter, and axes are controlled using the movement amounts of the axes which are obtained through conversion and compensation using this angle.

<Example of use> When the X axis serves as the basic axis and the Z axis serves as the inclined axis



Zp, the Z-axis position on the programmed coordinates (on the orthogonal coordinates), is the position of Xa and Za which are produced by synthesis of X axis and Z axis.
Therefore, the Z-axis (inclined axis) movement amount is expressed by the following formula:

$Za = Zp/\cos\theta$……(1)

The X-axis (basic axis) movement amount is compensated by the inclined movement of the Z axis, and it is expressed as follows:

$Xa = Xp - Zp \times \tan\theta$……..(2)

The Z-axis (inclined axis) speed is as follows:

$Fa = Fp/\cos\theta$

> Xa, Za and Fa are the actual movement amounts and speed.
> Xp, Zp and Fp are the movement amounts and speed on the program coordinates.

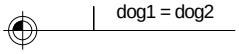
17.2.5 Position Switch

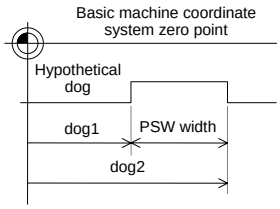
M system : O **L system : O**

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 (PSW1 to PSW16). 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



17.3 PLC Operation

17.3.1 Arbitrary Feed in Manual Mode

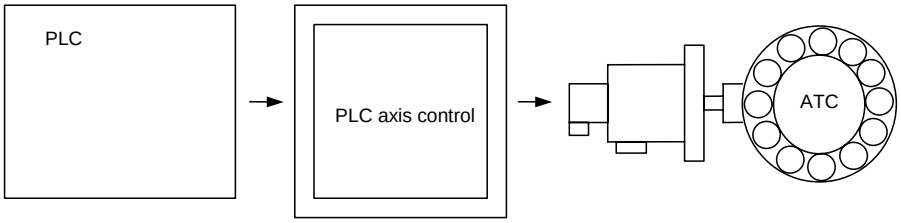
M system : ○ **L system : ○**

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).

17.3.3 PLC Axis Control

M system : Δ **L system : Δ**

Over and above the NC control axes, this function enables axes to be controlled independently by commands based on the PLC.



Item	Details
No. of control axes	Max. 7 axes
Simultaneous control axes	The PLC control axis is controlled independently of the CNC control axis. Simultaneous start of multiple PLC axes is possible.
Command unit	Min. command unit **(Note 1)** 0.001mm (0.0001 inch) 0.0001mm (0.00001 inch)
Feedrate	0 to 1000000mm/min (0 to 100000inch/min) (The feedrate is fixed regardless of the unit system.)
Movement commands	Incremental value commands from the current position. Absolute value commands of the machine coordinate system. 0 to ±99999999 **(Note 1)**
Operation modes	Rapid traverse, cutting feed Jog feed (+), (-) Reference position return feed (+), (-) Handle feed
Backlash compensation	Provided
Stroke end	Not provided
Soft limit	Provided
Rotation axis commands	Provided Absolute value commands..... Rotation amount within one rotation. (Rotates the remainder divided by rotational axis division count.) The axis rotates in + direction when the command value is positive, in - direction when negative. The axis does not rotate in shortcut direction. Incremental commands..... Rotates the commanded rotation amount.
Inch/mm changeover	Not provided Command to match the feedback unit.

17.4 PLC Interface

17.4.1 CNC Control Signal

M system : ○ **L system : ○**

Control commands to the CNC system are assigned from the PLC. Input signals with 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) Skip signals
When signals are input to the skip input interface, they are processed by interrupt processing. This enables functions requiring a high response speed to be implemented. (Maximum 4 points)

For further details, refer to the PLC Interface Manual.

17.4.2 CNC Status Signal

M system : O **L system : O**

The status signals are output from the CNC system. They can be utilized by referencing them from the PLC.

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
This signal is output when the controller is reset processing.
This signal will also be output when the reset & rewind command is input to the controller, 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.

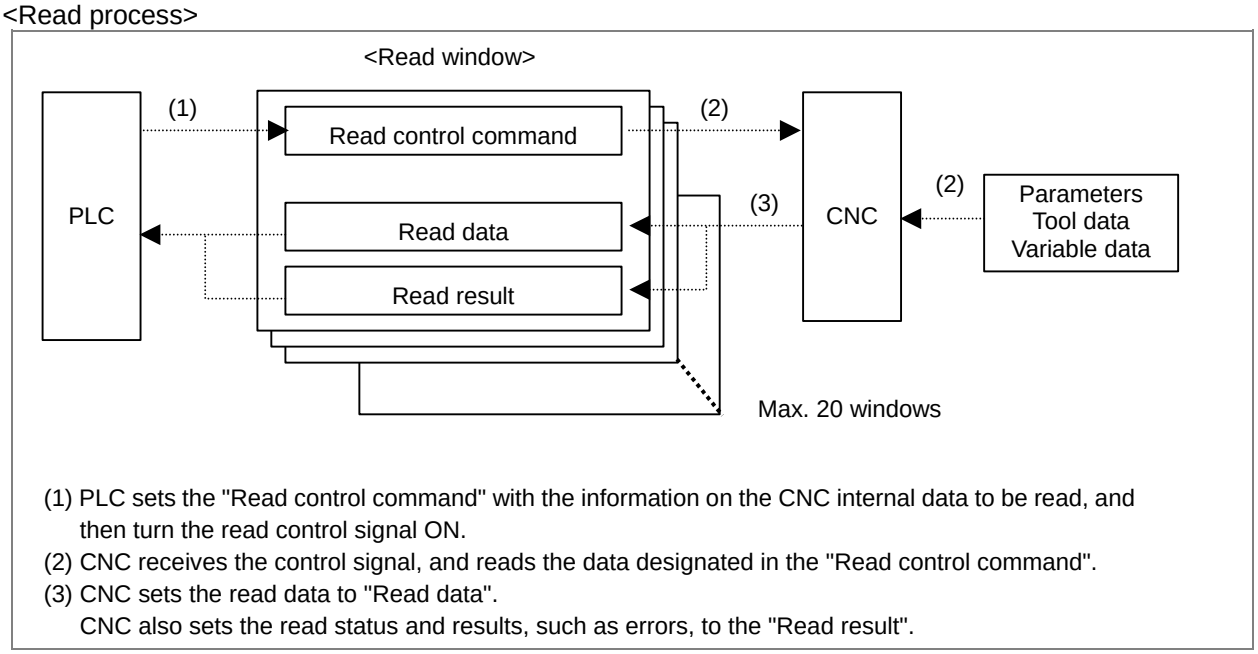
17.4.3 PLC Window

M system : Δ **L system : Δ**

PLC window is used to read/write the operation state, axis information, parameters and tool data through a cyclic trans mission area in CPU shared memory.

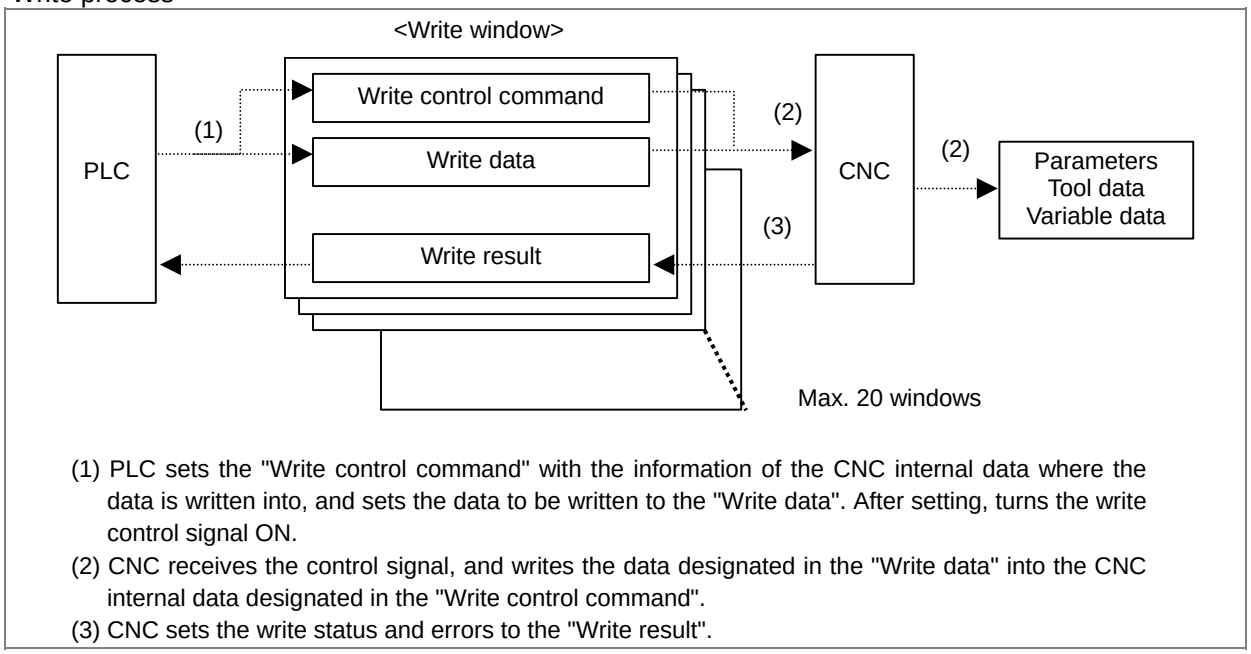
In the interface between CNC CPU for PLC window and PLC CPU, "Read control command", "Read data" and "Read result" are all called "Read window". "Write control command", "Write data" and "Write result" are all called "Write window".
These windows are used for the read and write operations. 40 units of windows, 20 units for each, are provided for "Read window" and "Write window".

Outlines of read and write processes are shown below.
<Read process>



(1) PLC sets the "Read control command" with the information on the CNC internal data to be read, and then turn the read control signal ON.
(2) CNC receives the control signal, and reads the data designated in the "Read control command".
(3) CNC sets the read data to "Read data".
CNC also sets the read status and results, such as errors, to the "Read result".

<Write process>



(1) PLC sets the "Write control command" with the information of the CNC internal data where the data is written into, and sets the data to be written to the "Write data". After setting, turns the write control signal ON.
(2) CNC receives the control signal, and writes the data designated in the "Write data" into the CNC internal data designated in the "Write control command".
(3) CNC sets the write status and errors to the "Write result".

17.4.4 External Search

M system : Δ **L system : Δ**

The program No., block No. and sequence No. to be automatically started in the memory, ladder can be searched from PLC. The currently searched details can be read.

17.6 External PLC Link

17.6.3 CC-Link (Master/Slave)

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.6.5 DeviceNet

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.6.6 FL-net

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.6.7 CC-Link/LT

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.6.101 As-i (Master)

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.6.102 MELSEC multiple CPU system

M system : Δ **L system : Δ**

Refer to each unit's manuals of MITSUBISHI Programmable Controller "MELSEC Q series" for the function and the performance.

17.7 Installing S/W for Machine Tools

17.7.3 EZSocket I/F

M system : ☆ **L system : ☆**

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.

17.7.5 Custom API Library

M system : ☆ **L system : ☆**

Reading/writing of each information within NC unit is possible by using custom API library.

17.8 Others

17.8.2 NC Monitoring Tool

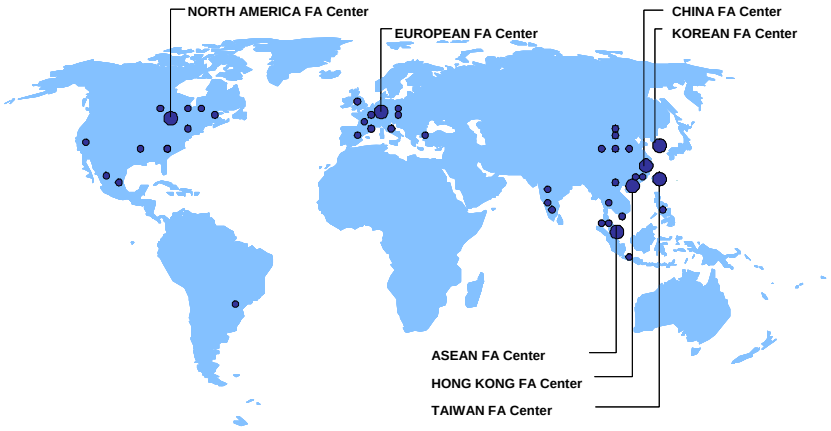
M system : Δ **L system : Δ**

NC monitoring tool is a PC compatible software tool that monitors information in NC unit connected with the Ethernet.

Revision History

Date of revision	Manual No.	Revision details
Dec. 2006	IB-1500259-A	First edition created.
Jan. 2007	IB-1500259-B	Mistakes were corrected.
May 2007	IB-1500259-C	The following sections are added. • 16. Drive System • 17. Machine Support Functions Other contents were added/revised/deleted according to specification.

Global service network



North America FA Center (MITSUBISHI ELECTRIC AUTOMATION INC.)

Illinois CNC Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL. 60061, U.S.A.
TEL: +1-847-478-2500 (Se FAX: +1-847-478-2650 (Se
California CNC Service Center
5665 PLAZA DRIVE, CYPRESS, CA. 90630, U.S.A.
TEL: +1-714-220-4796 FAX: +1-714-229-3818
Georgia CNC Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GA., 30097, U.S.A.
TEL: +1-678-258-4500 FAX: +1-678-258-4519
New Jersey CNC Service Center
200 COTTONTAIL LANE SOMERSET, NJ. 08873, U.S.A.
TEL: +1-732-560-4500 FAX: +1-732-560-4531
Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500 FAX: +1-269-673-4092
Ohio CNC Service Satellite
62 W. 500 S., ANDERSON, IN., 46013, U.S.A.
TEL: +1-847-478-2608 FAX: +1-847-478-2690
Texas CNC Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TX. 76051, U.S.A.
TEL: +1-817-251-7468 FAX: +1-817-416-1439
Canada CNC Service Center
4299 14TH AVENUE MARKHAM, ON. L3R OJ2, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935
Mexico CNC Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 FAX: +52-55-9171-7698
Monterrey CNC Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 FAX: +52-81-8365-4171
Brazil MITSUBISHI CNC Agent Service Center
(AUTOMOTION IND. COM. IMP. E EXP. LTDA.)
ACESSO JOSE SARTORELLI, KM 2.1 18550-000 BOITUVA – SP, BRAZIL
TEL: +55-15-3363-9900 FAX: +55-15-3363-9911

European FA Center (MITSUBISHI ELECTRIC EUROPE B.V.)

Germany CNC Service Center
GOTHAER STRASSE 8, 40880 RATINGEN, GERMANY
TEL: +49-2102-486-0 FAX:+49-2102486-591
South Germany CNC Service Center
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: +49-711-3270-010 FAX: +49-711-3270-0141
France CNC Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 FAX: +33-1-49-01-07-25
Lyon CNC Service Satellite

U.K CNC Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-282-846 FAX:-44-1707-278-992
Italy CNC Service Center
ZONA INDUSTRIALE VIA ARCHIMEDE 35 20041 AGRATE BRIANZA, MILANO ITALY
TEL: +39-039-60531-342 FAX: +39-039-6053-206
Spain CNC Service Satellite
CTRA. DE RUBI, 76-80 -APDO.420 08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 FAX:
Turkey MITSUBISHI CNC Agent Service Center
(GENEL TEKNIK SISTEMLER LTD. STI.)
DARULACEZE CAD. FAMAS IS MERKEZI A BLOCK NO.43 KAT2 80270 OKMEYDANI ISTANBUL, TURKEY
TEL: +90-212-320-1640 FAX: +90-212-320-1649
Poland MITSUBISHI CNC Agent Service Center (MPL Technology Sp. z. o. o)
UL SLICZNA 34, 31-444 KRAKOW, POLAND
TEL: +48-12-632-28-85 FAX:
Wroclaw MITSUBISHI CNC Agent Service Satellite (MPL Technology Sp. z. o. o)
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 FAX: +48-71-333-77-53
Czech MITSUBISHI CNC Agent Service Center
(AUTOCONT CONTROL SYSTEM S.R.O.)
NEMOCNICNI 12, 702 00 OSTRAVA 2 CZECH REPUBLIC
TEL: +420-596-152-426 FAX: +420-596-152-112

ASEAN FA Center (MITSUBISHI ELECTRIC ASIA PTE. LTD.)

Singapore CNC Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 FAX: +65-6476-7439
Thailand MITSUBISHI CNC Agent Service Center (F. A. TECH CO., LTD)
898/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 12,14 RAMA III RD BANGPONGPANG, YANNAWA, BANGKOK 10120. THAILAND
TEL: +66-2-682-6522 FAX: +66-2-682-6020
Malaysia MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN MALAYSIA
TEL: +60-3-5631-7605 FAX: +60-3-5631-7636
JOHOR MITSUBISHI CNC Agent Service Satellite
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
NO. 16, JALAN SHAHBANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 FAX: +60-7-557-3404
Indonesia MITSUBISHI CNC Agent Service Center
(PT. AUTOTEKNINDO SUMBER MAKMUR)
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 FAX: +62-21-3917-164
India MITSUBISHI CNC Agent Service Center (MESSUNG SALES & SERVICES PVT. LTD.)
B-36FF, PAVANA INDUSTRIAL PREMISES M.I.D.C., BHOASRI PUNE 411026, INDIA
TEL: +91-20-2711-9484 FAX: +91-20-2712-8115
BANGALORE MITSUBISHI CNC Agent Service Satellite
(MESSUNG SALES & SERVICES PVT. LTD.)
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 FAX: +91-80-532-0480
Delhi MITSUBISHI CNC Agent Parts Center (MESSUNG SALES & SERVICES PVT. LTD.)
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32ND MILESTONE GURGAON 122001, INDIA
TEL: +91-98-1024-8895 FAX:
Philippines MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM CORPORATION)
UNIT No.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 FAX: +63-2-807-2417
Vietnam MITSUBISHI CNC Agent Service Center (SA GIANG TECHNO CO., LTD)
47-49 HOANG SA ST. DAKAO WARD, DIST.1 HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 FAX: +84-8-910-2593

China FA Center (MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD.)

China CNC Service Center
2/F., BLOCK 5 BLDG.AUTOMATION INSTRUMENTATION PLAZA, 103 CAOBAO RD. SHANGHAI 200233, CHINA
TEL: +86-21-6120-0808 FAX: +86-21-6494-0178
Shenyang CNC Service Center
TEL: +86-24-2397-0184 FAX: +86-24-2397-0185
Beijing CNC Service Satellite
9/F, OFFICE TOWER1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE, DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 FAX: +86-10-6518-8030
China MITSUBISHI CNC Agent Service Center
(BEIJING JIAYOU HIGHTECH TECHNOLOGY DEVELOPMENT CO.)
RM 709, HIGH TECHNOLOGY BUILDING NO.229 NORTH SI HUAN ZHONG ROAD, HAIDIAN DISTRICT , BEIJING 100083, CHINA
TEL: +86-10-8288-3030 FAX: +86-10-6518-8030
Tianjin CNC Service Satellite
RM909, TAIHONG TOWER, NO220 SHIZILIN STREET, HEBEI DISTRICT, TIANJIN, CHINA 300143
TEL: -86-22-2653-9090 FAX: +86-22-2635-9050
Shenzhen CNC Service Satellite
RM02, UNIT A, 13/F, TIANAN NATIONAL TOWER, RENMING SOUTH ROAD, SHENZHEN, CHINA 518005
TEL: +86-755-2515-6691 FAX: +86-755-8218-4776
Changchun Service Satellite
TEL: +86-431-50214546 FAX: +86-431-5021690
Hong Kong CNC Service Center
UNIT A, 25/F RYODEN INDUSTRIAL CENTRE, 26-38 TA CHUEN PING STREET, KWAI CHUNG, NEW TERRITORIES, HONG KONG
TEL: +852-2619-8588 FAX: +852-2784-1323

Taiwan FA Center (MITSUBISHI ELECTRIC TAIWAN CO., LTD.)

Taichung CNC Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689
Taipei CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689
Tainan CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Korean FA Center (MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD.)

Korea CNC Service Center
1480-6, GAYANG-DONG, GANGSEO-GU SEOUL 157-200, KOREA
TEL: +82-2-3660-9631 FAX: +82-2-3664-8668

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	C70
MODEL CODE	100—009
Manual No.	IB-1500259 (ENG)

IB(NA)1500259(ENG)-C(0705)MEE