FANUC

GE Fanuc Automation

Computer Numerical Control Products

Series 0i-Model C
Series 0i-Mate Model C

Descriptions

GFZ-64112EN/01 August 2004

GFL-001

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

Contents

1 DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Also, supplementary information is described as a Note. Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

WARNING

Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

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

NOTE

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

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

2 GENERAL WARNINGS AND CAUTIONS

WARNING

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2. Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4. When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5. The parameters for the CNC and PMC are factory–set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7. The operator’s manual and programming manual supplied with a CNC unit provide an overall description of the machine’s functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

WARNING

8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

CAUTION

1. Do not remove the internal parts, including the ATA card and compact flash card, from within the CNC.

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied operator's manual and programming manual carefully such that you are fully familiar with their contents.

WARNING

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.
Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming.
Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal–direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely.
Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. Constant surface speed control

When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

WARNING

6. Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Tool post interference check

A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user.
After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9. Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10. Torque limit skip

Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

11. Programmable mirror image

Note that programmed operations vary considerably when a programmable mirror image is enabled.

12. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied operator's manual and programming manual carefully, such that you are fully familiar with their contents.

WARNING

1. Manual operation

When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2. Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed. An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Manual numeric command

When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4. Manual handle feed

In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

5. Disabled override

If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6. Origin/preset operation

Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

WARNING

7. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8. Software operator's panel and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9. Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

10. Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

11. Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

12. Cutter and tool nose radius compensation in MDI mode

Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

13. Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the maintenance section of the operator's manual or programming manual for details of the battery replacement procedure.

WARNING

2. Absolute pulse coder battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

Refer to the maintenance section of the operator's manual or programming manual for details of the battery replacement procedure.

WARNING

3. Fuse replacement

For some units, the chapter covering daily maintenance in the operator's manual or programming manual describes the fuse replacement procedure.
Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high–voltage circuit presents an extremely dangerous electric shock hazard.

Table of Contents

I. GENERAL

1 GENERAL

This manual describes the following models and may use the following abbreviations.

Model name	Abbreviation	
FANUC Series 0*i*–TC	0*i*–TC	Series 0*i*
FANUC Series 0*i*–MC	0*i*–MC	
FANUC Series 0*i*–PC	0*i*–PC	
FANUC Series 0*i* Mate–TC	0*i* Mate–TC	Series 0*i* Mate
FANUC Series 0*i* Mate–MC	0*i* Mate–MC	

For ease of understanding, the models may be categorized as follows:
T series: 0*i*–TC, 0*i* Mate–TC
M series: 0*i*–MC, 0*i* Mate–MC

Related manuals of Series 0*i*–C/0*i* Mate–C

The following table lists the manuals related to Series 0*i*–C, Series 0*i* Mate–C.
This manual is indicated by an asterisk(*).

Manual name	Specification number	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C DESCRIPTIONS	B–64112EN	*
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C CONNECTION MANUAL (HARDWARE)	B–64113EN	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C CONNECTION MANUAL (FUNCTION)	B–64113EN–1	
FANUC Series 0*i*–PC CONNECTION MANUAL (FUNCTION)	B–64153EN	
FANUC Series 0*i*–TC OPERATOR'S MANUAL	B–64114EN	
FANUC Series 0*i*–MC OPERATOR'S MANUAL	B–64124EN	
FANUC Series 0*i* Mate–TC OPERATOR'S MANUAL	B–64134EN	
FANUC Series 0*i* Mate–MC OPERATOR'S MANUAL	B–64144EN	
FANUC Series 0*i*–PC OPERATOR'S MANUAL	B–64154EN	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C MAINTENANCE MANUAL	B–64115EN	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C PARAMETER MANUAL	B–64120EN	
FANUC Series 0*i*–PC PARAMETER MANUAL	B–64160EN	
PROGRAMMING MANUAL		
Macro Compiler/Macro Executor PROGRAMMING MANUAL	B–61803E–1	
FANUC MACRO COMPILER (For Personal Computer) PROGRAMMING MANUAL	B–66102E	
PMC		
PMC Ladder Language PROGRAMMING MANUAL	B–61863E	
PMC C Language PROGRAMMING MANUA	B–61863E–1	
Network		
PROFIBUS–DP Board OPERATOR'S MANUAL	B–62924EN	
Ethernet Board/DATA SERVER Board OPERATOR'S MANUAL	B–63354EN	
FAST Ethernet Board/FAST DATA SERVER OPERATOR'S MANUAL	B–63644EN	
DeviceNet Board OPERATOR'S MANUAL	B–63404EN	

Related manuals of SERVO MOTOR α*is*/α*i*/β*is* series

The following table lists the manuals related to SERVO MOTOR α*i*s/α*i*/β*i*s series

Manual name	Specification number
FANUC AC SERVO MOTOR α*i*s/α*i* series DESCRIPTIONS	B–65262EN
FANUC AC SERVO MOTOR β*i*s series DESCRIPTIONS	B–65302EN
FANUC AC SERVO MOTOR α*i*s/α*i*/β*i*s series PARAMETER MANUAL	B–65270EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B–65272EN
FANUC AC SPINDLE MOTOR β*i*s series DESCRIPTIONS	B–65312EN
FANUC AC SPINDLE MOTOR α*i*/β*i* series PARAMETER MANUAL	B–65270EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B–65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B–65322EN
FANUC AC SERVO MOTOR α*i*s/α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B–65285EN
FANUC AC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B–65325EN

2 LIST OF SPECIFICATIONS

○ : Standard ○A : Standard of Package A ○B : Standard of Package B ● : Standard option ●A : Standard option of Package A
●B : Standard option of Package B ☆ : Option ☆A : Option of Package A ✳ : Function included in another option — : Not available
Note) Some combinations of these options are restricted.
In case of CNC Display with Personal Computer function, using CNC Screen Display Function is a premise.

Item		Specifications	Series 0*i*			Series 0*i* Mate	
			MC	TC	PC	MC	TC
Controlled axis							
Machine controlled axes	Controlled path	1 path	○	○	○	○	○
	Controlled axis (Including Cs axes)	4 axes	○	☆	○	—	—
		3 axes	—	○(Cs)/☆	—	○	○(Cs)
		2 axes	—	○	—	—	○
	Simultaneously controlled axes	Max. 4 axes	○	○	○	—	—
		Max. 3 axes	—	—	—	○	○(Cs)
	Axis control by PMC	Max. simultaneous 4 axes (Not available on Cs axes)	○	○	○	—	—
	Cs contouring control	1 axis	○	○	—	—	○
Axis name		Basic three axes are X, Y and Z, additional axes are optional from U, V, W, A, B and C.	○	—	—	○	—
		In case of G code system A, basic 2 axes are X and Z, additional axes are optional from Y, A, B and C.	—	○	—	—	○
		In case of G code system B/C, basic 2 axes are X and Z, additional axes are optional from Y, U, V, W, A, B and C.	—	○	—	—	○
		Basic two axes are X and Y, additional axes are optional from Z, U, V, W, A, B, C and T.	—	—	○	—	—
Spindle		2 units	○	○	—	—	—
		1 unit	—	—	—	○	○
Simple synchronous control			○	○	○	—	—
Angular axis control			☆	☆	—	—	—
Arbitrary angular axis control			—	☆	—	—	—
Tandem control			○	○	○	—	—
Tandem Disturbance Elimination Control			☆	☆	☆	—	—
Least input increment		0.001mm, 0.001deg, 0.0001inch	○	○	○	○	○
Increment system 1/10		0.0001mm, 0.0001deg, 0.00001inch	○	○	—	○	○
Flexible feed gear		Optional DMR	○	○	○	○	○
Fine Acc & Dec control			○	○	○	○	○
Servo HRV control		HRV3	○	○	○	○	○
Inch/metric conversion			○	○	○	○	○
Interlock		All axes/each axis/each direction/block start/cutting block start	○	○	○	○	○
Machine lock		All axes/each axis	○	○	○	○	○
Emergency stop			○	○	○	○	○
Overtravel			○	○	○	○	○
Stored stroke check 1			○	○	○	○	○
Stroke limit external setting			○	—	○	○	—
Stored stroke check 2			○	—	○	○	—
Stored stroke check 2, 3			—	○	—	—	○
Stroke limit check before move			○	○	—	○	○
Chuck and tail stock barrier			—	○	—	—	○
Mirror image		Each axis	○	○	○	○	○
Follow–up			○	○	○	○	○
Servo off			○	○	○	○	○
Chamfering on/off			—	○	—	—	○
Backlash compensation			○	○	○	○	○

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC
Backlash compensation for each rapid traverse and cutting feed		○	○	○	○	○
Stored pitch error compensation		○	○	○	○	○
Bi–directional pitch error compensation		☆	☆	☆	—	—
Position switch		○	○	○	○	○
Unexpected disturbance torque detection function		○	○	○	—	—
Control axis detach		○	○	—	○	○
Linear scale I/F with absolute address reference mark		☆	☆	☆	—	—
Linear scale with absolute address reference mark expansion		☆	☆	—	—	—
Temporary absolute coordinate setting		☆	☆	—	—	—

Operation

Item	Specifications	MC	TC	PC	MC	TC
Automatic operation (memory)		○	○	○	○	○
DNC operation	Reader/puncher interface is required.	○	○	○	○	○
DNC operation with Memory Card	PCMCIA Card Attachment is required.	○	○	○	○	○
MDI operation		○	○	○	○	○
Schedule function		○	○	○	○	○
Program number search		○	○	○	○	○
Sequence number search		○	○	○	○	○
Sequence number comparison and stop		○	○	○	○	○
Program restart		○	○	—	○	○
Manual intervention and return		○	○	—	○	○
Retraction for rigid tapping		○	—	—	○	—
Buffer register		○	○	○	○	○
Dry run		○	○	○	○	○
Single block		○	○	○	○	○
JOG feed		○	○	○	○	○
Manual reference position return		○	○	○	○	○
Reference position setting without DOG		○	○	○	○	○
Reference position setting with mechanical stopper		○	○	—	○	○
Reference position shift		○	—	○	—	—
Manual handle feed	MC, PC : Max. 3 units TC : Max. 2 units	○	○	○	○	○
Manual handle feed rate	×1, ×10, ×m, ×n m: 0 to 127, n: 0 to 1000	○	○	○	○	○
Manual handle interruption		○	○	○	○	○
Manual handle I/F for I/O Link β		☆	☆	☆	☆	☆
Incremental feed	×1, ×10, ×100, ×1000	○	○	○	○	○
Jog and handle simultaneous mode		○	○	○	○	○

Interpolation functions

Item	Specifications	MC	TC	PC	MC	TC
Positioning	G00 (Linear interpolation type positioning is possible)	○	○	○	○	○
Single direction positioning	G60	○	—	—	○	—
Exact stop mode	G61	○	—	○	○	—
Exact stop	G09	○	—	○	○	—
Linear interpolation		○	○	○	○	○
Circular interpolation	Multi–quadrant is possible	○	○	○	○	○
Dwell	Dwell in seconds and dwell in revolution (In case of dwell in revolution for M series, threading, synchronous cutting function is required.)	○	○	○	○	○
Polar coordinate interpolation		—	○	—	—	○
Cylindrical interpolation		○	○	—	—	○
Helical interpolation	Circular interpolation plus max. 2 axes linear interpolation	○	☆	○	○	—
Threading, synchronous cutting		○	○	—	○	○
Multiple threading		—	○	—	—	○
Threading retract		—	○	—	—	○
Continuous threading		—	○	—	—	○

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC
Interpolation functions						
Variable lead threading		—	○	—	—	—
Polygon turning		—	○	—	—	—
Skip	G31	○	○	○	○	○
High–speed skip	Input signal is 1 point	○	○	○	○	○
Multi–step skip	For grinding machine	☆	☆	—	☆	☆
Torque limit skip		—	○	—	—	○
Reference position return	G28	○	○	○	○	○
Reference position return check	G27	○	○	—	○	○
2nd reference position return		○	○	—	○	○
3rd/4th reference position return		○	○	—	○	○
Normal direction control		○	—	○	—	—
Continuous dressing	For grinding machine	☆	—	—	—	—
Infeed control	For grinding machine	☆	—	—	—	—
Index table indexing		○	—	—	—	—
General purpose retract		○	○	—	○	○
Feed function						
Rapid traverse rate	Max. 240m/min (1μm)	○	○	○	○	○
	Max. 100m/min (0.1μm)	○	○	○	—	—
Rapid traverse override	Fo, 25, 50, 100% PC : 25, 50, 75, 100%	○	○	○	○	○
Feed per minute		○	○	○	○	○
Feed per revolution		○	○	—	○	○
Tangential speed constant control		○	○	○	○	○
Cutting feedrate clamp		○	○	○	○	○
Automatic acceleration/deceleration	Rapid traverse: linear Cutting feed: exponential	○	○	○	○	○
Rapid traverse bell–shaped acceleration/deceleration		○	○	○	○	○
Linear acceleration/deceleration after cutting feed interpolation		○	○	○	○	○
Bell–type acceleration/deceleration after cutting feed interpolation		☆	—	—	—	—
Bell–type acceleration/deceleration before look ahead interpolation		☆A	—	—	—	—
Feedrate override	0 to 254%	○	○	○	○	○
One–digit F code feed		○	—	—	○	—
Inverse time feed		○	—	—	—	—
Jog override	0 to 655.34%	○	○	○	○	○
Override cancel		○	○	○	○	○
Manual per revolution feed		—	○	—	—	○
Error detection		—	○	—	—	○
Cutting mode		○	—	○	○	—
Tapping mode		○	—	—	○	—
Rapid traverse block overlap		○	○	—	○	○
External deceleration		○	○	○	○	○
Advanced preview control		—	☆	—	—	—
AI advanced preview control		○	—	—	○	—
AI contour control		☆A	—	—	—	—
Rigid tapping bell–shaped acceleration/deceleration		☆	—	—	—	—
Program input						
Tape code	EIA RS244/ISO840	○	○	○	○	○
Label skip		○	○	○	○	○
Parity check	Horizontal and vertical parity	○	○	○	○	○
Control in/out		○	○	○	○	○
Optional block skip	9	○	○	○	○	○
Max. programmable dimension	±8–digit	○	○	○	○	○
Program number	O4–digit	○	○	○	○	○

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC
Program input						
External memory and sub program calling function		○	○	○	○	○
Sequence number	N5–digit	○	○	○	○	○
Absolute/incremental programming	Combined use in the same block	○	○	○	○	○
Decimal point programming/pocket calculator type decimal point programming		○	○	○	○	○
Input unit 10 time multiply		○	○	○	○	○
Diameter/radius programming (X axis)		—	○	—	—	○
Plane selection	G17, G18, G19	○	○	○	○	○
Rotary axis designation		○	○	○	—	○
Rotary axis roll–over		○	○	○	—	○
Polar coordinate command		○	—	—	○	—
Coordinate system setting		○	○	○	○	○
Automatic coordinate system setting		○	○	○	○	○
Coordinate system shift		—	○	—	—	○
Direct input of coordinate system shift		—	○	—	—	○
Workpiece coordinate system	G52 to G59	○	○	○	○	○
Workpiece coordinate system preset		○	○	—	○	○
Addition of workpiece coordinate system pair	48 pairs	○	—	—	○	—
Direct input of workpiece origin offset value measured		○	○	—	○	○
Manual absolute on and off		○	○	○(on)	○	○
Direct drawing dimension programming		—	○	—	—	○
G code system	A/B/C	—	○	—	—	○
Chamfering/corner R		—	○	—	—	○
Optional chamfering/corner R		○	—	—	○	—
Programmable data input	G10	○	○	○	○	○
Sub program call	4 folds nested	○	○	○	○	○
Custom macro B		○	○	○	○	○
Addition of custom macro common variables	#100 to #199, #500 to #999	○	○	○	○	○
Pattern data input		○	○	—	○	○
Interruption type custom macro		○	○	○	○	○
Canned cycles		—	○	—	—	○
Multiple repetitive cycle		—	○	—	—	○
Multiple repetitive cycle II	Pocket profile	—	○	—	—	—
Canned cycles for drilling		○	○	—	○	○
Small–hole peck drilling cycle		○	—	—	○	—
Canned cycles for grinding	For grinding machine	☆	☆	—	—	—
Circular interpolation by R programming		○	○	○	○	○
Mirror image for double turret		—	○	—	—	—
Automatic corner override		○	—	○	○	—
Automatic corner deceleration		○	—	○	○	—
Feedrate clamp based on arc radius		○	—	○	○	—
Scaling		○	—	○	○	—
Coordinate system rotation		○	—	○	○	—
Programmable mirror image		○	—	—	○	—
Tape format for FANUC Series 10/11		○	○	—	○	○
Conversational programming with graphic function	Display unit with graphic is required.	✻	✻	✻	✻	✻
Macro executor		☆(512KB/2MB/4MB)			☆(512KB)	
MANUAL GUIDE 0*i*						
ISO code part programming		☆	☆	—	☆	☆
Process control information		☆	☆	—	☆	☆
G–code assistance		☆	☆	—	☆	☆
M–code assistance		☆	☆	—	☆	☆

Item		Specifications	Series 0*i* MC	Series 0*i* TC	Series 0*i* PC	Series 0*i* Mate MC	Series 0*i* Mate TC

MANUAL GUIDE 0*i*

Item		Specifications	MC	TC	PC	Mate MC	Mate TC
Contour programming							
	Max. figure block number		40	40	—	40	40
	Start point		☆	☆	—	☆	☆
	Offset setting	G40/G41/G42, No output	☆	☆	—	☆	☆
	Figure type	Line, Arc CW, Arc CCW, Corner R, Chamfering	☆	☆	—	☆	☆
	Figure modification	Insert, Delete, Re–calculation	☆	☆	—	☆	☆
	Check contour figure	Graphical display of entered figures	☆	☆	—	☆	☆
	Convert to ISO code prog.	G01/G02/G03 form	☆	☆	—	☆	☆
	Calculation of inputting data	+, –, ×, /, Sin, Cos, Tan, Square root	☆	☆	—	☆	☆
	Data value unit	IS–B, IS–C	☆	☆	—	☆	☆
Contour programming auxiliary calculation							
	A point by polar coordinate		☆	☆	—	☆	☆
	A point by angle and dist.		☆	☆	—	☆	☆
	A point by rotating a point		☆	☆	—	☆	☆
	Neighbor point of a line		☆	☆	—	☆	☆
	Cross point between 2 lines		☆	☆	—	☆	☆
	Cross point line and arc		☆	☆	—	☆	☆
	Cross point between 2 arcs		☆	☆	—	☆	☆
	Angle of 2 points line		☆	☆	—	☆	☆
	Angle of rectangular line		☆	☆	—	☆	☆
	Arc with 1 point and center		☆	☆	—	☆	☆
	Arc with 2 points and radius		☆	☆	—	☆	☆
	Arc with 3 points		☆	☆	—	☆	☆
Advanced canned cycle for milling							
	Hole machining	Center, Drilling, Tapping, Reaming, Boring, Fine boring, Back boring	☆	—	—	☆	—
	Hole pattern	Points, Line(EQ), Line(NE), Grid, Square, Circle, Arc(EQ), Arc(NE)	☆	—	—	☆	—
	Facing (Rough/Finish)	Square, Circle (fixed pattern)	☆	—	—	☆	—
	Pocketing (Rough/Finish)	Square, Circle, Track (fixed pattern)	☆	—	—	☆	—
	Pocketing (Drilling)	Square, Circle, Track (fixed pattern)	☆	—	—	☆	—
	Grooving (Rough/Finish)	Radial line (fixed pattern)	☆	—	—	☆	—
	Grooving (Chamfer/Drilling)	Radial line (fixed pattern)	☆	—	—	☆	—
Advanced canned cycle for lathe							
	Lathe drilling	Center, Drilling, Tapping, Reaming, Boring	—	☆	—	—	☆
	Stock removal in turning	Outer, Inner, Face	—	☆	—	—	☆
	Finishing	Outer, Inner, Face	—	☆	—	—	☆
	Threading	Outer, Inner	—	☆	—	—	☆
	Threading type	General, Metric, Unified, PT, PF	—	☆	—	—	☆
	Grooving	Outer, Face	—	☆	—	—	☆
	Grooving type	Normal, Trapezoidal	—	☆	—	—	☆

Item		Specifications	Series 0*i*			Series 0*i* Mate	
			MC	TC	PC	MC	TC

MANUAL GUIDE 0*i*

Item		Specifications	MC	TC	PC	MC	TC
Canned grinding cycle for surface grinding machine							
	Plunge grinding cycle	G75 (under development)	☆	—	—	☆	—
	Direct constant–dimension plunge grinding cycle	G77 (under development)	☆	—	—	☆	—
	Continuous–feed surfae grinding cycle	G78 (under development)	☆	—	—	☆	—
	Intermittent–feed surface grinding cycle	G79 (under development)	☆	—	—	☆	—
Canned grinding cycle for cylindrical grining machine							
	Traverse grinding cycle	G71	—	☆	—	—	☆
	Traverse direct fixed–dimension grinding cycle	G72	—	☆	—	—	☆
	Oscillation grinding cycle	G73	—	☆	—	—	☆
	Oscillation direct fixed dimension grinding cycle	G74	—	☆	—	—	☆

Auxiliary/Spindle speed function

Item	Specifications	MC	TC	PC	MC	TC
Auxiliary function	M8–digit	○	○	○	○	○
2nd auxiliary function	B8–digit	○	○	○	○	○
Auxiliary function lock		○	○	○	○	○
High–speed M/S/T/B interface		○	○	○	○	○
Multiple command of auxiliary function	3	○	○	○	○	○
Spindle speed function	S5–digit, binary output	○	○	—	○	○
Spindle serial output	S5–digit, serial output	●	●	—	●	●
Spindle analog output	S5–digit, analog output	●	●	—	●	●
Constant surface speed control		○	○	—	○	○
Spindle override	0 to 254%	○	○	—	○	○
Actual spindle speed output		—	○	—	—	○
Spindle speed fluctuation detection		—	○	—	—	—
1st spindle orientation		○	○	—	○	○
1st spindle output switching function		○	○	—	○	○
2nd spindle orientation		○	○	—	—	—
2nd spindle output switching function		○	○	—	—	—
Spindle synchronous control		○	○	—	—	—
Multi spindle control		—	○	—	—	—
Spindle positioning		—	○	—	—	○
Rigid tapping		○	○	—	○	○

Tool function/Tool compensation

Item	Specifications	MC	TC	PC	MC	TC
Tool function	T7 + 1/T6 + 2 digits	—	○	—	—	○
	T8 digits	○	—	○	○	—
Tool offset pairs	±6 digits 32	—	—	○	—	—
	±6 digits 64	—	○	—	—	○
	±6 digits 400	○	—	—	○	—
Tool offset memory C	Distinction between geometry and wear, or between cutter and tool length compensation.	○	—	—	○	—
Tool length compensation		○	—	—	○	—
Tool offset		○	○	—	○	○
Y–axis offset		—	○	—	—	—
Cutter compensation C		○	—	○	○	—
Tool nose radius compensation		—	○	—	—	○
Tool geometry/wear compensation		—	○	—	—	○
Tool life management		○	○	—	○	○
Extended tool life management		○	—	—	○	—

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC

Tool function/Tool compensation

Item	Specifications	MC	TC	PC	MC	TC
Tool offset value counter input		—	○	—	—	○
Tool length measurement		○	—	—	○	—
Automatic tool length measurement		○	—	—	○	—
Automatic tool offset		—	○	—	—	—
Direct input of tool offset value measured		—	○	—	—	○
Direct input of tool offset value measured B		—	○	—	—	○

Punch press function

Item	Specifications	MC	TC	PC	MC	TC
1 cycle press		—	—	○	—	—
Manual press	1 cycle/continuity	—	—	○	—	—
Positioning & press off	G70	—	—	○	—	—
Setting for press start signal		—	—	○	—	—
Press start lock		—	—	○	—	—
Press start wait		—	—	○	—	—
Nibbling	G68, G69, M code	—	—	○	—	—
Changeable nibbling mode 2 steps	By nibbling pitch	—	—	○	—	—
External motion function	EF output	—	—	○	—	—
Stroke limit check before move	Only for G00	—	—	○	—	—
Safety zone check		—	—	○	—	—
Changeable Rapid traverse rate/time constant according to positioning distance	X, Y axis: 7 steps	—	—	○	—	—
Positioning time constant control	X, Y axis: 2 steps	—	—	○	—	—
Changeable position loop gain	X, Y axis: 7 steps(Only for positioning)	—	—	○	—	—
Advanced preview control		—	—	○	—	—
G code system	A/B	—	—	○	—	—
Pattern function		—	—	○	—	—
Pattern base point command	G72	—	—	○	—	—
Memory and call by A/B macro	5	—	—	○	—	—
U/V/W macro		—	—	○	—	—
Multi–piece machining		—	—	○	—	—
Command for restarting multi–piece machining		—	—	○	—	—
Repositioning	G75, M code	—	—	○	—	—
Bending compensation	G38, G39	—	—	○	—	—
S function	S5–digit, binary output	—	—	○	—	—
Number of registerble tool number	136	—	—	○	—	—
T axis control		—	—	○	—	—
Tool offset		—	—	○	—	—
T–command neglect		—	—	○	—	—
Tool life management		—	—	○	—	—
C axis control		—	—	○	—	—
C axis backlash compensation for each index		—	—	○	—	—
C axis offset		—	—	○	—	—

Editing operation

Item	Specifications	MC	TC	PC	MC	TC
Part program storage length *2	320m (128Kbyte)	○B	○B	—	—	—
	640m (256Kbyte)	○A	○A	○A	○	○
Number of registerable programs	400	○	○	○	○	○
Part program editing		○	○	○	○	○
Program protect		○	○	○	○	○
Background editing		○	○	○	○	○
Extended part program editing		○	○	○	○	○
Playback		○	○	—	○	○
Password function		○	○	○	○	○

Setting and display

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC

Setting and display

Item	Specifications	Series 0*i* MC	Series 0*i* TC	Series 0*i* PC	Series 0*i* Mate MC	Series 0*i* Mate TC
Status display		○	○	○	○	○
Clock function		○	○	○	○	○
Current position display		○	○	○	○	○
Program display	Program name 31 characters	○	○	○	○	○
Parameter setting and display		○	○	○	○	○
Parameter set supporting screen		○	○	—	○	○
Machining condition selecting function		☆	☆	—	☆	—
Self–diagnosis function		○	○	○	○	○
Alarm display		○	○	○	○	○
Alarm history display		○	○	○	○	○
Operator message history display		○	○	○	○	○
Operation history display		○	○	○	○	○
Help function		○	○	○	○	○
Remote diagnostic	Machine remote diagnosis package and Ethernet are required. (Function:Reading CNC/PMC status, etc.)	＊	＊	＊	—	—
Run hour and parts count display		○	○	○	○	○
Actual cutting feedrate display		○	○	○	○	○
Display of spindle speed and T code at all screens		＊	○	＊	＊	○
Directory display of floppy cassette		○	○	○	○	○
Directory display and punch for each group		○	○	○	○	○
Graphic function		＊	＊	☆	＊	＊
Dynamic graphic display		☆	☆	—	☆	☆
Servo setting screen		○	○	○	○	○
Spindle setting screen	Serial spindle only	＊	＊	—	＊	＊
Servo waveform display	Display unit with graphic is required.	＊	＊	＊	＊	＊
Display of hardware and software configuration		○	○	○	○	○
Periodic maintenance screen		○	○	○	○	○
Maintenance information screen		○	○	○	○	○
Machine alarm diagnosis		☆	☆	—	☆	☆
Trouble diagnosis		○	○	○	○	○
Software operator’s panel		○	○	○	○	○
Software operator’s panel general purpose switch		○	○	○	○	○
Multi–language display	English	○	○	○	○	○
	Japanese (Chinese character)	○	○	○	○	○
	German/French	○	○	○	○	○
	Italian	○	○	○	○	○
	Chinese (Traditional Chinese)	○	○	—	○	○
	Chinese (Simplified Chinese) Display unit with graphic is required.	＊	＊	＊	＊	＊
	Spanish	○	○	○	○	○
	Korean	○	○	—	○	○
	Portuguese	○	○	○	○	○
	Polish	○	○	—	○	○
	Hungarian	○	○	—	○	○
	Swedish	○	○	—	○	○
	Dutch	○	—	—	○	—
	Czech	○	○	—	○	○
Data protection key	4 types	○	○	○	○	○
Erase CRT screen display		○	○	○	○	○
CNC screen display		＊	＊	＊	—	—
Dual screen of CNC screen display function		☆	☆	☆	—	—

Item	Specifications	Series 0*i* MC	Series 0*i* TC	Series 0*i* PC	Series 0*i* Mate MC	Series 0*i* Mate TC

Data input/output

Item	Specifications	MC	TC	PC	MC	TC
Reader/puncher interface	Reader/puncher (Ch.1) interface	○	○	○	○	○
	Reader/puncher (Ch.2) interface	○	○	○	○	○
Data server		☆	☆	☆	—	—
External I/O device control		○	○	○	○	○
DNC2 control	Uploading/downloading a part program, Reading/writing CNC data, Transfer of PMC data, Memory operation control, etc.	☆	☆	☆	—	—
External tool offset		○	○	○	○	○
External message		○	○	○	○	○
External machine zero point shift		○	○	○	○	○
External data input	Including above 3 items	○	○	○	○	○
External key input		○	○	○	○	○
External program input		○	○	○	○	○
External workpiece number search	9999	○	○	○	○	○
Expanded external workpiece number search		○	○	○	○	○
External program number search	1 to 9999	○	○	○	○	○
Memory card input/output		○	○	○	○	○
Power Mate CNC manager		○	○	○	○	○
External touch panel I/F	SNP protocol	☆	☆	☆	☆	☆

Interface function

Item	Specifications	MC	TC	PC	MC	TC
FAST Ethernet	Ethernet board is required.	☆	☆	☆	—	—
PROFIBUS–DP	Master/Slave	☆	☆	☆	—	—
DeviceNet	Master/Slave	☆	☆	☆	—	—
FL–net		—	—	—	—	—

Others

Item		Specifications	MC	TC	PC	MC	TC
Status output signal		NC ready, servo ready, automatic operation, automatic operation start lamp, feed hold, reset, NC alarm, distribution end, rewinding, inch input, cutting, inposition, thread cutting, tapping, etc.	○	○	○	○	○
Control unit incorporated display unit *1	7.2″ monochrome LCD/MDI Horizontal type	1 slot 400(W)×200(H)×70(D)mm	●	●	●	●	●
		2 slots 400(W)×200(H)×120(D)mm	●	●	●	—	—
	8.4″ color LCD/MDI Horizontal type	1 slot 400(W)×200(H)×70(D)mm	●	●	●	—	—
		2 slots 400(W)×200(H)×120(D)mm	●	●	●	—	—
	7.2″ monochrome LCD/MDI Virtical type	1 slot 260(W)×300(H)×70(D)mm	●	●	●	●	●
		2 slots 260(W)×300(H)×120(D)mm	●	●	●	—	—
	8.4″ color LCD/MDI Virtical type	1 slot 260(W)×300(H)×70(D)mm	●	●	●	—	—
		2 slots 260(W)×300(H)×120(D)mm	●	●	●	—	—
PMC SYSTEM	PMC–SA1	Basic instruction: 5μ sec/step Max. step number ladder: 5000	○B	○B	—	○	○
	PMC–SB7	Basic instruction: 0.033μ sec/step Max. step number ladder: 24000	○A	○A	○A	—	—
Machine interface (I/O Link) DI/DO;1024/1024 points		DI/DO;96/64 60(W)×380(H)×172(D)mm (with MPG I/F)	☆	☆	☆	—	—
		I/O module for machine I/F (with MPG I/F)	☆	☆	☆	—	—
		I/O module for machine I/F (without MPG I/F)	☆	☆	☆	—	—
		Operator's panel I/O module (with MPG I/F)	☆	☆	☆	—	—
		Operator's panel I/O module (without MPG I/F)	☆	☆	☆	—	—
		Standard operator's panel	☆	☆	☆	—	—
		Small operator's panel	☆	☆	☆	—	—
		Connection panel I/O module (DI/DO module, 2A output module, Analog input module)	☆	☆	☆	—	—
		I/O Unit–MODEL A	☆	☆	☆	—	—
		Additional peripheral axes (I/O Link β servo)	☆	☆	☆	—	—

Item	Specifications	Series 0*i*			Series 0*i* Mate	
		MC	TC	PC	MC	TC

Others

Item	Specifications	Series 0*i* MC	Series 0*i* TC	Series 0*i* PC	Series 0*i* Mate MC	Series 0*i* Mate TC
Machine interface (I/O Link) DI/DO;240/160 points	DI/DO;96/64 60(W)×380(H)×172(D)mm (with MPG I/F)	—	—	—	☆	☆
	I/O module for machine I/F (with MPG I/F)	—	—	—	☆	☆
	I/O module for machine I/F (without MPG I/F)	—	—	—	☆	☆
	Operator's panel I/O module (with MPG I/F)	—	—	—	☆	☆
	Operator's panel I/O module (without MPG I/F)	—	—	—	☆	☆
	Standard operator's panel	—	—	—	☆	☆
	Small operator's panel	—	—	—	☆	☆
	Connection panel I/O module (DI/DO module, 2A output module, Analog input module)	—	—	—	☆	☆
	Additional peripheral axes (I/O Link β servo 1axis)	—	—	—	☆	☆
Manual pulse generator		☆	☆	☆	☆	☆
Connectable servo motor	FANUC AC SERVO MOTOR α*is* series α (HV)*is* series	●	●	●	—	—
	FANUC AC SERVO MOTOR β*is* series	●	●	●	●	●
Connectable spindle motor	FANUC AC SPINDLE MOTOR α*i* series α (HV)*i* series	●	●	—	—	—
	FANUC AC SPINDLE MOTOR α*i* P series α*i* T series α*i* L series	●	●	—	—	—
	FANUC AC SPINDLE MOTOR β*i* series	●	●	—	●	●
Connectable servo amplifier	FANUC SERVO AMPLIFIER α*i* series (PSM,SVM,SPM)	●	●	●	—	—
	FANUC SERVO AMPLIFIER β*i* series (SVPM)	●	●	●	●	●
	FANUC SERVO AMPLIFIER β*i* series (SVM)	●	●	●	●	●
	Analog spindle I/F	☆	☆	—	☆	☆
Position detector unit for full–closed control (for full–closed control)	Pulse coder/optical scale (2–phase pulse interface)	☆	☆	☆	☆	☆
	Pulse coder/optical scale (serial interface)	☆	☆	☆	☆	☆
SERVO GUIDE		☆	☆	☆	☆	☆
Input power supply	24VDC±10%	○	○	○	○	○
Ambient temperature of unit	At operating: 0°C to 58°C At nonoperating: –20°C to 60°C	○	○	○	○	○
Ambient relative humidity	Normally: 75%RH or less (No dew, nor frost allowed) Short term (within one month): 95%RH or less (No dew, nor frost allowed)	○	○	○	○	○
Vibration	At operating: 0.5G or less At nonoperating: 1G or less	○	○	○	○	○

NOTE

*1 : Control unit is incorporated with disply unit.

*2 : The actual registrable value might changes according to the registered number of programs and the program sizes.

II. NC FUNCTION

PREFACE

This part describes the functions that can be performed on all models. For the functions available with each model, see the list of specifications in Part I.

1 CONTROLLED AXES

1.1 NUMBER OF THE ALL CONTROLLED AXES

	0*i* Mate–TC	**0*i* Mate–MC**	**0*i*–TC**	**0*i*–MC**
Number of controlled axes (including Cs axis)	3	3	4	4
Number of controlled paths	1	1	1	1
Number of basic controlled axes	2	3	2	3
Number of basic simultaneously controlled axes	2	3	2	3
Number of controlled axes expanded (all)	3	3	4	4
Number of simultaneously controlled axes expanded (all)	3	3	4	4
Axis control by PMC	0	0	4	4
Cs contour control	1	0	1	1

1.2 AXIS NAMES

T series :

The two basic axes are always set to X and Z. Additional axes can be selected from A, B, C, U, V, W, and Y freely.

> **NOTE**
> If U, V, or W is used as an axis name, the G code system must be either B or C.

M series :

The three basic axes are set to X, Y, and Z. Additional axes can be selected from A, B, C, U, V, and W freely.

1.3 INCREMENT SYSTEM

There are two increment systems as shown in the tables below. One of the increment systems can be selected using a parameter.

NOTE
If IS-C is selected, function “increment system 1/10” is required.

Table 1.3 (a) IS–B

		Least input increment	Least command increment	Abbreviation
Millimeter machine	Metric input	0.001 mm (diameter programming)	0.0005 mm	IS–B
		0.001 mm (radius programming)	0.001 mm	
		0.001 deg	0.001 deg	
	Inch input	0.0001 inch (diameter programming)	0.0005 mm	
		0.0001 inch (radius programming)	0.001 mm	
		0.001 deg	0.001 deg	
Inch machine	Metric input	0.001 mm (diameter programming)	0.00005 inch	
		0.001 mm (radius programming)	0.0001 inch	
		0.001 deg	0.001 deg	
	Inch input	0.0001 inch (diameter programming)	0.00005 inch	
		0.0001 inch (radius programming)	0.0001 inch	
		0.001 deg	0.001 deg	

Table 1.3 (b) IS–C

		Least input increment	Least command increment	Abbreviation
Millimeter machine	Metric input	0.0001 mm (diameter programming)	0.00005 mm	IS–C
		0.0001 mm (radius programming)	0.0001 mm	
		0.0001 deg	0.0001 deg	
	Inch input	0.00001 inch (diameter programming)	0.00005 mm	
		0.00001 inch (radius programming)	0.0001 mm	
		0.0001 deg	0.0001 deg	
Inch machine	Metric input	0.0001 mm (diameter programming)	0.000005 inch	
		0.0001 mm (radius programming)	0.00001 inch	
		0.0001 deg	0.0001 deg	
	Inch input	0.00001 inch (diameter programming)	0.000005 inch	
		0.00001 inch (radius programming)	0.00001 inch	
		0.0001 deg	0.0001 deg	

The least command increment is in millimeters or inches, depending on the machine tool. One of them must be selected using a parameter beforehand.
The least input increment can be switched between metric input and inch input by using a G code (G20 or G21) or a setting parameter.

1.3.1 Input Unit (10 Times)

The following least input increments can be set using a parameter:

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

NOTE
The minimum input increment for inch input is not affected.

1.4 MAXIMUM STROKE

The following table lists the maximum strokes of machine tools that are allowed by the control unit:

Maximum stroke = Least command increment × 99999999

Increment system		Maximum stroke
IS–B	Millimeter machine	±99999.999 mm ±99999.999 deg
	Inch machine	±9999.9999 inch ±99999.999 deg
IS–C	Millimeter machine	±9999.9999 mm ±9999.9999 deg
	Inch machine	±999.99999 inch ±9999.9999 deg

NOTE

1 The values (in mm or inches) in the table are diameter values if diameter programming is specified, or radius values if radius programming is specified.
2 A command that exceeds the maximum stroke is not allowed.

2 PREPARATORY FUNCTIONS

2.1 T SERIES

The following G codes are provided. The G codes are classified into three: A, B, and C. One of the G code types can be selected using a parameter. In this manual, G code system B is assumed.

G code list for T series (1/3)

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04	00	Dwell
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting
G36	G36	G36	00	Automatic tool compensation X (When the bit 3 (G36) of parameter No. 3405 is set to 0)
G37	G37	G37		Automatic tool compensation Z
G37.1	G37.1	G37.1		Automatic tool compensation X
G37.2	G37.2	G37.2		Automatic tool compensation Z

G code list for T series (2/3)

G code A	G code B	G code C	Group	Function
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordinate system preset
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscilation grinding cycle (for grinding machine)
G74	G74	G75		Oscilation direct constant–dimension grinding cycle (for grinding machine)

G code list for T series (3/3)

G code A	G code B	G code C	Group	Function
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per rotation feed
–	G90	G90	03	Absolute programming
–	G91	G91		Incremental programming
–	G98	G98	11	Return to initial level
–	G99	G99		Return to R point level

2.2 M SERIES

The following G codes are provided :

G code list for M series (1/3)

G code	Group	Function	
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G04	00	Dwell, Exact stop	
G05.1		AI contour/AI nano contour/AI advanced preview/smooth interpolation	
G05.4		HRV3 on/off	
G07.1 (G107)		Cylindrical interpolation	
G08		Advanced preview control	
G09		Exact stop	
G10		Programmable data input	
G11		Programmable data input mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G29		Automatic return from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Thread cutting	
G37	00	Automatic tool length measurment	
G40	07	Cutter compensation cancel	
G41		Cutter compensation left	
G42		Cutter compensation right	

G code list for M series (2/3)

G code	Group	Function
G40.1 (G150)	19	Normal direction control cancel mode
G41.1 (G151)		Normal direction control left side on
G42.1 (G152)		Normal direction control right side on
G43	08	Tool length compensation + direction
G44		Tool length compensation – direction
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset double increase
G48		Tool offset double decrease
G49	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Workpiece coordinate system 1 selection
G54.1		Additional workpiece coordinate system selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00/01	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle

G code list for M series (3/3)

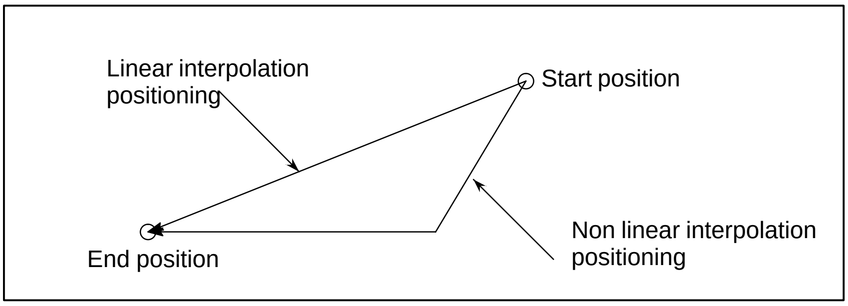
G code	Group	Function
G75	01	Plunge grinding cycle (for grinding machine)
G76	09	Fine boring cycle
G77	01	Direct constant–dimension plunge grinding cycle(for grinding machine)
G78		Continuous–feed surface grinding cycle(for grinding machine)
G79		Intermittent–feed surface grinding cycle(for grinding machine)
G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G94	05	Feed per minute
G95		Feed per rotation
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G160	20	In–feed control function cancel(for grinding machine)
G161		In–feed control function(for grinding machine)

3 INTERPOLATION FUNCTIONS

3.1 POSITIONING (G00)

Positioning is done with each axis separately (Non linear interpolation type positioning).
Either of the following tool paths can be selected accroding to bit 1 of parameter No. 1401.

- **Non linear interpolation positioning**
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- **Linear interpolation posioning**
  The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.
  When the acceleration/deceleration type is changed from the constant acceleration (inclination) type to the constant time (time constant) type, the tool can move along a specified path.



It is decelerated, to a stop at the end point, and imposition check is performed (checks whether the machine has come to the specified position). The in-position check can be suppressed using a parameter.
Width of imposition can be set as a parameter.

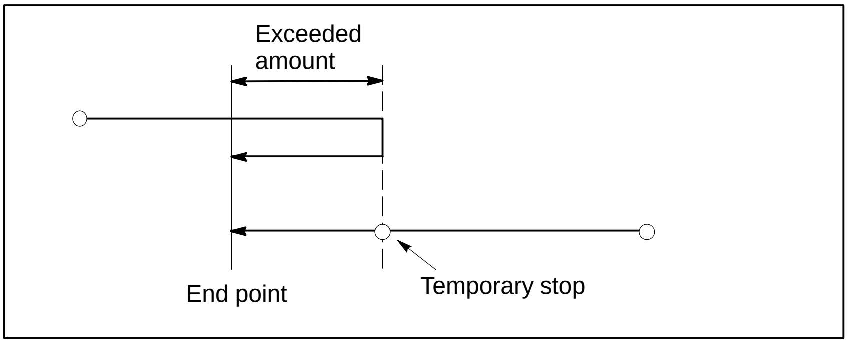
Format

G00 IP_ ;

3.2 SINGLE DIRECTION POSITIONING (G60)

M series

It is always controlled to perform positioning to the end point from a single direction, for better precision in positioning. If direction from start point to end point is different from the predecided direction, it once positions to a point past the end point, and the positioning is reperformed for that point to the end point.
Even if the direction from start point to end point is the same as predecided direction, the tool stops once before the end point.

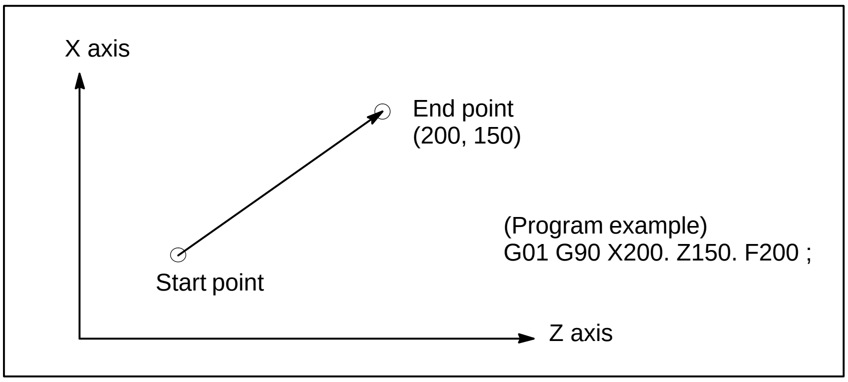


Format

G60 IP_ ;

3.3 LINEAR INTERPOLATION (G01)

Linear interpolation is done with tangential direction feed rate specified by the F code.



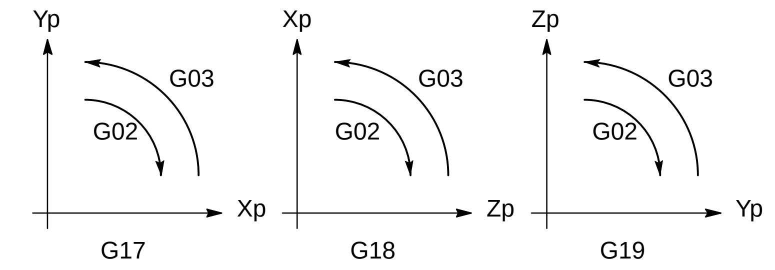
Format

G01 IP_ F_ ;

F : Feedrate

3.4 CIRCULAR INTERPOLATION (G02, G03)

Circular interpolation of optional angle from 0° to 360 ° can be specified.
G02: Clockwise (CW) circular interpolation
G03: Counterclockwise (CCW) circular interpolation



Feed rate of the tangential direction takes the speed specified by the F code. Planes to perform circular interpolation is specified by G17, G18, G19. Circular interpolation can be performed not only on the X, Y, and Z axis but also on the parallel axes of the X, Y, and Z axes.

G17: Xp-Yp plane
G18: Zp-Xp plane
G19: Yp-Zp plane
where

Xp: X axis or its parallel axis
Yp: Y axis or its parallel axis
Zp: Z axis or its parallel axis

Parameter is set to decide which parallel axis of the X, Y, Z axes to be the additional axis.

Format

Arc on the Xp-Yp plane

$$\mathbf{G17}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Xp\_\ Yp\_}\left\{\begin{matrix}\mathbf{R\_}\\ \mathbf{I\_\ J\_}\end{matrix}\right\}\mathbf{F\_\ ;}$$

Arc on the Zp-Xp plane

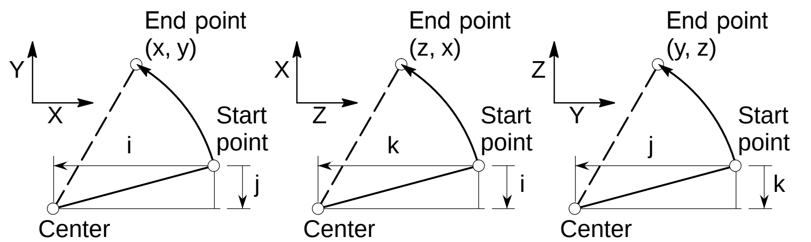
$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Zp\_\ Xp\_}\left\{\begin{matrix}\mathbf{R\_}\\ \mathbf{K\_\ I\_}\end{matrix}\right\}\mathbf{F\_\ ;}$$

Arc on the Yp-Zp plane

$$\mathbf{G19}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\ \mathbf{Yp\_\ Zp\_}\left\{\begin{matrix}\mathbf{R\_}\\ \mathbf{J\_\ K\_}\end{matrix}\right\}\mathbf{F\_\ ;}$$

I_. J_, K_:Distance of the X, Y, Z axes from the start point to the center of the circle

R:Arc radius (For an arc having a central angle of 180° or greater, specify an R value with a minus sign. A complete circumference cannot be specified.)



The following shows the arc radius range that can be specified.

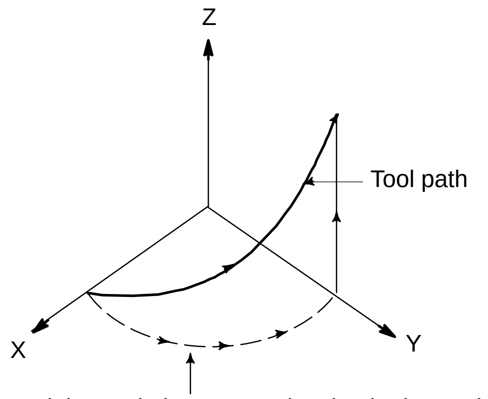
		Input increments	
		Metric input	**Inch input**
Increment system	IS–B	0.001 to 99999.999mm	0.0001 to 9999.9999inch
	IS–C	0.0001 to 9999.9999mm	0.00001 to 999.99999inch

3.5 HELICAL INTERPOLATION (G02, G03)

Helical interpolation performs circular interpolation of a maximum of two axes, synchronizing with other optional two axes circular interpolation. Thread cutting of large radius threads or machining of solid cams are possible by moving a tool in a spiral.

The commanded speed is the speed of the tangential direction of the arc. Thus, the speed of a linear axis is expressed as follows:

$$F \times \frac{Length of linear axis}{Arc length}$$



Tangential speed along an arc by circular interpolation is the rate specified in programming.

Format

Arc on the Xp-Yp plane

$$\mathbf{G17} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Xp\_\ Yp\_} \begin{Bmatrix} \mathbf{R\_} \\ \mathbf{I\_\ J\_} \end{Bmatrix} \alpha\_\ (\beta\_)\ \mathbf{F\_\ ;}$$

Arc on the Zp-Xp plane

$$\mathbf{G18} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Zp\_\ Xp\_} \begin{Bmatrix} \mathbf{R\_} \\ \mathbf{K\_\ I\_} \end{Bmatrix} \alpha\_\ (\beta\_)\ \mathbf{F\_\ ;}$$

Arc on the Yp-Zp plane

$$\mathbf{G19} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Yp\_\ Zp\_} \begin{Bmatrix} \mathbf{R\_} \\ \mathbf{J\_\ K\_} \end{Bmatrix} \alpha\_\ (\beta\_)\ \mathbf{F\_\ ;}$$

α, β: Optional axis other than the circular interpolation axes

3.6 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

The function in which contour control is done in converting the command programmed in a cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece) is the polar coordinate interpolation. It is an effective function when a straight line groove is cut on the outer diameter of a workpiece or when a cam shaft is ground.
Whether the polar coordinate interpolation is done or not is commanded by a G code.
These G codes shall be commanded in a single block.

Format

G12.1; Polar coordinate interpolation mode
(Polar coordinate interpolation shall be done.)
G13.1; Polar coordinate interpolation cancel mode
(Polar coordinate interpolation is not done.)

Explanations

- **Polar coordinate interpolation mode (G12.1)**

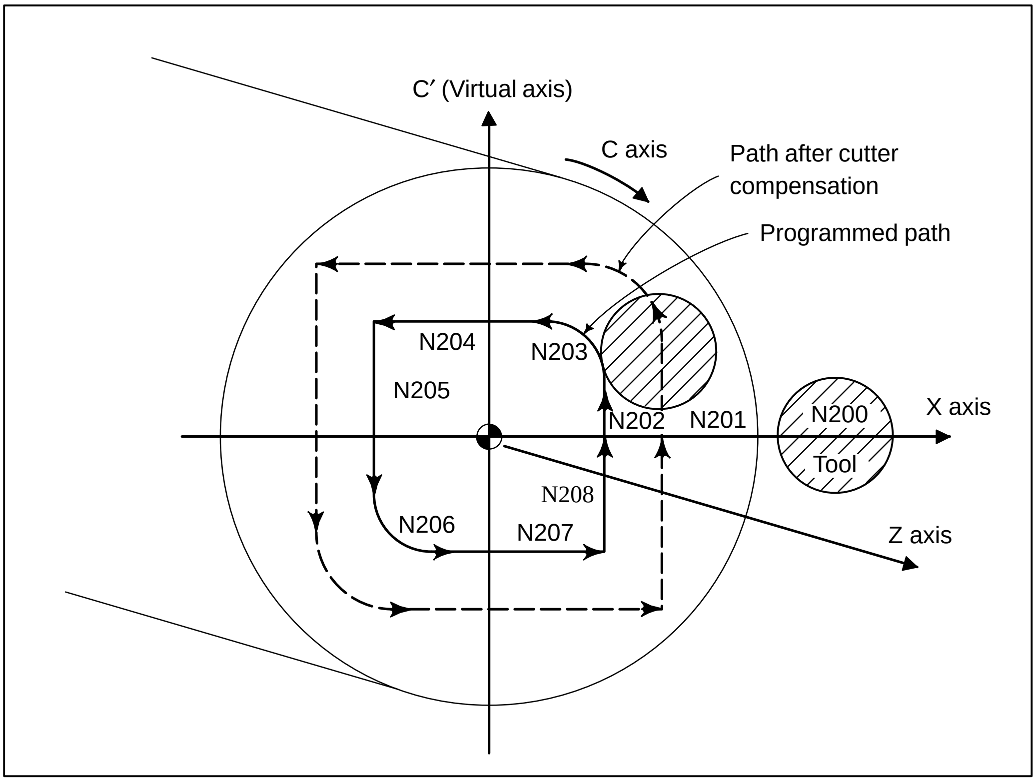
The axes (linear axis and rotary axis) on which polar coordinate interpolation is done are set beforehand by parameters.
Change the mode to polar coordinate interpolation mode by commanding G12.1, and a plane (hereinafter referred to as polar coordinate interpolation plane) is selected in which linear axis is made to the first axis of the plane, and virtual axis being a right angle with the linear axis is made to the second axis of the plane. Polar coordinate interpolation is carried out on this plane.
In the polar coordinate interpolation made, the command of linear interpolation (G01) and circular interpolation (G02, G03) is possible. And both absolute command (G90) and incremental command (G91) are possible.
For the program command it is possible to apply cutter compensation. For the path after cutter compensation is done, polar coordinate interpolation can be made.
As for feedrate, specify the tangential speed (relative speed between the workpiece and the tool) on the polar coordinate interpolation plane (cartesian coordinate system) with F.

- **Polar coordinate interpolation cancel mode (G13.1)**

The polar coordinate interpolation cancel mode is obtained by G13.1 command.

Examples

- **Polar coordinate interpolation by X axis (Linear axis) and C axis (Rotary axis)**



(X axis is diameter programming and C axis is radius programming)

00001;	
:	
N100 G90 G00 X120.0 C0 Z_ ;	Positioning to the starting position
N200 G12.1;	Starting polar coordinate interpolation
N201 G42 G01 X40.0 F D01;	Contour program (Program in cartesian coordinate system of X-C′ plane)
N202 C10.0;	
N203 G03 X20.0 C20.0 R10.0 ;	
N204 G01 X-40.0 ;	
N205 G-10.0 ;	
N206 G03 X-20.0 C-20.0 I10.0 K0 ;	
N207 G01 X40.0 ;	
N208 C0 ;	
N209 G40 X120.0 ;	
N210 G13.1 ;	Canceling polar coordinate interpolation
N300 Z_ ;	
N400 X_ C_ ;	
:	
M30 ;	

3.7 CYLINDRICAL INTERPOLATION (G07.1)

When the form on the expanded side view of a cylinder (from on the cylinder coordinate system) is commanded by a program command, the NC converts the form into a linear axis movement and a rotary axis movement then performs a contour control. This feature is called the cylindrical interpolation.
Cylindrical interpolation is commanded with G07.1.

Format

G07.1 (Name of rotary axis) Radius value of cylinder ;
Cylindrical interpolation mode
G07.1 (Name of rotary axis) 0 ;
Cancellation mode of cylindrical interpolation

Explanations

- **Cylindrical interpolation mode**

Cylindrical interpolation is made between the rotary axis specified in the block of G07.1 and the other optional linear axis.
Circle interpolation command is allowed as well as linear interpolation, during cylindrical interpolation mode. Also, absolute command and incremental command can be made. Cutter compensation can be added to the program command. Cylindrical interpolation is made for the path after cutter compensation.
Feed rate gives the tangential speed on the expanded plane of the cylinder with F.

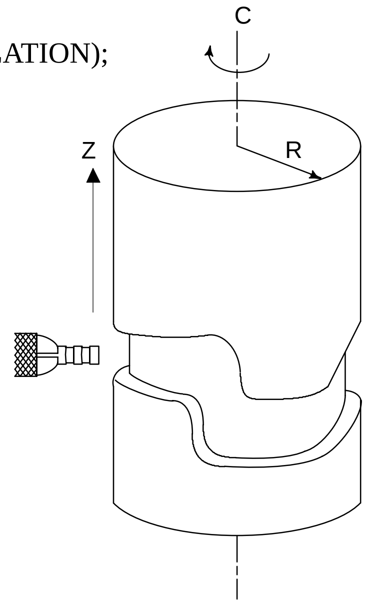
- **Cancellation mode of cylindrical interpolation**

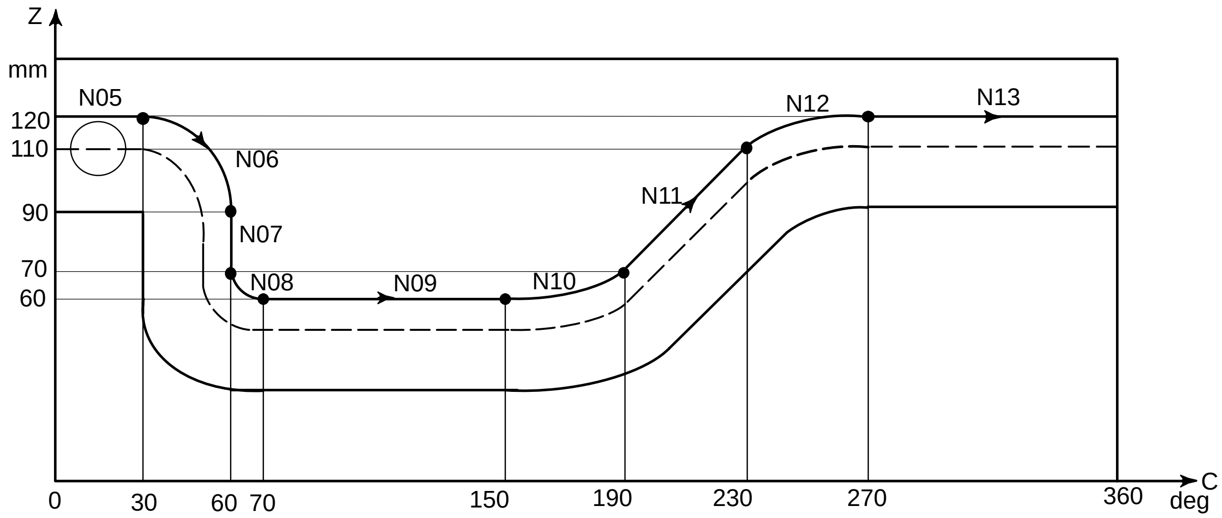
G07.1 (Name of rotary axis) 0; Cancellation mode of cylindrical interpolation is made when commanded as above.

Examples

An example of a program

```
O0001 (CYLINDRICAL INTERPOLATION);
  N1 G00 G00 Z100.0 C0;
  N2 G01 G18 Z0 C0;
  N3 G7.1 C57299;
  N4 G01 G42 Z120.0 D10 F250;
  N5 G40.0;
  N6 G03 Z90.0 C60.0 R30.0 ;
  N7 G01 Z70.0;
  N8 G02 Z60.0 C70.0 R10.0;
  N9 G01 C150.0;
  N10 G02 Z70.0 C190.0 R75.0;
  N11 G01 Z110.0 C230.0;
  N12 G03 Z120.0 C270.0 R75.0;
  N13 G01 G360.0;
  N14 G40 Z100.0;
  N15 G07.1 C0;
  N16
  M30;
```

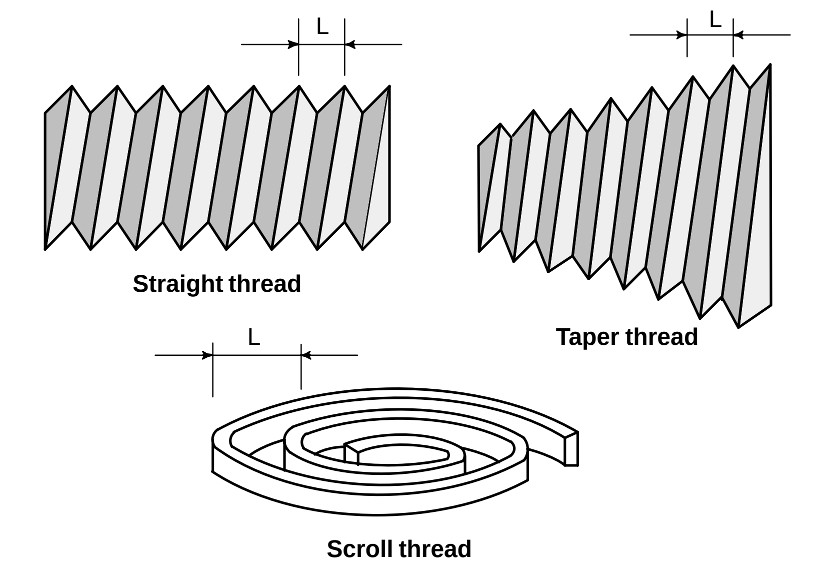




4 THREAD CUTTING

4.1 EQUAL LEAD THREAD CUTTING (G33) (WITH G CODE SYSTEM A: G32)

By feeding the tool synchronizing with the spindle rotation, thread cutting of the specified lead is performed. In addition to straight threads, taper threads and scroll threads can be cut with equal leads.



Format

G33 IP_ **F_ :**

F_ : Lead along the long axis (axis having the largest amount of travel)

Explanations

To form a single thread, threading is generally performed several times from rough machining to finish machining along the same path. Threading starts when the one-revolution signal from the position coder attached to the spindle is detected. So threading always starts at the same point on the circumference of the workpiece, and threading is performed along the same path on the workpiece. In this case, however, the shaft must rotate at a constant speed during operations from rough machining to finish machining. If the spindle speed changes, an accurate thread may not be produced.

The following shows the specifiable lead range:

- **M series**

	Specifiable lead range
Metric input	F1 to F50000 (0.01 to 500.00mm)
Inch input	F1 to F99999 (0.0001 to 9.9999inch)

- **T series**

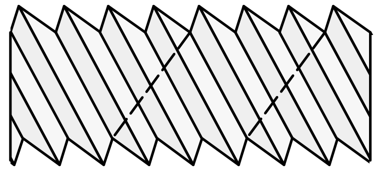
	Specifiable lead range
Metric input	0.0001 to 500.0000mm
Inch input	0.000001 to 9.999999inch

NOTE

Leads exceeding the maximum cutting feed speed when converted to per minute feed speed cannot be specified.

4.2 MULTIPLE–THREAD CUTTING (G33)

T series


Multiple–thread screws

Format

Constant–lead threading

G33 IP_ **F_ Q_ ;**

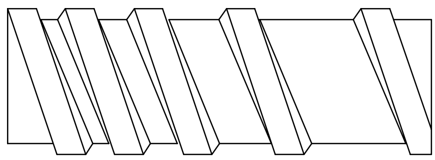
G33 IP_ **Q_ ;**

IP_ : End point
F_ : Lead in longitudinal direction
Q_ : Threading start angle

4.3 VARIABLE LEAD THREAD CUTTING (G34)

T series

Variable lead thread cutting can be done by commanding long axis direction lead and lead increase/decrease per spindle rotation.



Format

G34 IP_ **F_ K_ :**

IP_ : End point
F_ : Long axis direction lead at start point
K_ : Lead increase/decrease per spindle rotation

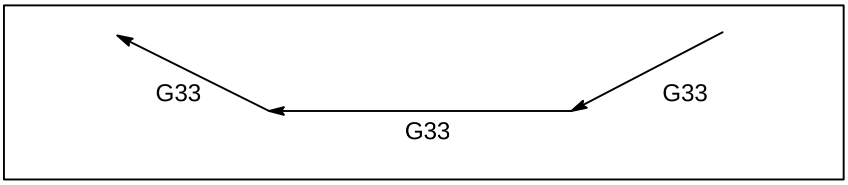
Command value range of lead increase/decrease (K) per spindle rotation:

	Command value range
Metric input	±0.0001 to ±500.0000 mm/rev
Inch input	±0.000001 to ±9.999999 inch/re

4.4 CONTINUOUS THREAD CUTTING

T series

Continuous thread cutting in which thread cutting command block is continuously commanded is available. As it is controlled so that the spindle synchronism shift (occurred when shifting from one block to another) is kept to a minimum, special threads like threads which leads or shape change during the cycle can also be cut.



5 FEED FUNCTIONS

5.1 RAPID TRAVERSE

Positioning of each axis is done in rapid motion by the positioning command (G00).
There is no need to program rapid traverse rate, because the rates are set in the parameter (per axis).

Least command increment	Rapid traverse rate range
0.001mm, deg	30 to 240000mm/min, deg/min
0.0001mm, deg	30 to 100000mm/min, deg/min
0.0001inch	3.0 to 9600.0inch/min
0.00001inch	3.0 to 4000.0inch/min

NOTE

The above feed rates are limits according to the NC's interpolation capacity when the high-resolution detection interface is equipped. When the whole system is considered, there are also limits according to the servo system. For details, refer to Appendix A.

5.2 CUTTING FEED RATE

Feed rates of linear interpolation (G01), and circular interpolation (G02, G03) are commanded with numbers after the F code.

5.2.1 Tangential Speed Constant Control

In cutting feed, it is controlled so that speed of the tangential direction is always the same commanded speed.

5.2.2 Cutting Feed Rate Clamp

Cutting feed rate upper limit can be set as parameters. If the actual cutting feed rate (feed rate with override) is commanded exceeding the upper limit, it is clamped to a speed not exceeding the upper limit.

5.2.3 Per Minute Feed (G94)

With the per minute feed mode G94, tool feed rate per minute is directly commanded by numerical value after F.

Least command increment	Cutting feed rate range
0.001mm, deg	1 to 240000mm/min, deg/min
0.0001mm, deg	1 to 100000mm/min, deg/min
0.0001inch	0.01 to 9600.0inch/min
0.00001inch	0.01 to 4000.0inch/min

NOTE

The above feed rates are limits according to the NC's interpolation capacity. When the whole system is considered, there are also limits according to the servo system. For details, see Appendix A.

5.2.4 Per Revolution Feed (G95)

With the per revolution feed mode G95, tool feed rate per revolution of the spindle is directly commanded by numeral after F. A position coder must be mounted on the spindle.
For the T series, however, the feed–per–revolution command can be enabled by setting the corresponding parameter accordingly, even when the position coder is not installed (feed per revolution without position coder).

- **M series**

Least command increment	Cutting feed rate range
0.001mm, deg	0.01 to 500.00mm/rev, deg/rev
0.0001mm, deg	0.01 to 500.00mm/rev, deg/rev
0.0001inch	0.0001 to 9.9999inch/rev
0.00001inch	0.0001 to 9.9999inch/rev

- **T series**

Least command increment	Cutting feed rate range
0.001mm, deg	0.0001 to 500.0000mm/rev, deg/rev
0.0001mm, deg	0.0001 to 500.0000mm/rev, deg/rev
0.0001inch	0.000001 to 9.999999inch/rev
0.00001inch	0.000001 to 9.999999inch/rev

NOTE
The above feed rates are limits according to the CNC's interpolation capacity. When the whole system is considered there are also limits according to the servo system. For details, See Appendix A.

5.2.5 One–digit F Code Feed

M series

When a 1-digit number from 1 to 9 is commanded after the F, the preset speed corresponding the 1-digit number commanded is set as feed rate. When F0 is commanded, rapid traverse is set.
Set the one-digit F code feed rate change input signal on from the machine side, and rotate the manual pulse generator. Feed rate of the currently selected speed can be changed.
Feed rate set or changed will be memorized even after power is turned off.

5.3 OVERRIDE

5.3.1 Feed Rate Override

The per minute feed (G94) and per rotation feed (G95) can be overrided by:

0 to 254% (per every 1%).

In inverse time, feed rate converted to per minute feed is overridden. Feed rate override cannot be performed to F1-digit feed.

Feed rate also cannot be performed to functions as thread cutting and tapping in which override is inhibited.

5.3.2 Rapid Traverse Override

Rapid traverse rate can be overridden by:

F0, 25%, 50%, 100%.

F0: A constant speed per axis is set by parameter

An override of 0% to 100% can be applied in 1% steps using a signal.

5.3.3 Override Cancel

Feed rate override and the second feed rate override can be clamped to 100% by a signal from the machine side.

5.3.4 Jog Override

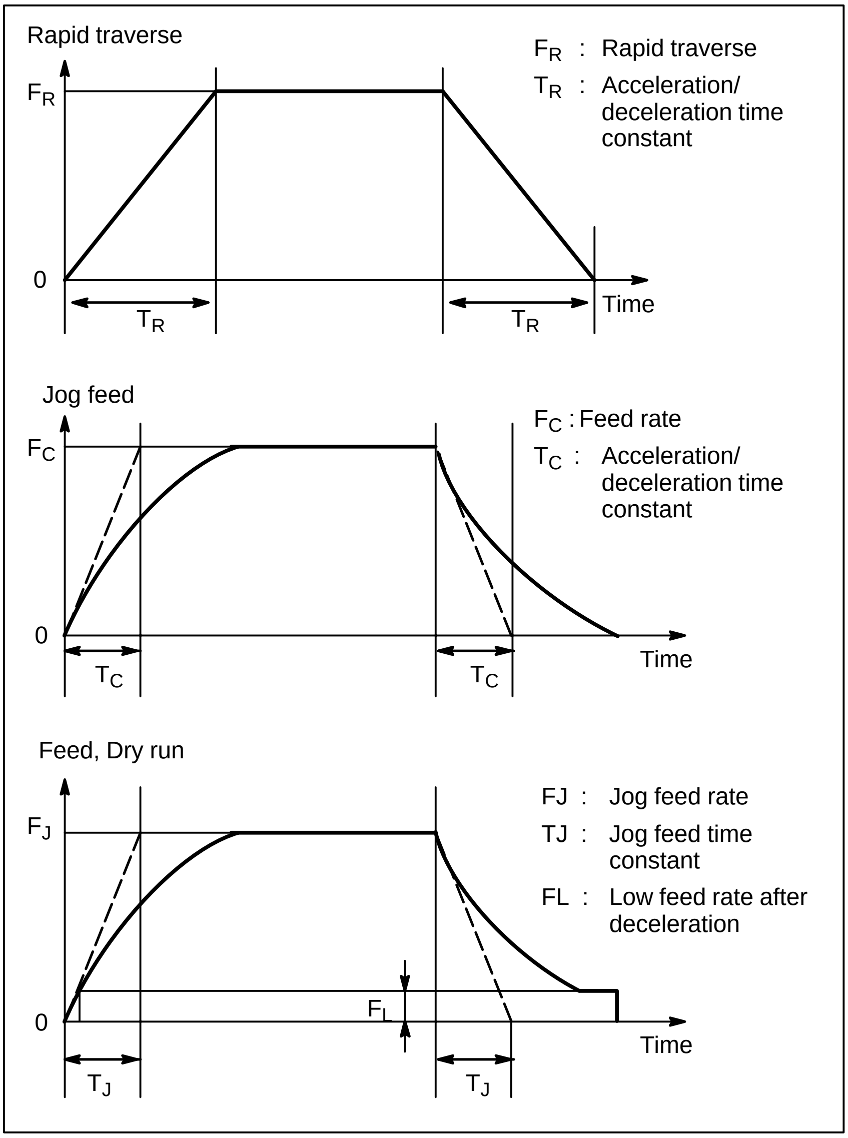
The manual continuous feedrate and incremental feed rate can be overridden by:

0% to 655.34% (in steps of 0.01%)

5.4 AUTOMATIC ACCELERATION/ DECELERATION

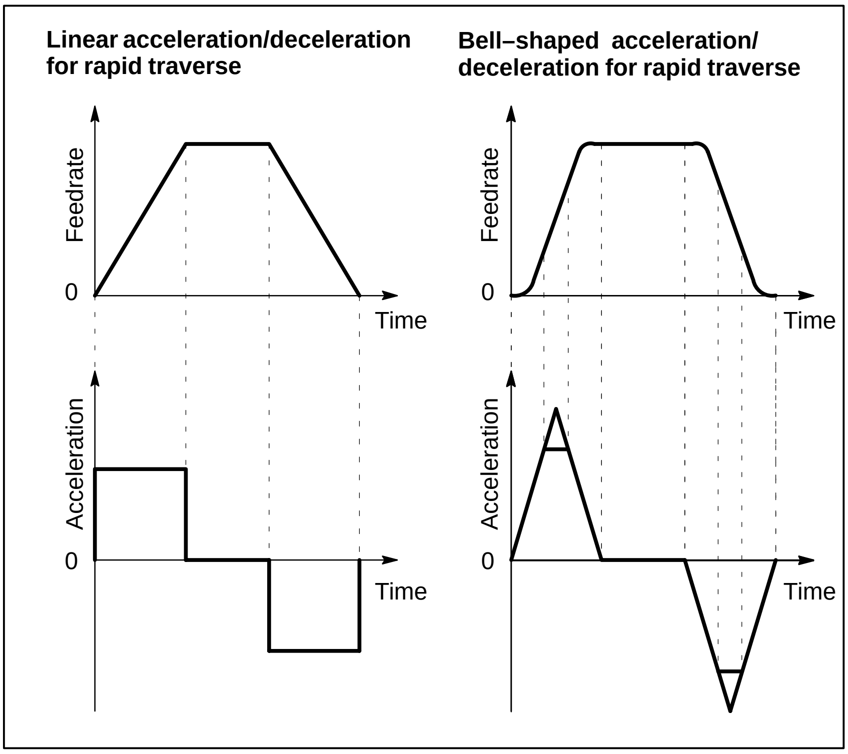
Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feed rate changes, so change in speed is also smoothly done.

Rapid traverse	:	Linear acceleration/deceleration (time constant is parameter set per axis)
Cutting feed	:	Exponential acceleration/deceleration (time constant is parameter set per axis)
Jogging	:	Exponential acceleration/deceleration (time constant is parameter set per axis)

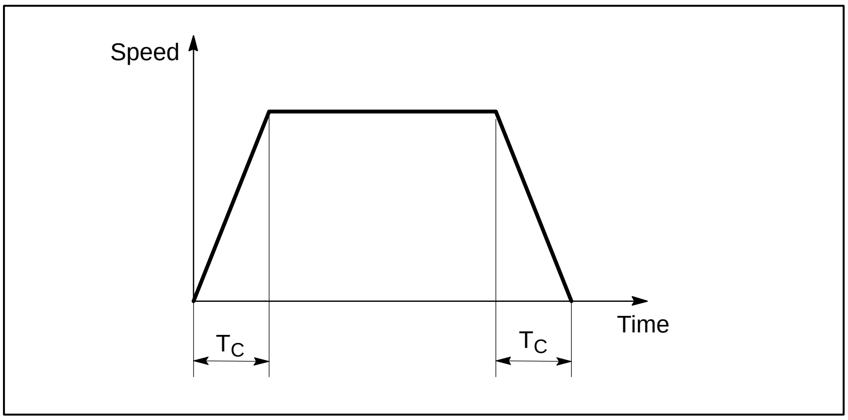


5.5 RAPID TRAVERSE BELL–SHAPED ACCELERATION/ DECELERATION

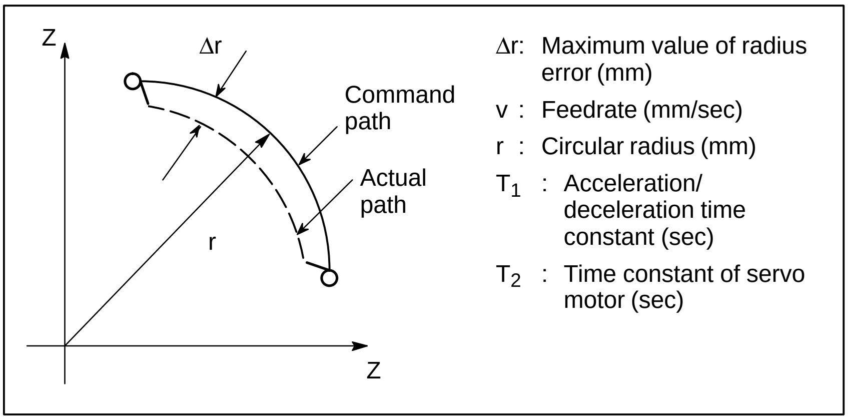
The function for rapid traverse bell–shaped acceleration/deceleration increases or decreases the rapid traverse feedrate smoothly.
This reduces the shock to the machine system due to changing acceleration when the feedrate is changed.
As compared with linear acceleration/deceleration, bell–shaped acceleration/deceleration allows smaller time constants to be set, reducing the time required for acceleration/deceleration.



5.6 LINEAR ACCELERATION/ DECELERATION AFTER CUTTING FEED INTERPOLATION



In the linear acceleration/deceleration, the delay for the command caused by the acceleration/ deceleration becomes 1/2 compared with that in exponential acceleration/deceleration, substantially reducing the time required for acceleration and deceleration.
Also, the radius direction error in the circular interpolation caused by the acceleration/deceleration is substantially reduced.



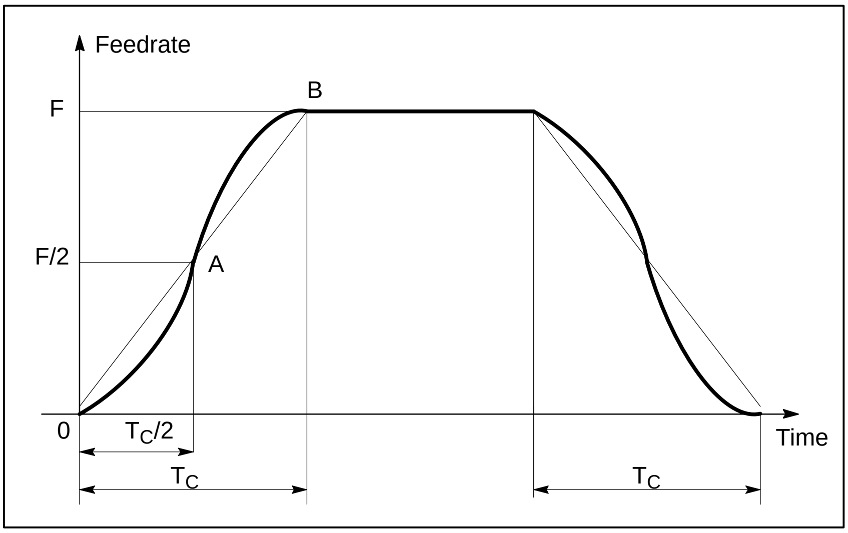
The maximum value of error in this radius direction is obtained approximately by the following equation.

$$\Delta r = (\frac{1}{2}T_1{}^2 + \frac{1}{2}T_2{}^2)\frac{V^2}{r} \quad \ldots\ldots \text{ For exponential acceleration/deceleration}$$

$$\Delta r = (\frac{1}{24}T_1{}^2 + \frac{1}{2}T_2{}^2)\frac{V^2}{r} \quad \ldots\ldots \text{ For linear acceleration/deceleration after cutting feed interpolation}$$

Consequently, in case of the linear acceleration/deceleration after interpolation, if an error caused by the servo loop time constant is excluded, the radius directional error will be reduced to 1/12, compared with the exponential acceleration/deceleration.

5.7 BELL–SHAPED ACCELERATION/ DECELERATION AFTER CUTTING FEED INTERPOLATION



As shown above in the quadratic curve, it is possible to accelerate and decelerate the cutting feedrate.

When the acceleration and deceleration section are connected, the composed curve shapes just like a hanging bell. That is why this kind of acceleration/deceleration is called bell–shaped acceleration/deceleration.

Considering a time constant as Tc (time spent to accelerate from feedrate 0 up to commanded feedrate F or time spent to decelerate from commanded feedrate F down to feedrate 0), feedrate accelerates up to 1/2 of the commanded feedrate (F/2) for 1/2 of the time constant (Tc/2). The acceleration/deceleration curve 0A shown in the figure above can be expressed by the following equation :

$$f(t) = \frac{2F}{{T_C}^2} t^2$$

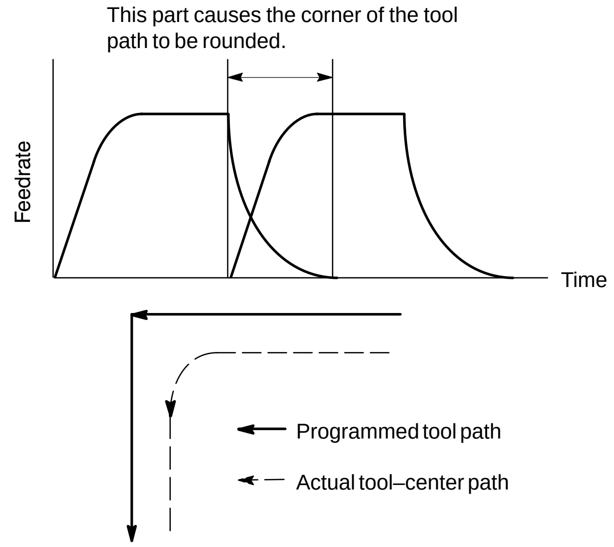
The curve AB and 0A are symmetric with respect to point A.

The feature of this acceleration/deceleration is that the feedrate change is small near feedrate 0 and the commanded feedrate.

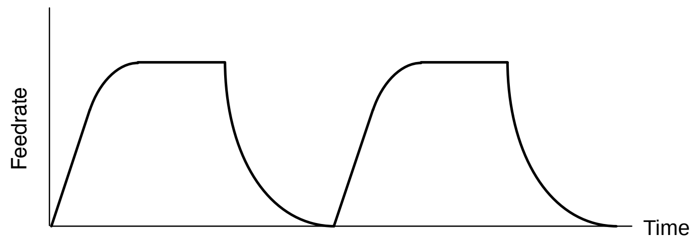
5.8 ERROR DETECTION

T series

Generally, the CNC does not zero the feedrate at the interface of two blocks during cutting feed.
Because of this, a corner of a tool path may be rounded.

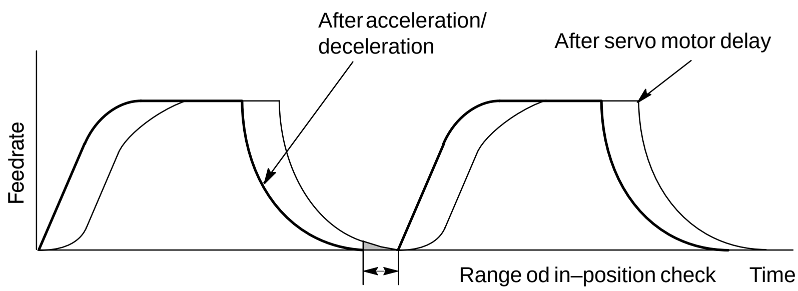


If the error detect signal is used, it is possible to specify that a block not be started until the acceleration/deceleration of the previous block has been completed.



NOTE

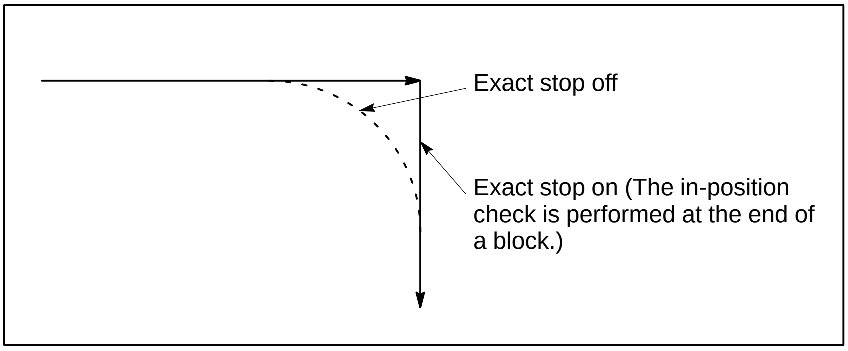
If the error detect signal is on, a cutting block is not executed until the acceleration/deceleration of the previous cutting block has been completed.
This function alone cannot prevent corner rounding due to delay caused by the servo motor, however.
To prevent corner rounding due to delay caused by the servo motor, use the in–position check function together with this function.



5.9 EXACT STOP (G09)

M series

Move command in blocks commanded with G09 decelerates at the end point, and in–position check is performed. G09 command is not necessary for deceleration at the end point for positioning (G00) and in–position check is also done automatically. This function is used when sharp edges are required for workpiece corners in cutting feed.



5.10 EXACT STOP MODE (G61)

M series

When G61 is commanded, deceleration of cutting feed command at the end point and in–position check is performed per block thereafter. This G61 is valid till G64 (cutting mode), G62 (automatic corner override), or G63 (tapping mode) is commanded.

5.11 CUTTING MODE (G64)

M series

When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid till G61 (exact stop mode), G62 (automatic corner override), or G63 (tapping mode) is commanded.

5.12 TAPPING MODE (G63)

M series

When G63 is commanded, feed rate override is ignored (always regarded as 100%), and feed hold also becomes invalid. Cutting feed does not decelerate at the end of block to transfer to the next block. And in-tapping signal is issued during tapping operation. This G63 is valid till G61 (exact stop mode), G62 (automatic corner override), or G64 (cutting mode) is commanded.

5.13 AUTOMATIC CORNER OVERRIDE (G62)

M series

When G62 is commanded during cutter compensation, cutting feed rate is automatically overridden at corner. The cutting quantity per unit time of the corner is thus controlled not to increase. This G62 is valid till G61 (exact stop mode), G64 (cutting mode), or G63 (tapping mode) is commanded.

5.14 DWELL (G04)

With the G04 command, shifting to the next block can be delayed.
When commanded with a per minute feed mode (G94), shifting to the next block can be delayed for the commanded minutes.
When commanded with a per rotation feed mode (G95), shifting to the next block can be delayed till the spindle rotates for the commanded times.
Dwell may always be performed by time irrespective of G94 and G95 by parameter selection.

Format

Per second dwell

G94 G04 { P_ / X_ } ;

P_ or X_ : Dwell time commanded in seconds (0.001-99999.999 sec)

Per revolution dwell

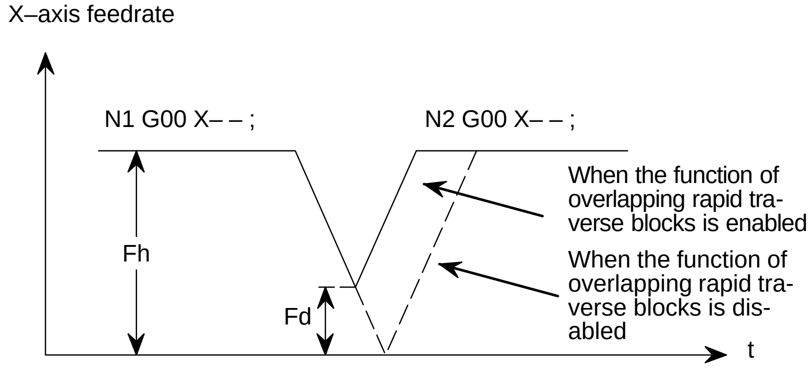
G95 G04 { P_ / X_ } ;

P_ or X_ : Spindle rotation angle commanded in rev. (0.001-99999.999 rev)

5.15 RAPID TRAVERSE BLOCK OVERLAP

If rapid traverse blocks are specified successively, or if the block next to a rapid traverse block does not include any tool movements, the execution of the next block can be started when the feedrate of each axis in the rapid traverse block has decreased to the parameter–set deceleration ratio.

Examples



Fh : Rapid traverse feedrate
α : Setting of parameter No.1722 (feedrate reduction ratio)
Fd : Feedrate where deceleration is terminated: Fh x α/100

> **NOTE**
> The parameter No.1722 is effective when parameter No.1601 #4 (RT0) is set to 1.

6 REFERENCE POSITION

6.1 MANUAL REFERENCE POSITION RETURN

Positioning to the reference position can be done by manual operation. With jogging mode (JOG), manual reference position return (ZRN) signals, and signal for selecting manual reference position return axis (±J1 to ±J8) on, the tool the machine is turned on, it decelerates, and when it is turned off again, it stops at the first grid point, and reference position return end signal is output. This point is the reference position.
By performing manual reference position return, the machine coordinate system and the work coordinate system is established.
There is only one method available to perform manual reference point return:
In the grid method, a certain grid of the position detection is appointed as the reference position. The reference position position can be shifted by the grid shift function.

6.2 SETTING THE REFERENCE POSITION WITHOUT DOGS

This function moves the machine to around the reference position set for each axis in the manual continuous feed mode. Then it sets the reference position for the machine in the manual reference position return mode without the deceleration signal for reference position return. With this function, the machine reference position can be set at a given position without the deceleration signal for reference position return.

Explanations

- **Setting the reference position**

1 Place the machine in the manual continuous feed mode, and perform positioning to a position near but not exceeding the reference position from reference position return direction (setting by parameter).

2 Enter the manual reference position return mode, then input the feed axis direction select signal (+) or (–) for the axis.

3 Positioning is made at the grid point located nearest from the current point to reference position return direction. This point is recorded as the reference position. If the absolute-position detector is provided, the set reference position is retained after the power is turned off. In this case, when the power is turned on again, there is no need for setting the reference position again.

- **Reference position return**

1 After the reference position is set, when the feed axis select signal (+) or (–) is input for the axis in the reference position return mode, reference position return operation is performed in rapid traverse regardless of which signal (+) or (–) is input.

6.3 AUTOMATIC REFERENCE POSITION RETURN (G28, G29)

M series

- **Return to reference position (G28)**

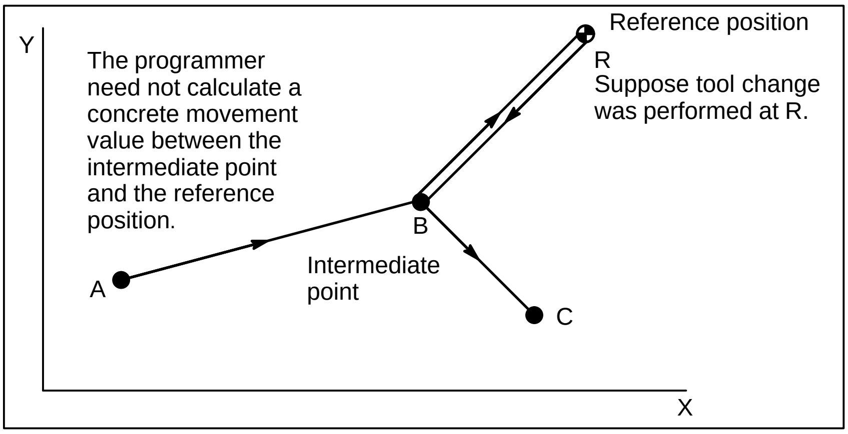
With the G28 command, the commanded axis is positioned to the reference position via the commanded point. After positioning, the reference position return end lamp lights. If G28 was commanded when reference position return is not performed after power on, reference position return is done in the same sequence as the manual reference position return.

G28 IP _ ;
IP : Command intermediate point

- **Return from reference position (G29) (M series)**

With the G29 command, the commanded axis is positioned to the point commanded by G29, via the intermediate point commanded by G28.

G29 IP _ ;



Example of use of G28 and G29

6.4 REFERENCE POSITION RETURN CHECK (G27)

This function is used to check whether the reference position return command was performed correctly.
When G27 is commanded, the commanded axis is positioned to the specified position, reference position return end signal is output if reference position return is performed to the correct position, and alarm arises it is not positioned correctly to the reference point.
This function is available after power is turned on an reference point return is performed.

Format

G27 IP _ ;

6.5 2ND, 3RD AND 4TH REFERENCE POSITION RETURN (G30)

With the G30 command, the commanded axis is positioned to the 2nd, 3rd, or the 4th reference position, via the commanded point. 2nd, 3rd, or 4th reference position return end signal is output when positioning ends. Set the 2nd, 3rd, and 4th reference position position as parameters.
This function is available after power is turned on and reference position return is performed.
G29 can be used to return from the 2nd, 3rd, and 4th reference point (same as reference position return, G28) (M series only).
This function can be used once reference position return has been performed after power–on.

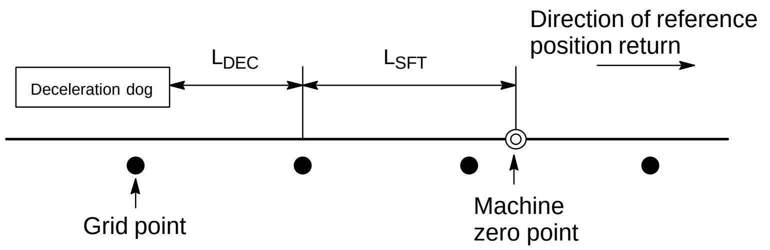
Format

$$\textbf{G30} \left\{ \begin{matrix} \textbf{P2} \\ \textbf{P3} \\ \textbf{P4} \end{matrix} \right\} \textbf{IP \_ ;}$$

P2, P3, P4: Select from 2nd, 3rd, or 4th reference positions. If not selected, 2nd reference position return is automatically selected.

6.6 REFERENCE POSITION SHIFT

For reference position return using the grid method, you can shift the reference position without having to move the deceleration dog, simply by setting the amount of shift in a parameter.
The time required to adjust the reference position is thus greatly reduced because the deceleration dog need not be adjusted.



L_{SFT} : Reference position shift
L_{DEC} : Distance the tool moves between the deceleration dog being turned off and the first grid point encountered subsequently (grid point when the reference position shift is 0) (L_{DEC} is displayed on the diagnostic screen.)

6.7 REFERENCE POSITION SETTING WITH MECHANICAL STOPPER

The reference position setting function with mechanical stopper automates the setting of a reference position by butting the tool against a mechanical stopper on an axis. This function is provided to eliminate the variations in reference position setting that arise when the procedure is performed by different operators, and to minimize the amount of work involved in making fine adjustments after reference position setting.
Select the axis for which the reference position is to be set, then perform cycle start. The following operations are performed automatically:

1. The torque (force) of the selected axis is reduced so that the butting feedrate is constant. The tool is butted against the mechanical stopper. Then, the tool is drawn back a parameter–set amount from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, then the tool is butted against the mechanical stopper. Then, the tool is drawn back a parameter–set amount from the mechanical stopper.
3. The point on the axis to which the tool is drawn back is set as the reference position.

6.8 LINEAR SCALE WITH ABSOLUTE ADDRESSING REFERENCE MARKS

The linear scale with absolute addressing reference marks has reference marks (one–rotation signals) at intervals that change at a constant rate. By determining the reference mark interval, the corresponding absolute position can be deduced. The CNC makes a small movement along an axis to measure the one–rotation signal interval, then calculates the absolute position. The reference position can be established without performing positioning to the reference position.

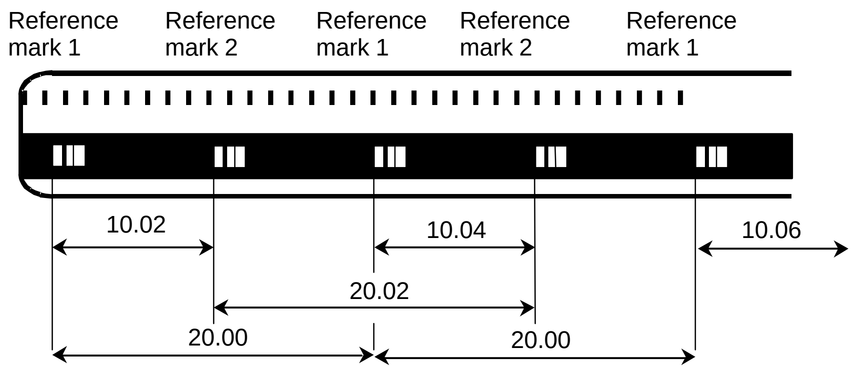


Fig. 6.8 Sample linear scale with absolute addressing reference marks

6.9 LINEAR SCALE EXPANSION FUNCTION WITH ABSOLUTE ADDRESSING REFERENCE MARKS

The linear scale expansion function with absolute addressing reference marks which is an optional function measures the reference mark interval automatically to establish a reference position when a G00 command or a jog–feed movement is specified.

7 COORDINATE SYSTEMS

By teaching the CNC the position the tool is to arrive, the CNC moves the tool to that position. The position is specified using coordinates on a certain coordinate system.
There are three types of coordinate systems.

- **Machine coordinate system**
- **Workpiece coordinate system**
- **Local coordinate system**

As necessary, one of the above coordinate systems is used for specifying coordinates for the target position of the tool.

7.1 MACHINE COORDINATE SYSTEM (G53)

Machine coordinate system is a coordinate system set with a zero point proper to the machine system.
A coordinate system in which the reference point becomes the parameter-preset coordinate value when manual reference point return is performed, is set. With G53 command, the machine coordinate system is selected and the axis is able to be moved in rapid traverse to the position expressed by the machine coordinates.

Format

G53 IP_ ;

• High–speed G53 function

When the function for overlap between rapid traverse blocks is enabled between a block containing a machine coordinate system selection command (G53) and a block containing a positioning (rapid traverse) command (G00), the rapid traverse command (G00) can be executed before the tool stops through deceleration at the end of the machine coordinate system selection command (G53). This can speed up positioning even when the machine coordinate system selection command (G53) is used.

Specifying P1 in the G53 block enables the high–speed G53 function.

Format

G53 IP_ P1 ;

G53 : G code for selecting the machine coordinate system (00 group)
IP_ : End point dimension word
P1 : Enables the high–speed G53 function.

7.2 WORKPIECE COORDINATE SYSTEM

A coordinate system in which the zero point is set to a fixed point on the workpiece, to make programming simple.
A workpiece coordinate system may be set by using the following methods:

(1) Using G92 (G50 for T series with G code system A)

(2) Automatic setting

(3) Using G54 to G59

When (1) is used, a workpiece coordinate system is established using the numeric value programmed after G92.

When (2) is used, a workpiece coordinate system is automatically established upon a manual reference position return, as specified in a parameter.

When (3) is used, six workpiece coordinate systems must be set from the MDI panel in advance. The workpiece coordinate system to be used is selected by specifying a code selected from G54 to G59.

7.2.1 Setting a Workpiece Coordinate System (Using G92) (with G Code System A: G50)

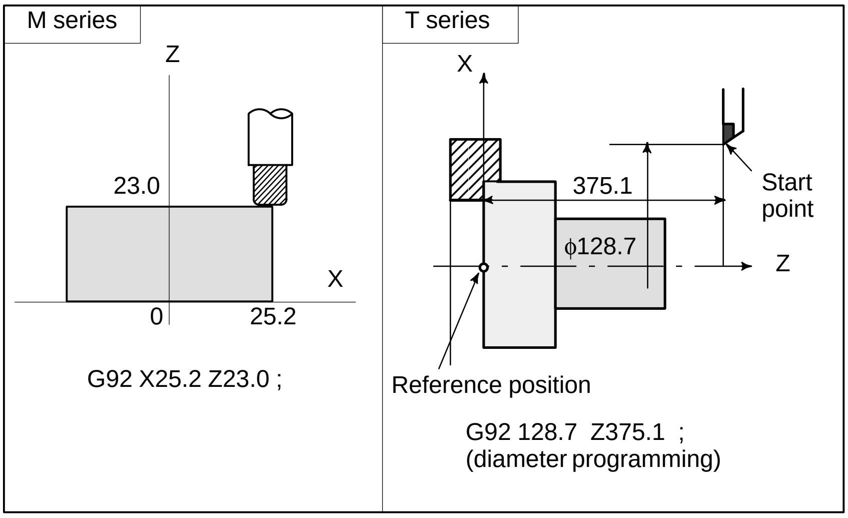
Format

(G90) G92 IP _ ;

Examples

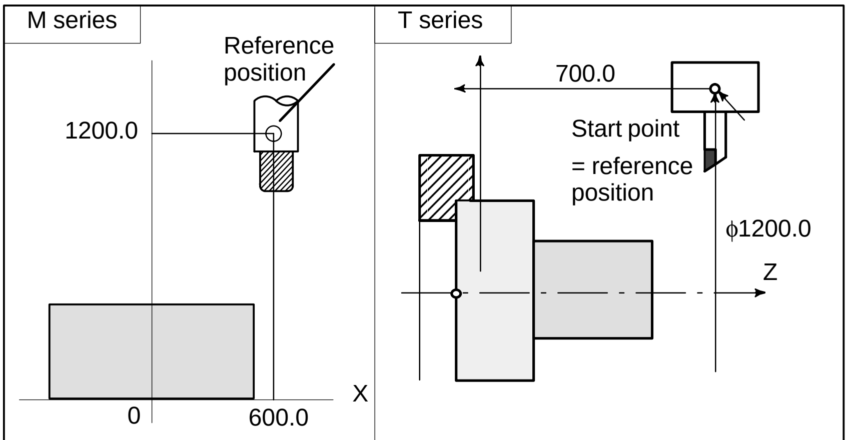
- **Example 1**

By using the above command, a workpiece coordinate system can be set so that the current tool position is at a specified position.



- **Example 2**

Set the reference position on the tool holder or turret as shown in the figure below, then specify G92 at the beginning of the program. By specifying an absolute command in this condition, the reference position is moved to a specified position. To move the tool tip to a specified position, compensate the distance between the reference position and the tool tip by using tool length compensation (for the M system) or tool offset (for the T system).



M series	T series
G92 600.0 Z1200.0 ;	G92 1200.0Z700.0 ; (diameter programming)
Setting the start point at a given reference position on the tool holder	Setting the start point at a given reference position on the turret

When a new workpiece coordinate system is created by specifying G92, it is determined so that a given point on the tool has a given coordinate value. So, there is no need to be concerned with old workpiece coordinate systems. Particularly when the start point for machining is determined based on the workpiece, the G92 command is useful. In this case, a desired coordinate system can newly be created even if an old workpiece coordinate system is invalid.

- **Example 3**

  (Shift of a workpiece coordinate system) (T series)

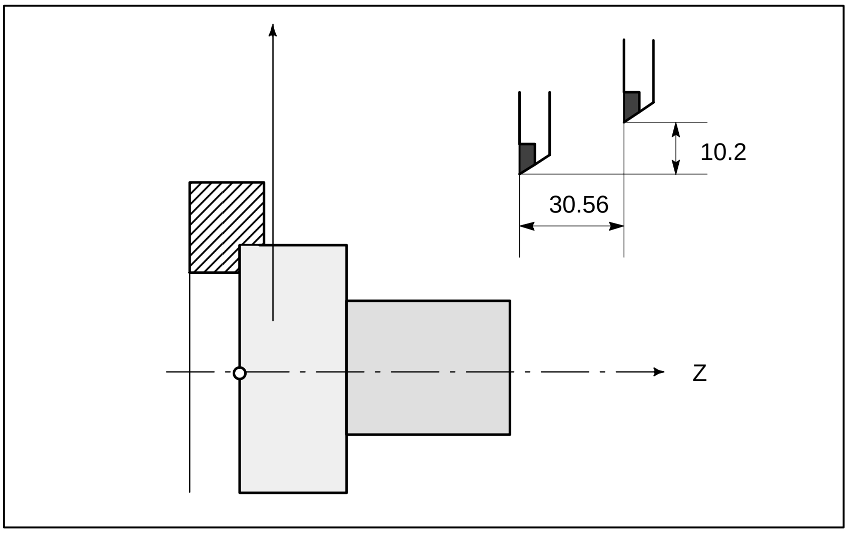
A workpiece coordinate system can be shifted by using the following command:
When this command is specified, a new coordinate system is created so that the current coordinate value (x, z) at a given point on the tool (for example, the tool tip) becomes (x+u, z+w).
For the x and u values, diameters must be set if diameter programming is specified, or radii must be set if radius programming is specified.

Format

(G91) G92 X(u) Z(w) ;

With G code system A: G50U(u)W(w);

Examples



When tool A is switched to tool B, G91 G92 X20.4 Z30.56 (diameter programming) is specified.

7.2.2 Automatic Coordinate System Setting

When manual reference position return is performed, a workpiece coordinate system can be set automatically. This functions as if G92IP__; were specified at the reference position.

7.2.3 Setting a Workpiece Coordinate System (Using G54 to G59)

Explanations

- **Setting a workpiece coordinate system**

Set six coordinate systems specific to the machine in advance. Then, select one of the six coordinate systems by using G54 to G59.

Format

$$\left\{\begin{array}{l}\textbf{G54}\\ \textbf{G55}\\ \textbf{G56}\\ \textbf{G57}\\ \textbf{G58}\\ \textbf{G59}\end{array}\right\}\ \mathrm{IP\_}\ ;$$

G54 Workpiece coordinate system 1
G55 Workpiece coordinate system 2
G56 Workpiece coordinate system 3
G57 Workpiece coordinate system 4
G58 Workpiece coordinate system 5
G59 Workpiece coordinate system 6

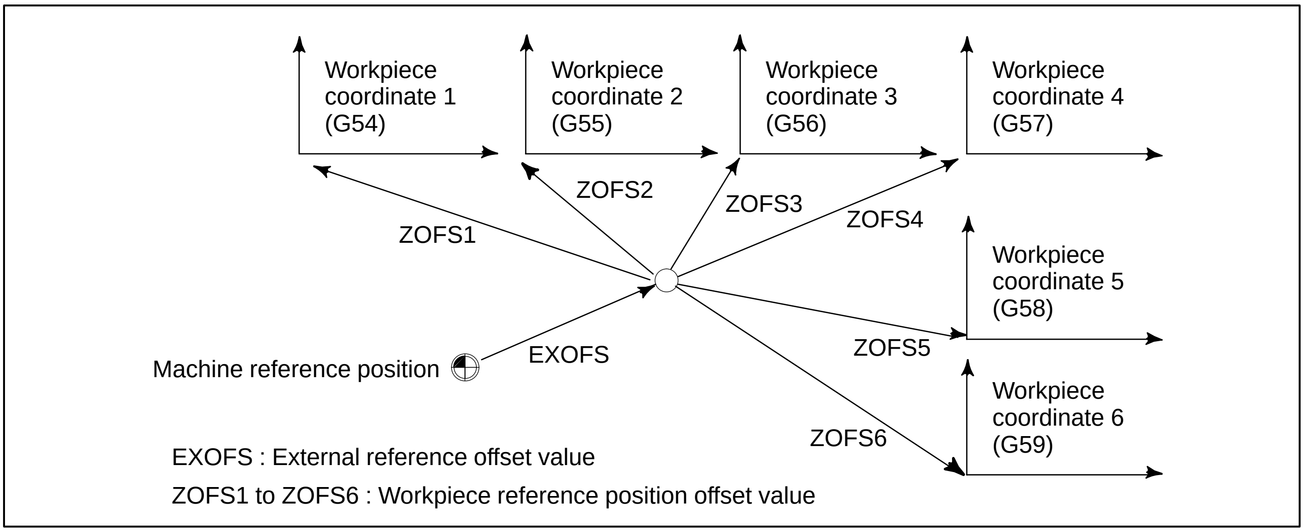
Set the distance between the machine zero point and the zero point of each of the six coordinate systems (offset from the workpiece zero point) in advance.
There are two setting methods.

- **Using the MDI**
- **Using a program (See Section 7.4.)**

Workpiece coordinate systems 1 to 6 are established properly when return to the reference position is performed after power is turned on. Immediately after power is turned on, G54 is selected.

- **Shift of workpiece coordinate systems**

The six workpiece coordinate systems can be shifted by a specified amount (external offset from the workpiece zero point).



7.2.4 Counter Input in a Workpiece Coordinate System

Explanations

- **Inputting counter values in a workpiece coordinate system**

On the workpiece coordinate system screen, when an axis address is specified, then the [INP.C.] soft key is pressed, the relative coordinate value for the specified axis is set at the cursor position as workpiece coordinate system data.

Examples

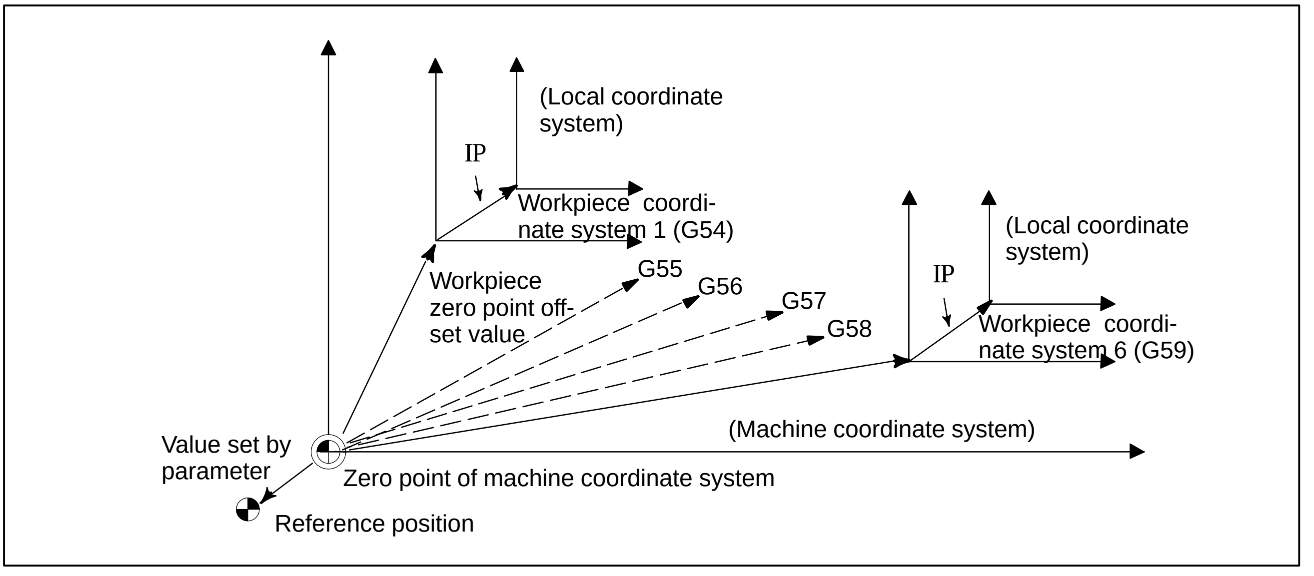
When the cursor is positioned on the X–axis of the offset from the workpiece zero point (G54) with the following relative coordinates:

[Relative coordinates]	[Workpiece coordinate system G54]		
X 100.000	X=	0.000	←Cursor position
Y 200.000	Y=	0.000	
Z 300.000	Z=	0.000	

(1) Inputting X then [INP.C.] sets X = 100.000.
(2) Inputting Y then [INP.C.] sets X = 200.000.
(3) Inputting Z then [INP.C.] sets X = 300.000.

7.3 LOCAL COORDINATE SYSTEM (G52)

With G52 commanded, the local coordinate system with the commanded position as zero point can be set. Once the local coordinate system is set, values specified in subsequent move commands are regarded as coordinate values on that coordinate system. Coordinates once set is valid till a new G52 is commanded. This is used when, for example, programming of a part of the workpiece becomes easier if there is a zero point besides the workpiece coordinates' zero point.



Format

```
G52 IP_ ;
```

Explanations

When local coordinate system is set, local coordinate system 1 - 6, corresponding to workpiece coordinate system 1 - 6 is set. Distance between zero points are all the same preset value.
If G52 IP0; is commanded, local coordinate system is canceled.

7.4 WORKPIECE ORIGIN OFFSET VALUE CHANGE (PROGRAMMABLE DATA INPUT) (G10)

G10 command is used to change workpiece origin offsets.
When G10 is commanded in absolute command (G90), the commanded workpiece origin offsets becomes the new workpiece origin offsets, and when G10 is commanded in incremental command (G91), the currently set workpiece origin offsets plus the commanded workpiece origin offsets becomes the new workpiece offsets.

Format

G10 L2 Pp IP _ ;

p : Specification the external workpiece origin offset value

p=1–6 : Specifiration the workpiece origin offset value corresponded to workpiece coordinate systems 1–6

IP : Workpiece origin offset value

7.5 ADDITIONAL WORKPIECE COORDINATE SYSTEMS

M series

Forty-eight workpiece coordinate systems can be added when existing six workpiece coordinate systems (G54 - G59) are not enough for the operation. Make a command as follows for selection of workpiece coordinate system.

Format

G54.1 Pp IP _ ; or **G54 Pp IP _ ;**

P: 1-48 or 1–300 Number of the additional workpiece coordinate system

The following are the methods of setting and changing of the workpiece origin offset value as well as those used for the existing workpiece coordinate systems of G54 to G59.

- **Method via MDI**
- **Method via program**
  - ❑ G10L20Pp;
  - ❑ Custom macro

7.6 WORKPIECE COORDINATE SYSTEM PRESET (G92.1)

The workpiece coordinate system with its zero position away by the workpiece zero offset amount from the machine coordinate system zero position is set by returning the tool to the reference point by a manual operation. Also, when the absolute position detector is provided, the workpiece coordinate system is automatically set by reading the machine coordinate value from the detector when power on without performing manual reference point return operation. The set workpiece coordinate may shift by any of the following commands or operation:

- When manual interruption is performed with the manual absolute signal off
- When the travel command is performed by the machine lock
- When axis travel is performed by the handle interrupt or auto/manual simultaneous operation
- When operation is performed by mirror image
- When the setting of local coordinate system is performed by the G52 or change of workpiece coordinate system is performed by the G82

The workpiece coordinate system shifted by the above operation can be preset by the G code instruction or MDI operation the same as conventional manual reference point return.

Explanations

- **Workpiece coordinate system preset by G code command**

The workpiece coordinate system can be preset by commanding the

G92.1 IP 0 ;

IP 0 : The axis address to be preset the workpiece coordinate system
Uncommanded axis is not preset.

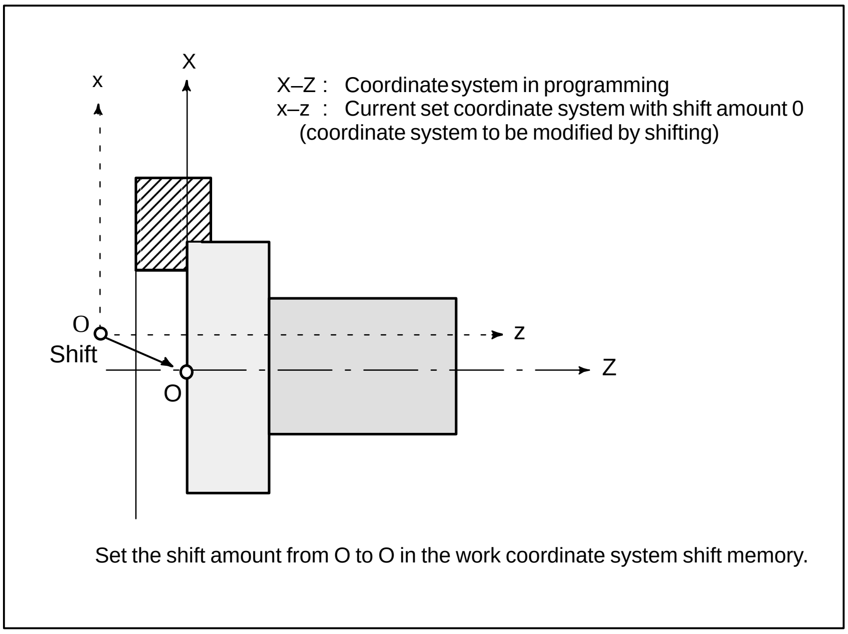
- **Workpiece coordinate system preset by MDI operation**

The workpiece coordinate system can be preset by the MDI operation with soft keys.

7.7 WORKPIECE COORDINATE SYSTEM SHIFT

T series

When the coordinate system actually set by the G50 command or the automatic system settingdeviates from the programmed work system,the set coordinate system can be shifted.
Set the desired shift amount in the workpiece coordinates system shift memory.



Workpiece coordinate system shift

7.8 PLANE SELECTION (G17, G18, G19)

A plane subject to circular interpolation, cutter compensation, coordinate system rotation, or drilling can be selected by specifying a G code.

G code	Selected plane	Xp	Yp	Zp
G17	Xp–Yp plane	X axis or an axis parallel to the X axis	Y axis or an axis parallel to the Y axis	Z axis or an axis parallel to the Z axis
G18	Zp–Xp plane			
G19	Yp–Zp plane			

Explanations

One of the existing parallel axes is determined by an axis address that appears in the block for which G17, G18, or G19 is specified.

- **Example 1**

When X and U, Y and V, and Z and W are parallel to each other, respectively

G17 X_ Y_ . . . . XY plane
G17 U_ Y_ . . . . UY plane
G18 X_ W_ . . . WX plane
G18 U_ W_ . . . WU plane

- **Example 2**

Planes remain unchanged in blocks for which G17, G18, or G19 is not specified.

G18 X_ Z_ . . . . ZX plane
X_ Y_ . . . . Plane not changed (ZX plane)

- **Example 3**

If G17, G18, or G19 is specified for a block, and no axis address is specified in that block, the axis addresses for the basic three axes are assumed to be omitted.

G17 . . . . XY plane
G17 X_ . . . . XY plane
G17 U_ . . . . UY plane

NOTE

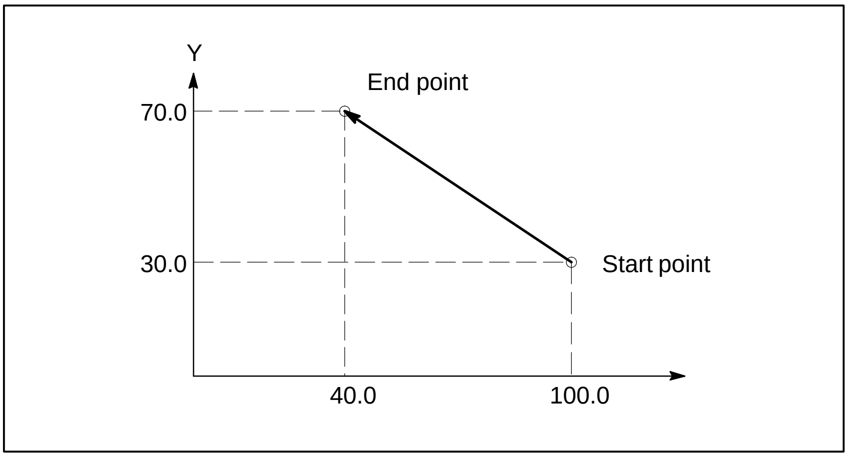
A parameter is used to specify which axis, X, Y, or Z the additional axis is parallel to. The move command functions regardless of the plane selection.
For example, suppose that the following is specified:
G17 Z_ ;
Axis Z does not exist on the XpYp plane. The XY plane is just selected, and the Z axis is moved regardless of the plane.

8 COORDINATE VALUE AND DIMENSION

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

There are two ways to command travels to the axes; the absolute command, and the incremental command. In the absolute command, coordinate value of the end point is programmed; in the incremental command, move distance of the axis itself is programmed.
G90 and G91 are used to command absolute or incremental command.

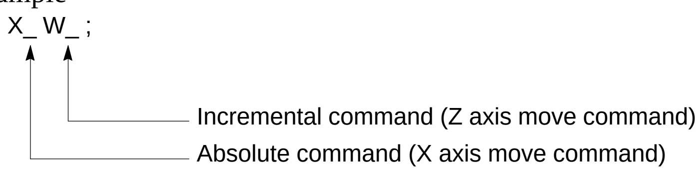
G90 : Absolute command
G91 : Incremental command



For the above figure, incremental command programming results in:
G91 X–60.0 Y40.0 ;
while absolute command programming results in:
G90 X40.0 Y70.0 ;
Absolute/incremental command, when G code system A at T series is selected, is not distinguished by G90/G91 but is distinguished by the address word.
For the A and B axes, no incremental commands are provided.

Absolute command	Incremental command	Notes
X	U	X axis move command
Z	W	Z axis move command
Y	V	Y axis move command
C	H	C axis move command
A	None	A axis move command
B	None	B axis move command

Example



8.2 POLAR COORDINATE COMMAND (G15, G16)

M series

The end point coordinate value can be input in polar coordinates (radius and angle). Use G15, G16 for polar coordinates command.

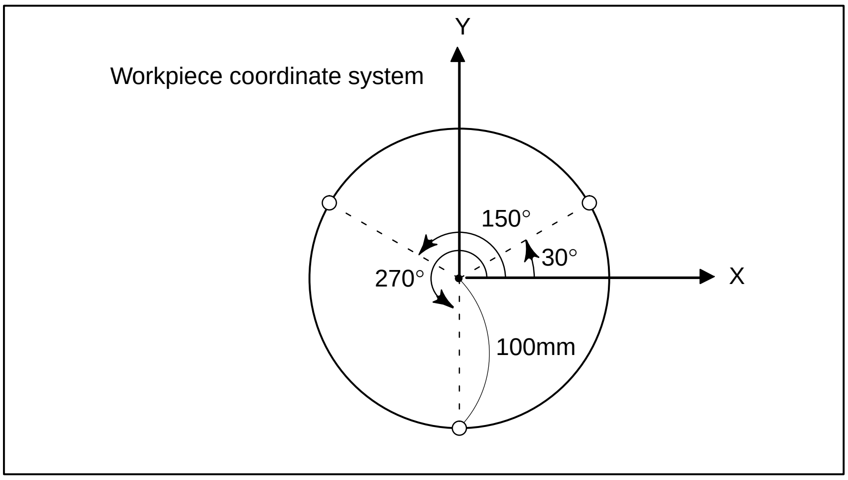
G15 : Polar coordinate system command cancel
G16 : Polar coordinate system command

Plane selection of the polar coordinates is done same as plane selection in circular interpolation, using G17, G18, G19.
Command radius in the first axis of the selected plane, and angle in the second axis. For example, when the X-Y plane is selected, command radius with address X, and angle with address Y. The plus direction of the angle is counter clockwise direction of the selected plane first axis + direction, and the minus direction the clockwise direction.
Both radius and angle can be commanded in either absolute or incremental command (G90, G91).
The center of the polar coordinates is the zero point of the workpiece coordinates. (However, if the local coordinates are set, it is the zero point of the local coordinates.)

Examples

● **Both hole cycle**

N1 G17 G90 G16;
Polar coordinates command, X-Y plane
N2 G81 X100. Y30. Z-20. R-5. F200.;
100mm radius, 30° angle
N3 X100. Y150;
100mm radius, 150° angle
N4 X100. Y270;
100mm radius, 270° angle
N5 G15 G80;
Polar coordinates cancel



8.3 INCH/METRIC CONVERSION (G20, G21)

Conversion of inch and metric input can be commanded by the G code command.

G20 : Inch input
G21 : Metric input

Whether the output is in inch system or metric system is parameter-set when the machine is installed.
Command G20, G21 at the head of the program. (This specification must be performed at the reference position.)
Inch/metric conversation can also be done by MDI setting.
The contents of setting data differs depending on whether G20 or G21 is commanded.

8.4 DECIMAL POINT INPUT/POCKET CALCULATOR TYPE DECIMAL POINT INPUT

Numerals can be input with decimal points. Decimal points can be used basically in numerals with units of distance, speed, and angle. The position of the decimal point is at the mm, inch, deg position.
There are two types of decimal point notation: calculator–type notation and standard notation.
When calculator–type decimal notation is used, a value without decimal point is considered to be specified in millimeters, inch or deg. When standard decimal notation is used, such a value is considered to be specified in least input increments.
Use parameters to select input method; whether to input by pocket calculator type input, or by the usual decimal point input.
Values can be specified both with and without decimal point in a single program.

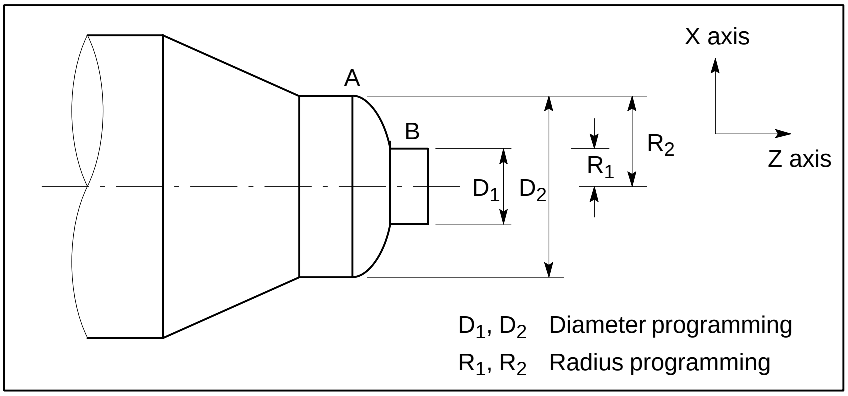
Program command	Pocket calculator type decimal point programming	Usual decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

8.5 DIAMETER AND RADIUS PROGRAMMING

T series

Since the work cross section is usually circular in latches, its dimensions can be specified in two ways when performing a thing:

When the diameter is specified, it is called diameter programming, and when the radius is specified, it is called radius programming.
The diameter programming or radius programming can be selected by parameter for each axis.



8.6 LINEAR AXIS AND ROTATION AXIS

A linear axis refers to an axis moving linearly, and for it values are specified in mm or inches.
A rotation axis refers to a rotating axis, and for it values are specified in degrees.
For rotation axes, note the following:

- Inch-metric switching is not performed.
- The machine coordinate system is always normalized to the range from 0deg to 360deg.

8.7 ROTARY AXIS ROLL-OVER FUNCTION

The rotation axis roll-over function rounds the absolute coordinate value and relative coordinate value of a rotation axis to a coordinate value within one rotation. This prevents coordinate values to overflow.
In an incremental command, the specified value is regarded as the amount of travel.
In an absolute command, the specified value is rounded to within one rotation. The resulting coordinate value is used as the end point. A parameter is used to specify whether to determine the move direction by the sign of the specified value or by the move distance (the shortest move distance to the end point is selected).

9 SPINDLE SPEED FUNCTIONS

9.1 S CODE OUTPUT

Specify the spindle speed with up to five digits immediately after address S. The 5-digit numeric value is output to the PMC as a 32-bit binary code. The code is maintained until another S is specified. The maximum number of input digits for S can be specified using a parameter.

9.2 SPINDLE SPEED ANALOG OUTPUT (S ANALOG OUTPUT)

The speed of the analog interface spindle is controlled. Specify the spindle speed with up to five digits immediately after address S. According to the specified spindle speed, a speed command is output to the spindle motor in a form of analog voltage. During constant surface speed control, an analog voltage is output so that it matches the spindle speed reached after constant surface speed control.

9.3 SPINDLE SPEED SERIAL OUTPUT (S SERIAL OUTPUT)

The speed of the serial interface spindle is controlled.
Specify the spindle speed with up to five digits immediately after address S. A speed command is output to the spindle motor according to the specified spindle speed. During constant surface speed control, a speed command is output so that it matches the spindle speed reached after constant surface speed control.

9.4 SPINDLE OUTPUT CONTROL BY THE PMC

If a speed command for the spindle motor is input in a form of [sign + 12-bit binary code], the command is output to the spindle motor according to the input.

9.5 CONSTANT SURFACE SPEED CONTROL

Whether to perform constant surface speed control is specified using G96 or G97.

G96 : Constant surface speed control mode
G97 : Constant surface speed control cancel mode

If the surface speed is specified with an S code (S followed by a numeric value) in the constant surface speed control mode, the spindle speed is controlled so that a constant surface speed can be maintained while the tool position is changing.

The axis on which the calculation for constant surface speed control is based can be specified with either a parameter or the following command:

G96 Pα ; P0 : Axis specified with a parameter
Pα : αth axis (α = 1 to 8)

The specifiable range of the S code is as follows:

1 to 99999 m/min or feet/min

In the constant surface speed control cancel mode, the spindle speed is specified using an S code.

In the constant surface speed control mode, a constant surface speed control on signal is output.

By specifying the following command, the maximum spindle speed can be set:

G50 S_ ; (with the G code system A in T series)
or
G92 S_ ; (where, S-- is the maximum spindle speed in min^{-1})

The spindle speed is clamped when it reaches the specified maximum spindle speed.

A machine that does not have (or which does not use) a position coder cannot perform feed per rotation under constant surface speed control.

9.6 SPINDLE OVERRIDE

To the spindle speed specified by S, an override from 0% to 254% can be applied (in steps of 1%).

9.7 ACTUAL SPINDLE SPEED OUTPUT

T series

Actual spindle speed calculated by the return pulses of the position coder on the spindle is output in 16-bit binary code.

9.8 SPINDLE POSITIONING

T series

In turning operation, the spindle connected to the spindle motor rotates at a certain speed, and the workpiece attached to the spindle is then turned. The spindle positioning function moves the spindle connected to the spindle motor by a given angle so that the workpiece attached to the spindle is positioned at a desired angle.
With this function, any portion of the workpiece can be drilled.
The spindle position is detected by the position corder attached to the spindle.
Whether to use the spindle for spindle positioning (spindle positioning mode) or to use the spindle for spindle rotation (spindle rotation mode) is command by special M code (set by parameters).

- **Move command**
  When commanded:
  G00 C_ ;,
  The spindle is positioned to the commanded position by rapid traverse. Absolute (G90) and incremental (G91) command, as well as decimal point input is possible.
- **Increment system**
  Least input increment: 0.001 deg.
  Detection unit: (360×N)/4096 deg.
  N: Combination ratio of position coder and spindle (N=1, 2, 4)

9.9 SPINDLE SPEED FLUCTUATION DETECTION (G25, G26)

This function monitor spindle speed, detects a higher level of fluctuation than the commanded speed and signals an abnormality, if any, to the machine side, using an alarm, thereby preventing the spindle from seizure, for example. Whether the spindle speed fluctuation detection is done or not is specified by G code.

G25 : Spindle speed fluctuation detection is off.
G26 : Spindle speed fluctuation detection is on.

Format

G26 P_ Q_ R_ ;

P_ : Time from the change of spindle speed to the start of the spindle speed fluctuation detection (Unit: msec)

Q_ : The ratio of spindle speed to the specified spindle speed where spindle speed fluctuation detection starts (Unit: %)

R_ : Fluctuation ratio regarded as an alarm (Unit: %)

NOTE

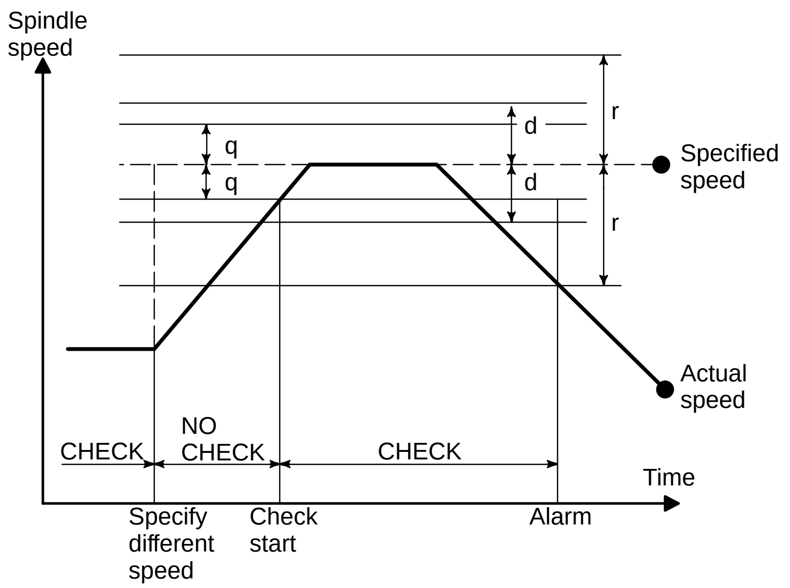
1 The value of P, Q, and R remains after the power off.
2 The actual spindle speed is calculated by the return pulses generated from the position coder attached to the spindle.

Explanations

There are two ways in generating an alarm:

An alarm is generated before the specified spindle speed reaches.
An alarm is generated after the specified spindle speed reaches.

- **When an alarm is generated after the spindle speed becomes the commanded speed.**



Commanded speed : (Speed commanded by S) x (Spindle override)

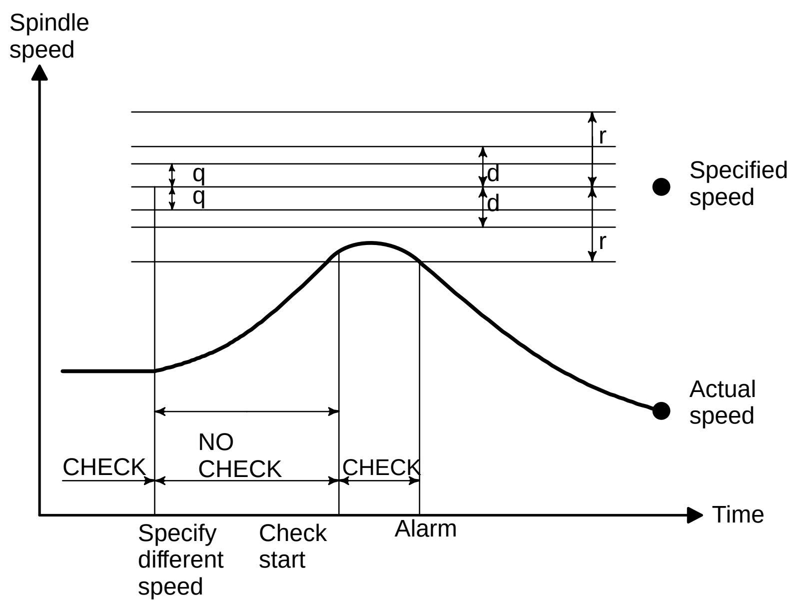
Actual speed : Speed detected by position coder

q : (Allowable rate for starting checkup) x (Commanded speed)

r : (Fluctuation rate in which an alarm is given) x (Commanded speed)

d : Fluctuation width in which an alarm is given Cutting by parameter

- **When an alarm is generated before the spindle speed becomes the commanded speed.**



p : Time between changes in commanded speed and check start.

9.10 Cs CONTOUR CONTROL

The serial interface spindle permits positioning and linear interpolation with another servo axis. Thus, linear interpolation between the spindle and a servo axis can be specified.

Explanations

• Control mode

The serial interface spindle has two modes.

- ❑ The spindle rotation control mode controls the speed of the spindle. (The spindle is rotated according to a speed command.)
- ❑ The spindle contour control mode (also called Cs contour control) controls the position of the spindle. (The spindle is rotated according to a move command.)

These modes are switched by a signal sent from the PMC.

• Spindle contour control axis

The axis subject to spindle contour control is placed as one of the CNC control axes. Any of the control axes can be selected as the spindle contour control axis.

• Move command

In manual and automatic operation, a move command for the spindle contour control axis is programmed in the same way as for a servo axis.

Example) Let the name of the spindle contour control axis be C.
G00 C30.0 ; (Positioning)
G01 X100.0 Y100.0 C90.0 F1000.0 ; (Linear interpolation)

• Automatic loop gain setting when switching between spindle rotation control and spindle contour control is made

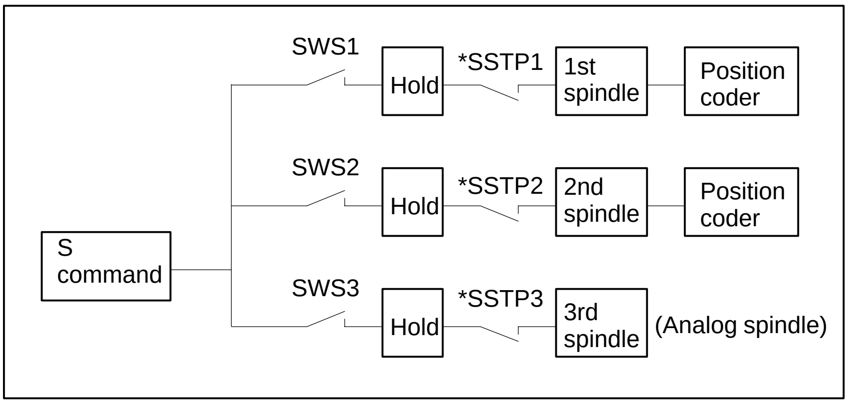
❑ **Switching from spindle rotation control to spindle contour control**
If there is a difference in servo loop gain between the axis subject to spindle contour control and the other servo axes, linear interpolation with the spindle contour control axis cannot be performed properly.
As soon as spindle rotation control is switched to spindle contour control, an appropriate spindle contour control servo loop gain for a selected gear is automatically set for a necessary servo axis. The axis for which the servo loop gain is to be changed, and the spindle contour control servo loop gain for this axis must be set in parameters for each gear beforehand.

❑ **Switching from spindle contour control to spindle rotation control**
As soon as spindle contour control is switched to spindle rotation control, the original servo loop gain is set for the servo axis automatically.

9.11 MULTI–SPINDLE CONTROL

Up to two spindles can be controlled. The three spindles are called the first and second spindles. The first and second spindles are made up of serial interface spindles, and the third spindle is of an analog interface spindle. (However, the second and third spindles must be selected.)
A spindle speed is specified with a 5-digit numeric value following S. This command functions on the spindle selected by spindle selection signals (SWS1 to SWS3). More than one spindle can be selected so that they can be rotated at the same time by specifying the same command.
Each spindle holds a specified command (spindle speed). When the spindle is not selected by the spindle selection signal, the spindle rotates at the held spindle speed. By using this feature, the spindles can be rotated at different speeds at the same time. For each spindle, a signal to stop spindle rotation is provided (*SSTP1 to *SSTP3). With these signals, unnecessary spindles can be placed in the stopped state.
Feedback pulses from the position coders connected to the first and second spindles can be input to the CNC to perform threading and feed per rotation. One of the position coders connected to the first and second spindles is selected by a signal. The feedback pulse from the selected coder is then input to the CNC. From the third spindle, no feedback pulse can be input.
The multi–spindle control functions of the M and T series differ as follows:

- For the M series, multi–spindle control is possible only when spindle gear selection type T is specified.
- For the M series, rigid tapping spindle selection signals (RGTSP1, RGTSP2, and RGTSP3) cannot be used.



* The second and third spindles must be selected.

9.12 SPINDLE SYNCHRONOUS CONTROL

In machine tools having two spindles (such as a lathe), the speeds of the two spindles sometimes have to match. This requires when a workpiece held on the first spindle is transferred to the second spindle while the spindles are rotating, and when acceleration/deceleration is performed while a workpiece is being held by the first and second spindles.
When a workpiece having a different figure is transferred between the spindles, the rotation phases (rotation angles) of the spindles must also match.
The serial interface spindle synchronous control function is provided to provide synchronous control for two spindles.

9.13 SPINDLE ORIENTATION

You can perform spindle orientation simply by mounting a position coder on the spindle. Stoppers or pins for physically stopping the spindle at a specified position are not necessary. A spindle can be instantly oriented, even when rotating at high speed, thereby greatly reducing the orientation time.

9.14 SPINDLE OUTPUT SWITCHING

Spindle output switching switches between the two windings, one for low speed and the other for high speed, incorporated into the special spindle motors. This ensures that the spindle motor demonstrates stable output characteristics over a wide range.

10 TOOL FUNCTIONS

10.1 T CODE OUTPUT

M series

A tool can be selected by specifying a tool number of up to eight digits immediately after address T. The tool number is output to the PMC in a 32-bit binary code. This code is kept till the next T code is commanded. Maximum input digits are set by parameters.

T series

A tool and offset can be selected by specifying a tool number and offset number of up to eight digits (in total) immediately after address T. The offset number is specified with the last one or two digits of the T code. The tool number is specified with the remaining digits after excluding the one or two digits used to specify the offset number.

When the last one digit is used to specify the offset number:

T ○○○○○○○☆

Offset number

Tool number

When the last two digits are used to specify the offset number:

T ○○○○○○☆☆

Offset number

Tool number

The tool number is output in a 32-bit binary code. This code is kept till the next T code is commanded. Maximum input digits are set by parameters.

10.2 TOOL LIFE MANAGEMENT

10.2.1 Tool Life Management

Tools are classified into groups, and tool life (hours and times of use) is set for each group. When use of the tool exceeds the preset hours or times of use, another tool in the same group which has not yet exceeded the preset life time is selected. If all the tool in a group exceeds the preset life time, a signal is output to inform the operator that the tools must be changed to new tools. With setting the cutter radius compensation number and the tool length compensation number of the tools, compensation corresponding to each tool can also be done. (M series)
With use of this function Factory Automation (FA) comes to a reach.
This function has the following features:

- Tool life can be set in hours or times of use.
- **New tool select signal output**
  This signal is output when a new tool is selected in a group. This can also be used for automatic measurement in compensations of the new tools.
- **Tool change signal**
  When all the tools of a group has exceeded their life time, this signal is output to inform the operator.
- **Tool skip signal**
  By inputting this signal, tools still not exceeding their life time, can also be changed.
- **Tool life management data is display/modification**
  Tool life management data is displayed on the cRT screen, informing the operator of the condition of the tools at a single view. If necessary, the counter value of tool life can be modified via the MDI panel.
  Number of groups and number of tools per group is selected by parameter from the following.

M series		T series	
Number of groups	Number of tools	Number of groups	Number of tools
16	16	16	16
32	8	32	8
64	4	64	4
128	2	――	――

10.2.2 Extended Tool Life Management

M series

The following features are added to the tool life management function for easier handling:

- **Setting tool life management data for each tool group by program**
  Addition, modification, and deletion can be made to only the tool life management data of a specified group; the tool life management data of the other groups is left intact.
  A tool life is set for each tool group by time or use count.
- **Displaying and editing tool life management data**
  All tool life management data is displayed on the screen, and so the user can understand the current tool state instantly. The following data items are displayed:
  - Tool group number of the tool currently used
  - Tool group number selected next
  - Tool life management data
    - Life, life counter value
    - List of tool numbers in the group
    - Cutter compensation number and tool length compensation corresponding to each tool number
    - Use state for each tool (for example, indicating whether tool life is reached)

  Tool life management data can be modified at the MDI panel. In addition, tool numbers can be added, changed, and deleted.
- **Life count override**
  If a tool life is set by time, actual time obtained by multiplying the use time of a tool by a magnification (override value) can be added to the life counter. An override value from 0 to 99.9 is specified in steps of 0.1 by a signal sent from the PMC.

  Example)
  If the override value is 0.1, and the use time of a tool is ten minutes, the life counter is incremented by one minute.
- **Tool life notice signal**
  When the rest of the tool life (remainder value) until a new tool is selected is set as a value common to all groups, a signal is output to the PMC when the value obtained by the subtraction (the life value (LIFE) minus the counter value (COUNT)) has reached the set remainder value.

11 AUXILIARY FUNCTIONS

11.1 AUXILIARY FUNCTIONS

When up to eight digits immediately after address M are specified, a 32–bit binary code is output. The maximum number of input digits can be specified with a parameter. This binary code is used for on/off control of the machine. A block can usually contain up to three M codes although only one of them is effective.
The following M codes are used for special purposes:

M00 : Program stop
M01 : Optional stop
M02 : End of program
M30 : End of program and tape rewind

The above M codes can also be output in binary codes.
M98 (sub program call) and M99 (return from sub program) and always processed in the CNC so, signal will not be output.

11.2 1–BLOCK PLURAL M COMMAND

Up to three M codes can be simultaneously specified in one block.
As these M codes are simultaneously sent to PMC side, the machining cycle time compared with the conventional 1-block single M command is reduced.

Example)

(i) **1-block single M command**

```
M40;
M50;
M60;
G28G91X0Y0Z0;
   :
```

(ii) **1-block plural M command**

```
M40M50M60;
G28G91X0Y0Z0;
   :
```

NOTE

1 The maximum input value of the first M code is 99999999, while the maximum input values of the second and third M codes are 65535.
2 A strobe signal is provided for each of the first to third M codes (MF, MF2, and MF3).
When all the operations for the first to third M codes are completed, completion signal FIN is output.

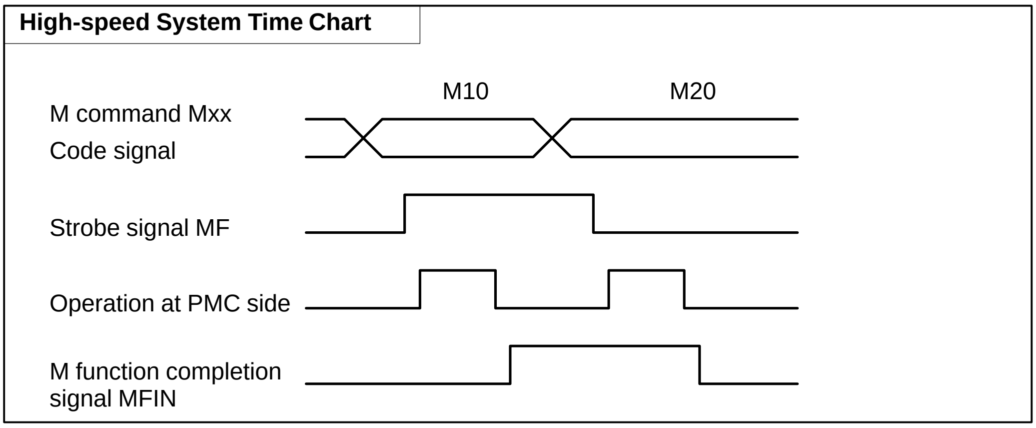
11.3 SECOND AUXILIARY FUNCTIONS

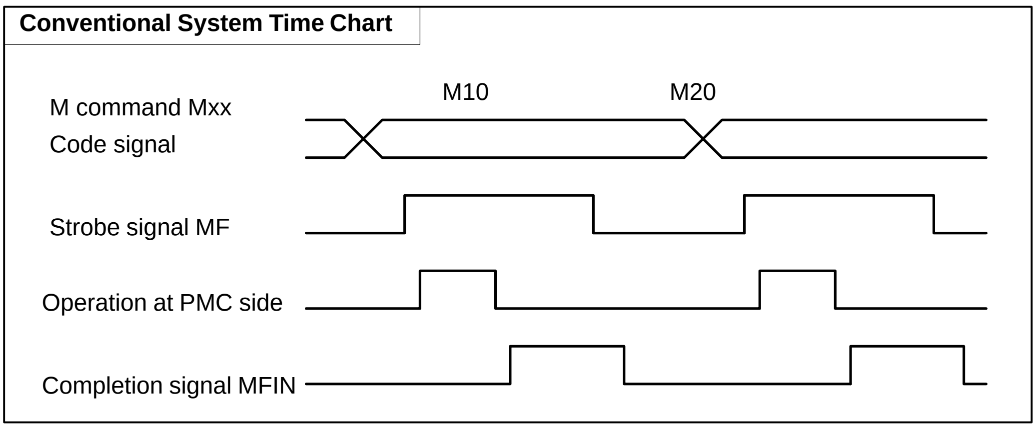
When an 8-digit number after address B is commanded, a 32-bit binary code is output to the PMC. This code is kept till the next B code is commanded.

11.4 HIGH-SPEED M/S/T/B INTERFACE

The communication of execution command signal (strobe signal) and completion signal is the M/S/T/B function were simplified to realize a high-speed execution of M/S/T/B function.
The time required for cutting can be minimized by speeding up the execution time of M/S/T/B function.
The following describes an example of auxiliary function M code command. The same applies to the T, S, and B (second auxiliary function) functions.
When an M code is specified, the CNC inverts the logical level of strobe signal MF. Thus, when the signal is 0, it becomes 1. When it is 1, it becomes 0. After inverting strobe signal MF, the CNC assumes the completion of PMC operation once the logical level of completion signal MFIN from PMC has become the same as the logical level of strobe signal MF.
In the usual system, if the leading edge (from “0” to “1”) of the completion signal FIN of M/S/T/B is received and then the trailing edge (from “1” to “0”) of the signal FIN is received, it is considered that the operation has been completed. However, in this system, the operation is considered to have been completed by a single change of completion signal MFIN.

Example) M10;
M20;





NOTE
1 Either the conventional system or the high-speed system can be selected for communication of strobe signal and completion signal.
2 In the conventional system, only one completion signal is available for all functions of M/S/T/B. However, in the high-speed system, one completion signal is available for each of M/S/T/B functions.

12 PROGRAM CONFIGURATION

12.1 PROGRAM NUMBER

A program number is given to each program to distinguish a program from other programs. The program number is given at the head of each program, with a 4-digit number after the address O.
Program number of the program currently under execution is always displayed on the CRT screen. Even during the execution of a sub program, the program number of the main program can also be displayed by parameter setting. Program search of programs registered in the memory is done with the program number. The program number can be used in various ways.

12.2 PROGRAM NAME

A program name can be given to the program to distinguish the program from other programs when displaying all the registered program on a screen. Register the name between the control-out and the control-in.
Any codes usable in the CNC can be used for the program name.
The program name is displayed with the program number in the directory display of registered programs. Note that the program name displayed is within 31 characters.

Example) 01234 (PROGRAM FOR ATC);

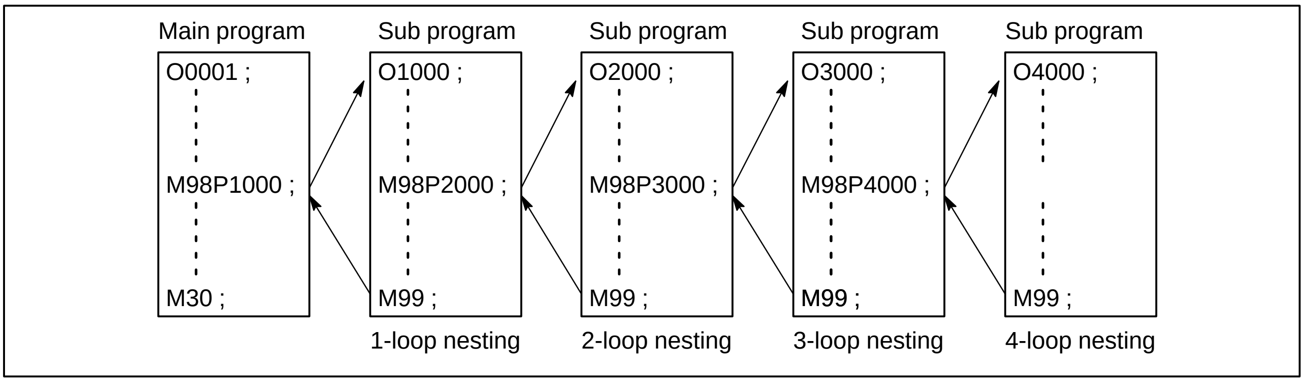
12.3 MAIN PROGRAM

A program is divided into the main program and the sub program. The CNC normally operates according to the main program, but when a command calling a sub program is encountered in the main program, control is passed to the sub program. When a command indicating to return to the main program is encountered in the sub program, control is returned to the main program.

12.4 SUB PROGRAM

When there are fixed sequences or frequently repeated patterns in a program, programming can be simplified by entering these pattern as sub programs to the memory. Sub program is called by M98, and M99 commands return from the sub program. The sub program can be nested 4 folds.

A sequence number in a sub program can also be specified for sub program call.



Format

Sub program call

M98 P ○○○ ○○○○ Q ○○○○○ ;

Number of repetitive calls — Subprogram number — Sequence number

If the number of repetitive calls is omitted, 1 is assumed.

When Q and a sequence number are omitted, the beginning of the sub program is called.

Return from sub program

M99 ;

12.5 EXTERNAL MEMORY AND SUB PROGRAM CALLING FUNCTION

When memory is used, a program cataloged in the floppy cassette can be called and executed as a sub program.
A sub program is called from the floppy cassette when the program using the memory executes the following block.

Format

Sub program calling

M198 P ○○○ ○○○○ ;

↑ Number of repetitive calls ↑ Subprogram number or file number

If the number of repetitive calls is omitted, 1 is assumed.

NOTE
1 Whether address P specifies the file number or program number is selected by a parameter.
2 In the program called by M198, no more sub program can be called by M198.

12.6 SEQUENCE NUMBER

Sequence number can be given in a 5-digit number after the address N at the head of the program block.
The sequence number of the program under execution is always displayed on the screen. The sequence number can also be searched in the program by the sequence number search function.

12.7 TAPE CODES

Either the EIA or the ISO code can be used as tape code. The input program code is distinguished with the first end of block code (EIA: CR, ISO: LF). See the List of Tape Codes for tape codes used.

12.8 BASIC ADDRESSES AND COMMAND VALUE RANGE

The following table shows the basic addresses and the range of values to be specified. The range, however, is that of CNC. Note that the range of the machine is different from this.

- **Basic Addresses and Range of Values to Be Specified (M series)**

Function		Address	Metric input	Inch input
Program number		O (Note1)	1–9999	1–9999
Sequence number		N	1–99999	1–99999
Preparatory function		G	0–999	0–999
Dimension word	IS–B	X, Y, Z, Q, R, I, J, K, A, B, C, U, V, W	±99999.999mm ±99999.999deg	±9999.9999inch(Note2) ±99999.999deg
	IS–C		±99999.9999mm ±99999.9999deg	±999.99999inch(Note2) ±9999.9999deg
Feed per minute	IS–B	F	1–240000mm/min	0.01–9600.00inch/min
	IS–C		1–100000mm/min	0.01–4000.00inch/min
Feed per rotation, Setting unit		F	0.01–500.00mm/rev	0.0001–9.9999inch/rev
Spindle function		S	0–20000	0–20000
Tool function		T	0–99999999	0–99999999
Miscellaneous function		M	0–99999999	0–99999999
		B	0–99999999	0–99999999
Dwell	IS–B	X, P	0–99999.999 (sec or rev)	0–99999.999 (sec or rev)
	IS–C		0–9999.9999 (sec or rev)	0–9999.9999 (sec or rev)
Program number specification		P	1–9999	1–9999
Number of subprogram repetitions		P	1–999	1–999
Offset number		H, D	0–400	0–400

- **Basic Addresses and Range of Values to Be Specified (T series)**

Function		Address	Metric input	Inch input
Program number		O (Note1)	1–9999	1–9999
Sequence number		N	1–99999	1–99999
Preparatory function		G	0–999	0–999
Dimension word	IS–B	X, Y, Z, U, V, W, A, B, C, I, J, K, R	±99999.999mm ±9999.9999deg	±9999.9999inch(Note2) ±99999.999deg
	IS–C		±99999.9999mm ±99999.9999deg	±999.99999inch(Note2) ±9999.9999deg
Feed per minute	IS–B	F	1–240000mm/min	0.01–9600.00inch/min
	IS–C		1–100000mm/min	0.01–4000.00inch/min
Feed per rotation, Screw lead		F	0.0001–500.00 mm/rev	0.000001–9.999999 inch/rev
Spindle function		S	0–20000	0–20000
Tool function		T	0–99999999	0–99999999
Miscellaneous function		M	0–99999999	0–99999999
		B	0–99999999	0–99999999
Dwell	IS–B	P, X, U	0–99999.999 (sec or rev)	0–99999.999 (sec or rev)
	IS–C		0–9999.9999 (sec or rev)	0–9999.9999 (sec or rev)
Program number specification		P	1–9999	1–9999
Number of subprogram repetitions		P	1–999	1–999
Sequence number specification		P, Q	1–99999	1–99999

NOTE
1 ":" can be used for 0 in ISO Code.
2 Coordinates maximum command value for inch input/metric output is limited to: ±3937.0078 inch (IS–B)/ ±393.70078 inch (IS–C).

12.9 TAPE FORMAT

The variable block word address format with decimal point is adopted as tape format. See List of Tape Format in Appendix C for details on tape formats.

12.10 LABEL SKIP

Label skip function is valid in the following cases, and "LSK" is displayed on the screen.

- When power is put on.
- When the NC is reset.

When label skip function is in valid, all codes to the first encountered end of block (EOB) code are ignored.
The ignored part is called "Reader part", and section after the first end of block (EOB) code, "significant information".

12.11 CONTROL-IN/ CONTROL-OUT

Information between the control-in and the control-out are regarded as notes and are ignored.
The reset codes (ISO code: %, EIA code: ER) cannot be used in this part.
The ignored part is called "Notes".

	ISO code	EIA code
Control–out	(	Channel 2–4–5 on
Control–in	)	Channel 2–4–7 on

12.12 OPTIONAL BLOCK SKIP

When a slash and number (/n) is programmed at the head of a program, and when the machine is operated with the optional block skip switch n on the machine operator's panel on, information in the block commanded with the /n corresponding to the switch number n is ignored.
If the optional block skip switch n is turned off, information in the /n commanded block will not be ignored. The block with /n commanded can be skipped by the operator's selection.
I can be used for n. The 1 to /1 can be omitted.
Example) /1 N12345 G00 X100.Z200.;

12.13 ADDITIONAL OPTIONAL BLOCK SKIP

2 to 9 can also be used for the n of the /n.

12.14 TAPE HORIZONTAL (TH) PARITY CHECK AND TAPE VERTICAL (TV) PARITY CHECK

A parity check is made on the number of punch holes for each input tape character. If the parity does not match, an alarm occurs (TH check). A parity check is made on each input data block. If the number of characters in one block (from the code next to EOB to another EOB) is odd, an alarm occurs (TV check). The TH or TV check cannot be made on the area skipped by the label skip function. The TH check is not made on the command field.A parameter can be used to specify whether the characters constituting comments are to be counted when obtaining the number of characters for TV check. The TV check function is validated or invalidated according to the value set on the MDI panel.

13 FUNCTIONS TO SIMPLIFY PROGRAMMING

13.1 CANNED CYCLES (G73, G74, G76, G80-G89, G98, G99)

M series

Canned cycle is a function to simplify commands for machining (boring, drilling, or tapping, etc. The canned cycle has the positioning plane and the drilling axis. The positioning plane is specified with the plane selection of G17, G18, and G19. The drilling axis is the basic axis X, Y or Z (that does not compose the positioning plane) or its parallel axis.

G code	Positioning plane	Drilling axis
G17	Xp–Yp plane	Xp
G18	Zp–Xp plane	Yp
G19	Yp–Zp plane	Zp

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

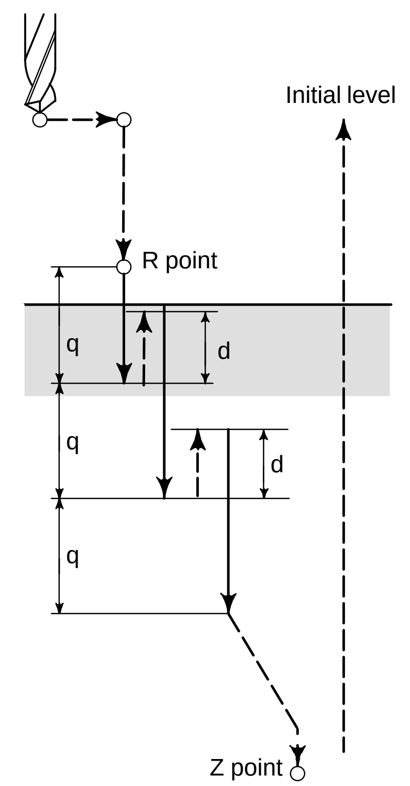
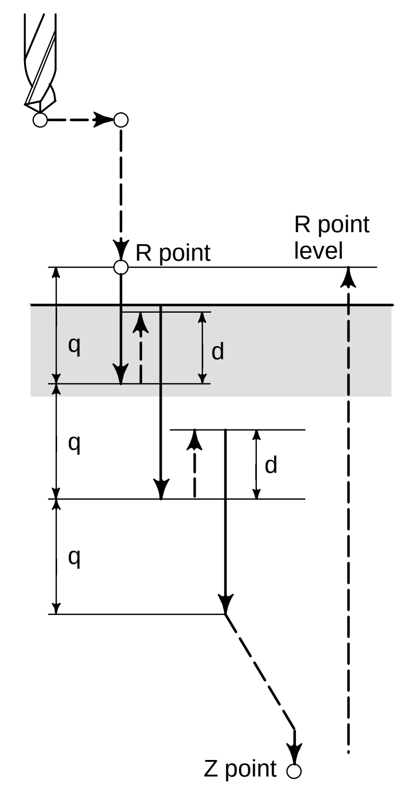
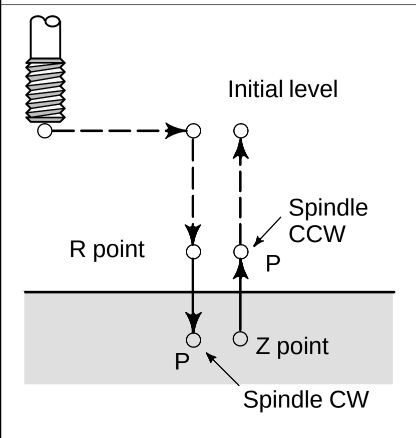
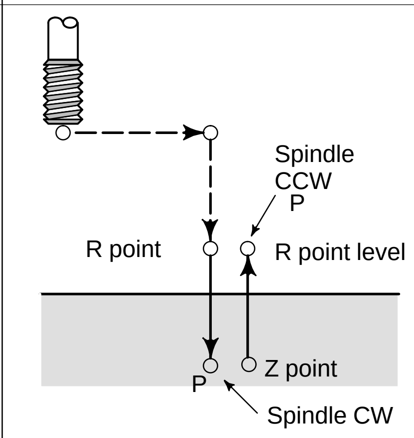
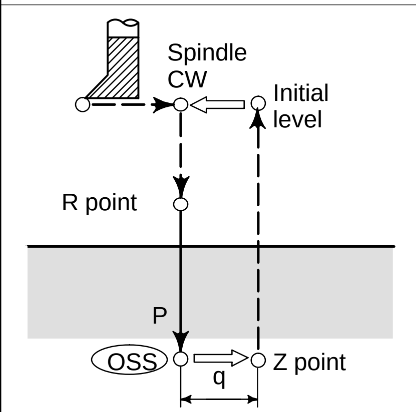
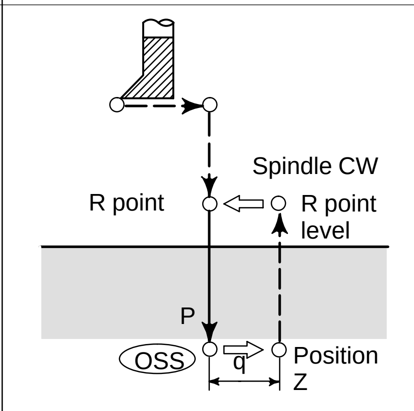
The drilling axis address commanded in the same block as the G codes, G73 - G89, decides whether the drilling axis is the basic axis or its parallel axis. If the drilling axis address was not commanded, the basic axis becomes the drilling axis.
Axis other than the drilling axis becomes the positioning axis.

Example)
When U, V, W axes are set as parallel axes for X, Y, Z axes respectively.
G17G81 . . . . Z_ ; Drilling axis is Z axis.
G17G81 . . . . W_; Drilling axis is W axis.
G18G81 . . . . Y_ ; Drilling axis is Y axis.
G18G81 . . . . V_ ; Drilling axis is V axis.
G19G81 . . . . X_ ; Drilling axis is X axis.
G19G81 . . . . U_ ; Drilling axis is U axis.
It is not always necessary to command G17, G18, G19 in the same block as G73 - G89.

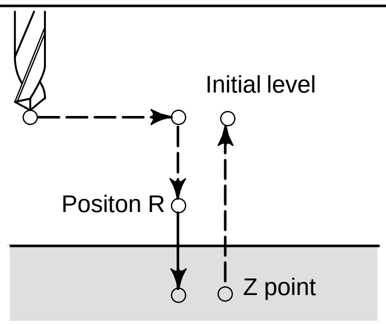
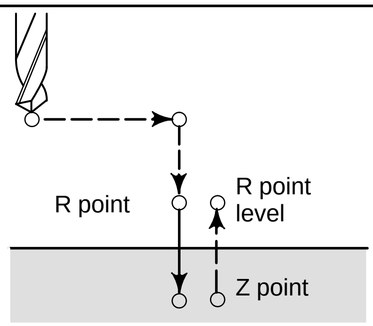
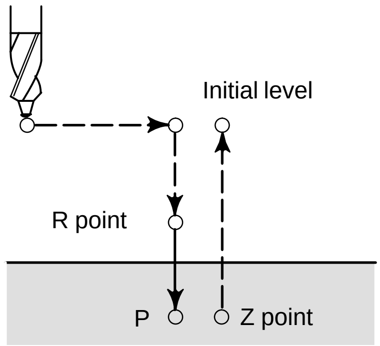
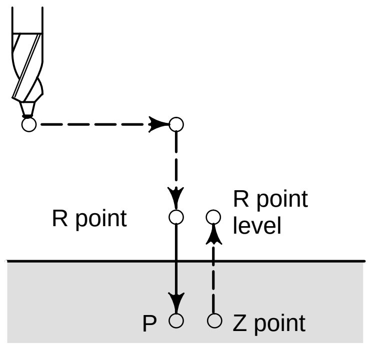
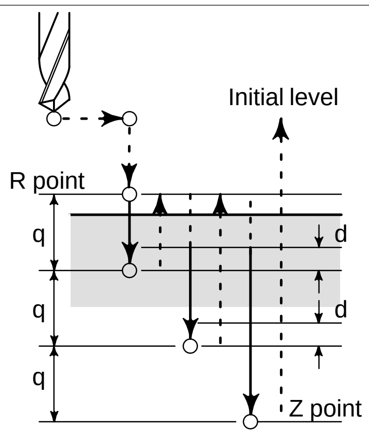
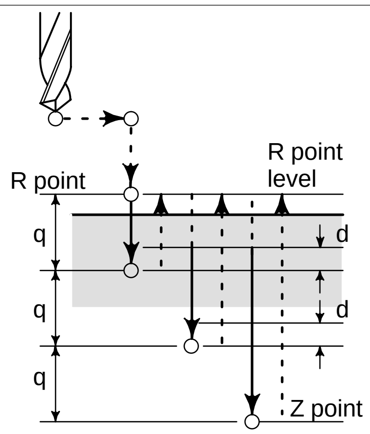
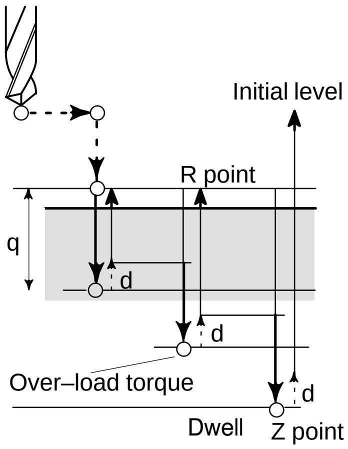
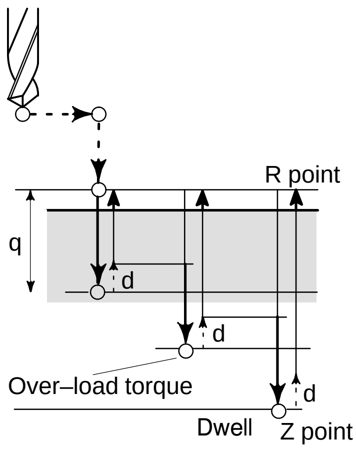
NOTE
Z axis can always be appointed the drilling axis by parameter setting.

Positioning can be commanded with optional axes other than the drilling axis. The drilling cycle starts after the positioning.
The following explanations are done on the XY plane, and Z axis as the drilling axis.
The following 13 types of canned cycles are available.

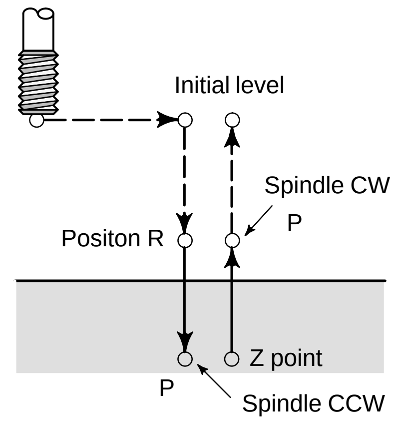
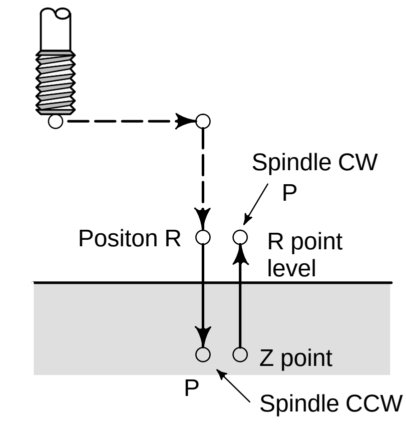
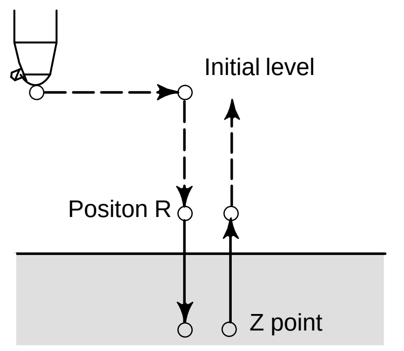
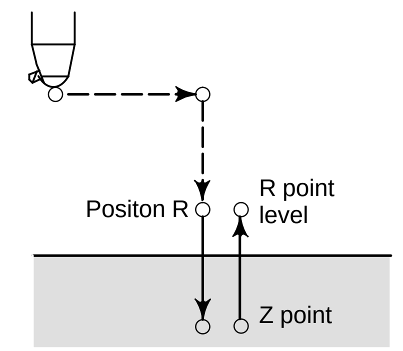
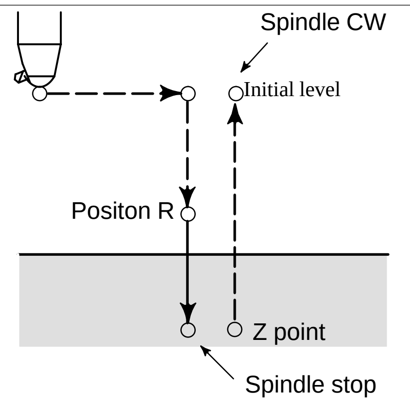
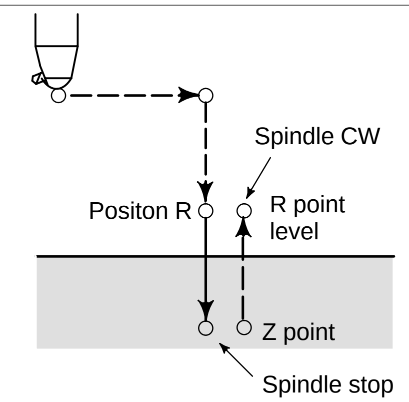
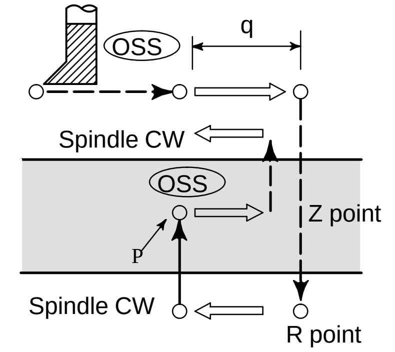
13 types of canned cycles (1/4)

G code	Operation		Function
	G98 mode	G99 mode	
G73			High–speed peck drilling cycle (Note 1)
G74			Counter tapping cycle
G76			FIne boring cycle

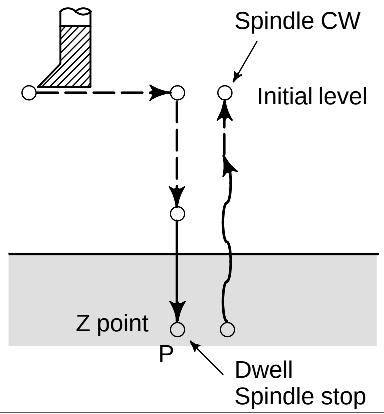
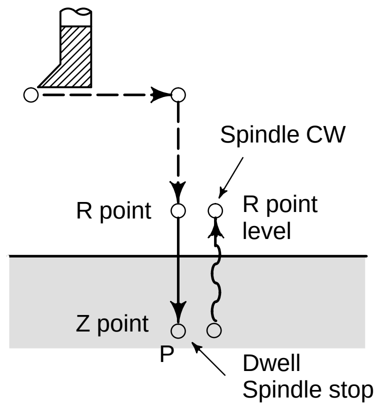
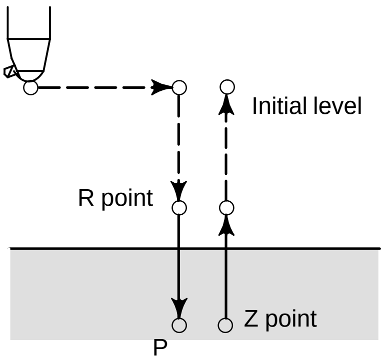
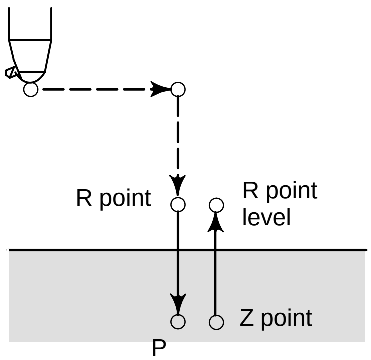
13 types of canned cycles (2/4)

G code	Operation		Function
	G98 mode	G99 mode	
G81			Drilling cycle (Spot drilling)
G82			Drilling cycle (Counter boring)
G83			Peck drilling cycle (Note 1)
G83			Small hole pock drilling cycle

13 types of canned cycles (3/4)

G code	Operation: G98 mode	Operation: G99 mode	Function
G84			Tapping cycle
G85			Boring cycle
G86			Boring cycle
G87		Not used	Back boring cycle

13 types of canned cycles (4/4)

G code	Operation		Function
	G98 mode	G99 mode	
G88			Boring cycle
G89			Boring cycle

⟶ Cutting feed

– – → Rapid traverse

～→ Manual feed

P Dwell

Z Z point (Hole bottom position)

OSS Oriented spindle stop (Spindle stops at constant rotation position)

⇨ Shift

Note 1 "d" of G73 and G83 is set by parameters.

When the drilling axis is Z axis, machining data in the canned cycle is commanded as follows:

Format

G○○ X_ Y_ Z_ R_ Q_ P_ K_ F_ ;

Drilling mode G○○ ; See previous table.
Drilling position dataX, Y ; Command position of the hole.

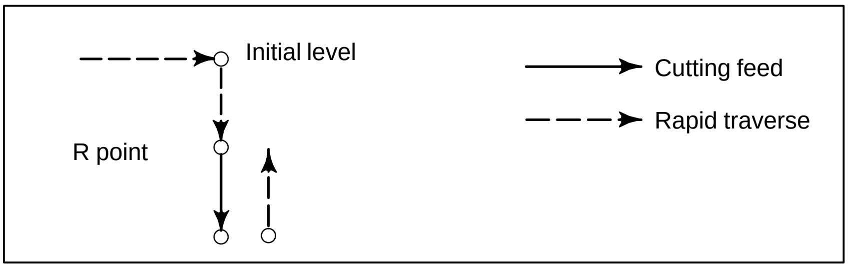
Z	:	Specify hole end position shown in the previous table.
R	:	Specify R point position shown in the previous table.
Q	:	Specify cutting quantity with G73, G83, and shift quantity with G76, G87,
P	:	Specify dwell time at the hole bottom.
K	:	Specify how may times to repeat. When specified K0, drilling data will be set, but no drilling will be done.
F	:	Specify feed rate for cutting.

Explanations

• R point level return (G99)

By specifying G99, return point in canned cycle is specified to R point. The drilling starts from the end point of the previous block. If the previous block has ended in the initial point, it begins from the initial point and returns to the R point.

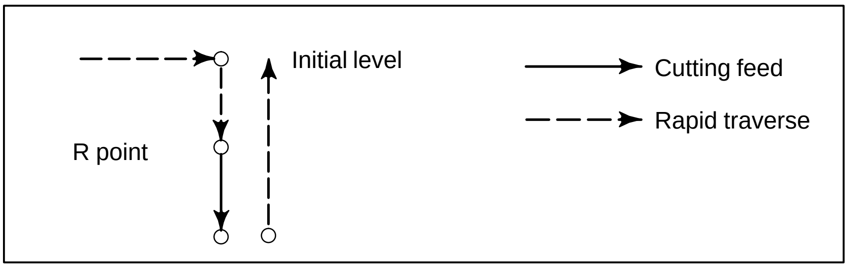
Example) When G81 was commanded under G99 mode



• Initial level return (G98)

By specifying G98, return point in canned cycle is specified to the initial level. The drilling starts from the end point of the previous block. If the previous block has ended in the R point, it begins from the R point and returns to the initial point.

Example) When G81 was commanded under G98 mode



13.2 RIGID TAPPING

13.2.1 Rigid Tapping

In tapping, the feed amount of drilling axis for one rotation of spindle should be equal to the pitch of screw of tapper. Namely, the following conditions must be satisfied in the best tapping:

P= F/S,

where P : Pitch of screw of tapper (mm)
F : Feed rate of drilling axis (mm/min)
S : Spindle speed (rpm)

The rotation of spindle and feed of Z axis are independently controlled in the tapping cycle G74/G84 (M series), G84/G88 (T series). Therefore, the above conditions may not always be satisfied. Especially at the hole bottom, both the rotation of spindle and feed of drilling axis reduce the speed and stop. After that, they move in the inverse direction while increasing the speed. However, the above conditions may not be satisfied in general since each acceleration/deceleration is performed independently. Therefore, in general, the feed is compensated by mounting a spring to the inside of holder of tapper to improve the accuracy of tap cutting.

The rotation of spindle and feed of drilling axis are controlled so that they are always synchronous each other in the rigid tapping cycle. Namely, in other than rigid tapping, control for speed only is performed. In the rigid tapping however, position control is also performed during the rotation of spindle, that is, the rotation of spindle and feed of drilling axis are controlled as linear interpolation of two axes.

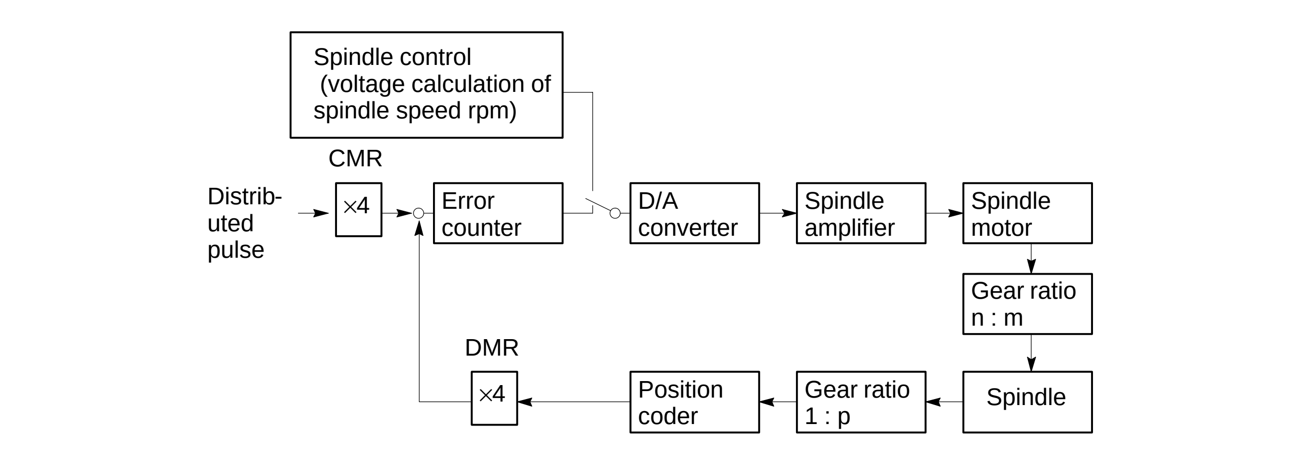
This allows the following condition to be satisfied also during acceleration/deceleration at the hole bottom and a tapping of improved accuracy to be made.

P = F/S

The pitch of screw tap can be directly specified.

Rigid tapping can be performed by executing any of the following commands:

- M29 S○○○○○ before tapping command G74/G84 (M series) or G84/G88 (T series)
- M29 S○○○○○ in the same block as tapping command G74/G84 (M series) or G84/G88 (T series)
- G74/G84 (M series) or G84/G88 (T series) as rigid tapping G code (Whether G74/G84 (G84/G88) is used as rigid tapping G code or ordinary tapping G code can be selected with a parameter.)



The Control System of Spindle during Rigid Tapping

Gear ratio of spindle to position coder (1 : p)	Least command increment (detection unit) deg
1 : 1	0.088 (1x360 / 4096)
1 : 2	0.176 (2x360 / 4096)
1 : 4	0.352 (4x360 / 4096)
1 : 8	0.703 (8x360 / 4096)

Even use of the spindle motor incorporating the position coder enables rigid tapping. In this case, the gear ratio of the spindle motor and the spindle is set by the parameter.
In addition, use of the spindle motor incorporating the position coder enables rigid tapping but disables threading and per revolution dwell.

Pull–out override

- **Parameter setting–based method**

Override with a previously set parameter value can be applied to the pull–out operation.
Theoretically, it is possible to apply a pull–out override of up to 2000% (20 times). (Also take machine–imposed restrictions into account.)
If an override result would exceed the maximum permissible spindle rotation speed (specified in a parameter) for rigid tapping, the actual spindle speed is clamped at the maximum permissible spindle rotation speed.

- **Program instruction–based method**

A program instruction can specify to apply override to a pull–out operation. (Using this method requires setting an additional parameter.)
To specify pull–out override with a program instruction, specify a pull–out spindle rotation speed as one of the hole making data items in a G84 block, using the J address.
This instruction applies override with a value obtained in the following expression to a pull–out operation.

$$\frac{\text{Spindle rotation speed (instruction with the J address) for a return}}{\text{Spindle rotation speed (instruction with the S address)}} \times 100 = \text{Pull–out override value}$$

If the result of conversion made with the above expression does not fall in a pull–out override range of from 100% to 2000%, the spindle rotation instruction is disabled for the pull–out operation; the actual spindle rotation becomes 100%.
If the J instruction value is greater than the maximum permissible spindle rotation speed (specified in a parameter) for rigid tapping, the actual spindle rotation speed is clamped at or near the maximum permissible spindle rotation speed.

Example)

To make a pull–out operation twice faster than a cut–in operation, use the following instructions:

```
.
M29 S1000;
G84 Z–1000. F1000 J2000 ;
.
```

Feedrate override

Appropriate parameter setting enables the conventional feedrate override select signal and override cancel signal even when rigid tapping is under way.
In this case, the override value selected by the override select signal can be used for rigid tapping.

- <G012#0–7> is used as the override select signal.
- <G006#4> is used as the override cancel signal.
- The override value can be changed even when rigid tapping is under way.
- Using an applicable option enables the second feedrate override function. This function is applied to the second feedrate that is determined according to the first feedrate override function.
- The spindle override value is fixed at 100% when rigid tapping is under way. Because the spindle operation is synchronized with the operation of the tapping axis, the spindle speed is affected indirectly by the feedrate override value.
- Enabling the feedrate override select signal disables pull–out override (regardless of whether it is specified by a program instruction or owing to a rigid tapping return), causing feedrate override to be applied to the pull–out operation.

- Even if the feedrate override select signal is enabled, setting the override cancel signal to 1 causes 100% override to be applied to a cut–in operation. If pull–out override is enabled, it is applied to the pull–out operation.

The following table lists rigid tapping versus override value correspondence.

Rigid tapping versus feedrate override value correspondence

<table>
<tr><th colspan="2"></th><th>Cut–in operation</th><th>Pull–out operation</th></tr>
<tr><td colspan="2">Feedrate override = Disabled
Pull–out override = Disabled</td><td>100%</td><td>100%</td></tr>
<tr><td colspan="2">Feedrate override = Disabled
Pull–out override = Enabled</td><td>100%</td><td>Pull–out override (*)</td></tr>
<tr><td rowspan="2">Feedrate override = Enabled
Pull–out override = Disabled</td><td>Override cancel signal <G006#4>=0</td><td>Feedrate override signal value</td><td>Feedrate override signal value</td></tr>
<tr><td>Override cancel signal <G006#4>=1</td><td>100%</td><td>100%</td></tr>
<tr><td rowspan="2">Feedrate override = Enabled
Pull–out override = Enabled</td><td>Override cancel signal <G006#4>=0</td><td>Feedrate override signal value</td><td>Feedrate override signal value</td></tr>
<tr><td>Override cancel signal <G006#4>=1</td><td>100%</td><td>Pull–out override (*)</td></tr>
</table>

* The pull–out override can be any of the following:
<1> Parameter–set pull–out override
<2> Program–specified pull–out override
<3> Rigid tapping return–based pull–out override

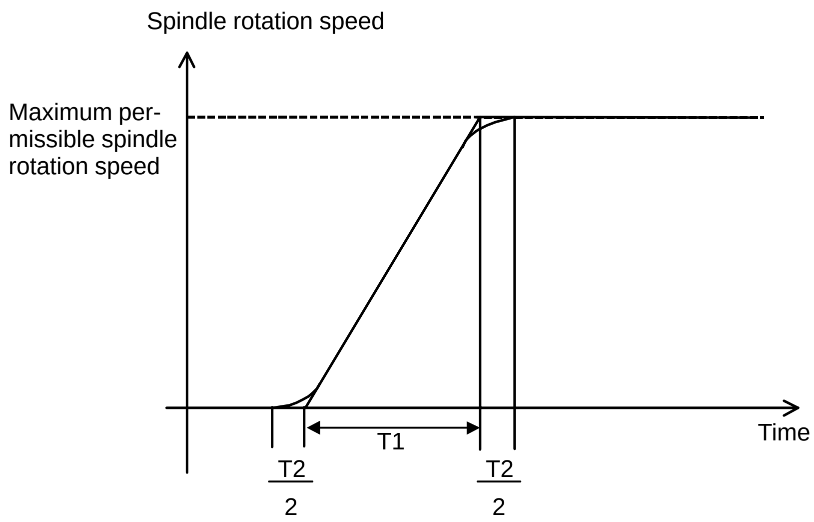
13.2.2 Rigid Tapping Bell–shaped Acceleration/ Deceleration

M series

Bell–shaped acceleration/deceleration can be used for rigid tapping. Generally, using bell–shaped acceleration/deceleration can reduce the required acceleration/deceleration time because the time constant of rigid tapping can be decreased.

For bell–shaped acceleration/deceleration for rigid tapping, the linear acceleration/deceleration constant and the time corresponding to the bell–shaped curve are specified using parameters.

The bell–shaped acceleration/deceleration time for rigid tapping is the sum of the time constant of linear acceleration/deceleration for the spindle and rigid tapping axis (conventional parameter setting T1) and the time corresponding to the curved portion (parameter setting T2).



T1 = Time constant of linear acceleration/deceleration for the spindle and rigid tapping axis

T2 = Time constant for the curved portion of bell–shaped acceleration/deceleration

T1 + T2 = Time required for acceleration/deceleration

The actual time constant of liner acceleration/deceleration for the spindle and tapping axis, T1, is determined according to a ratio of the maximum permissible spindle rotation speed to the actually specified S. However, the time constant for the curved portion of bell–shaped acceleration/ deceleration is not proportional to the actual S instruction, so a constant acceleration/deceleration (specified using a parameter) is always maintained.

13.3 EXTERNAL OPERATION FUNCTION (G81)

M series

With the above program, external operation signal is output after positioning. G80 command cancels the external operation function.

Format

G81 IP_ ;

IP : Optional combination of axis address X, Y, Z, U, V, W, A, B, C

13.4 CANNED CYCLES FOR TURNING

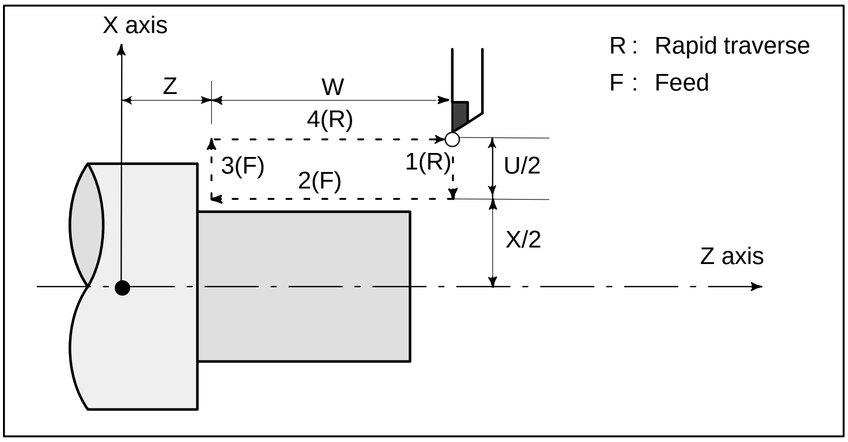
T series

The following three kinds of canned cycle are provided.

13.4.1 Cutting Cycle A (G77) (with G Code System A: G90)

- **Straight cutting cycle.**

The command below actuates a straight cutting cycle.

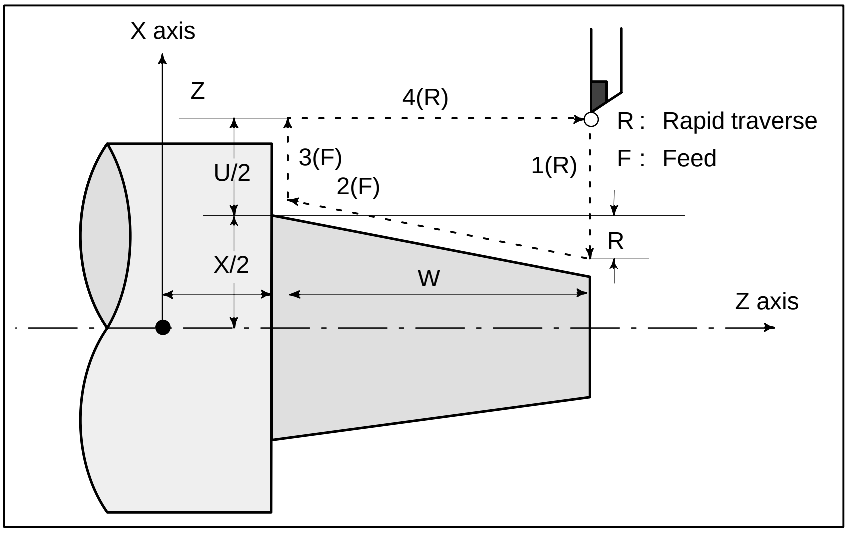


Format

G77 X_ Z_ F_ ;

- **Tapered cutting cycle**

The command below actuates a tapered cutting cycle.
In the figure below, when the direction of route 1 is –X, R is a negative value. Inverting the sign of R enables reverse taper cutting.



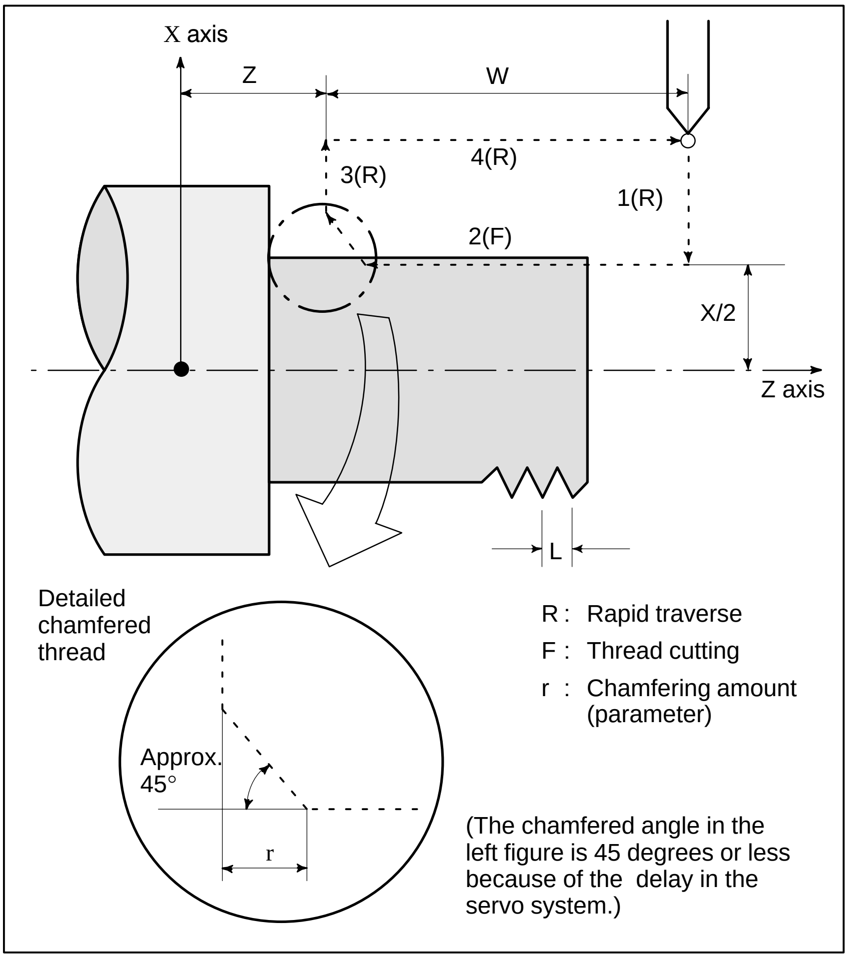
Format

G77 X_ Z_ R_ F_ ;

13.4.2 Thread Cutting Cycle (G78) (with G Code System A: G92)

- **Straight thread cutting cycle**

The command below actuates a straight thread cutting cycle.

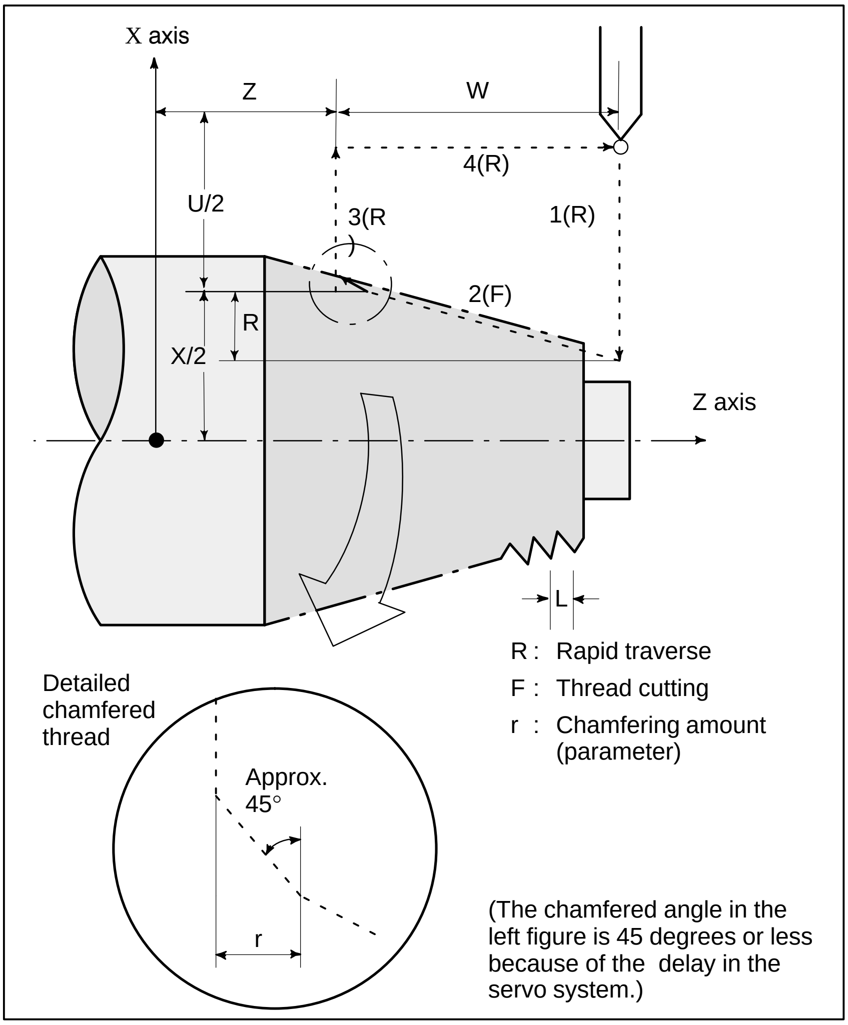


Format

G78 X_ Z_ F_ ;

- **Tapered thread cutting cycle**

The command below actuates a tapered thread cutting cycle.



Format

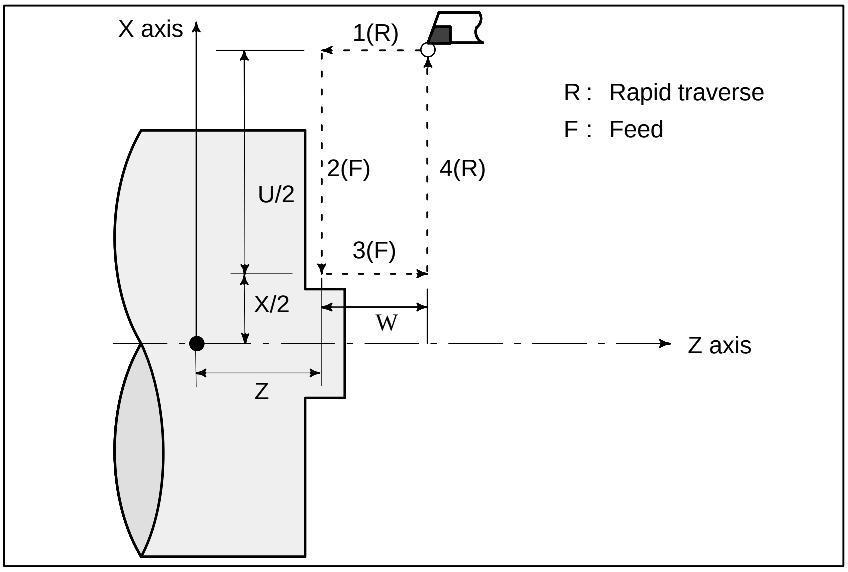
G78 X_ Z_ R_ F_ ;

NOTE
Screw chamfering can be inhibited by entering the chamfering signal.

13.4.3 Turning Cycle in Facing (G79) (with G Code System A: G94)

- **Face cutting cycle**

The command below actuates a face cutting cycle.

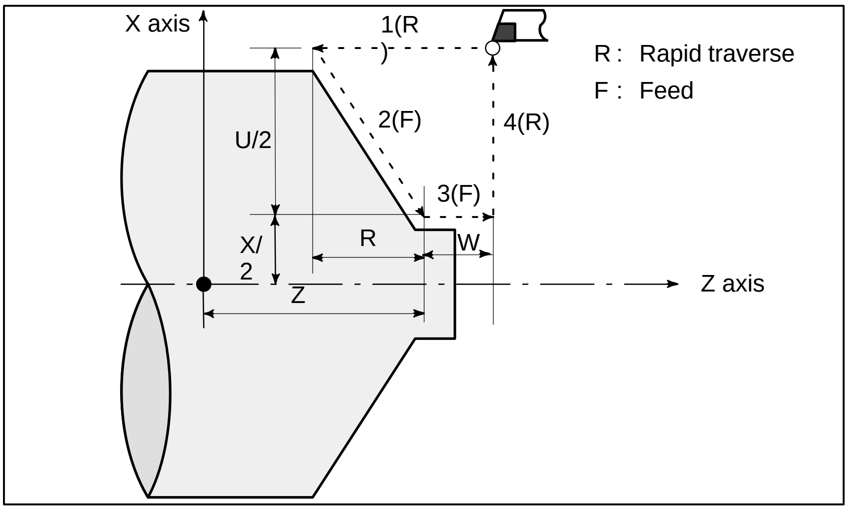


Format

G79 X_ Z_ F_ ;

- **Face tapered cutting cycle**

The command below actuates a face tapered cutting cycle.
In the following figure, if the direction of the path 1 is negative in Z axis, the sign of the number following address R is negative.



Format

G79 X_ Z_ R_ F_ ;

13.5 MULTIPLE REPETITIVE CYCLES FOR TURNING (G70 - G76)

T series

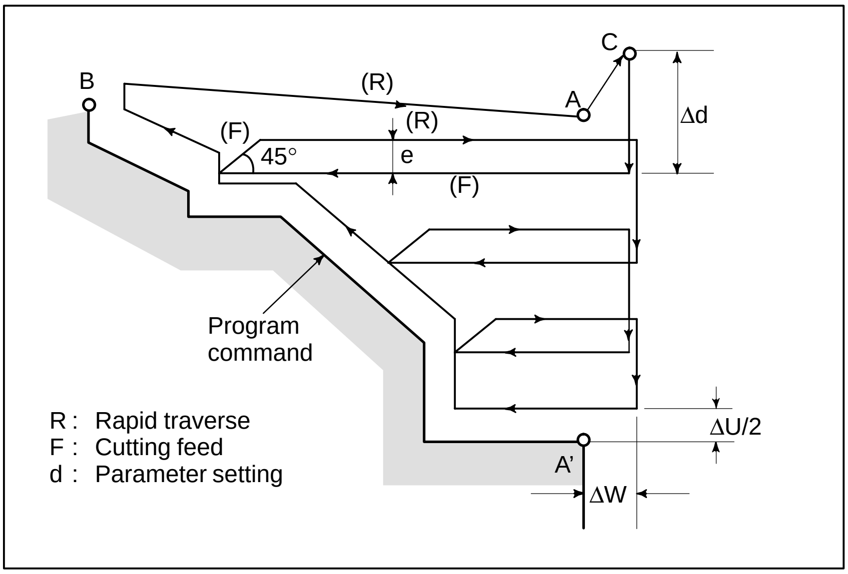
A multiple repetitive cycle is composed of several canned cycles. A tool path for rough machining, for example, is determined automatically by giving the data of the finishing work shape. A thread cutting cycle has also been prepared.

13.5.1 Stock Removal in Turning (G71)

There are two types of rough cutting cycles for external surfaces, type I and type II.

- **Type I**

If a finishing shape of A to A' to B is given in the figure below, machining is done with the cutting depth Δd and the finishing allowance ΔU/2 and ΔW.



Format

G71 U(Δd) R(e) ;

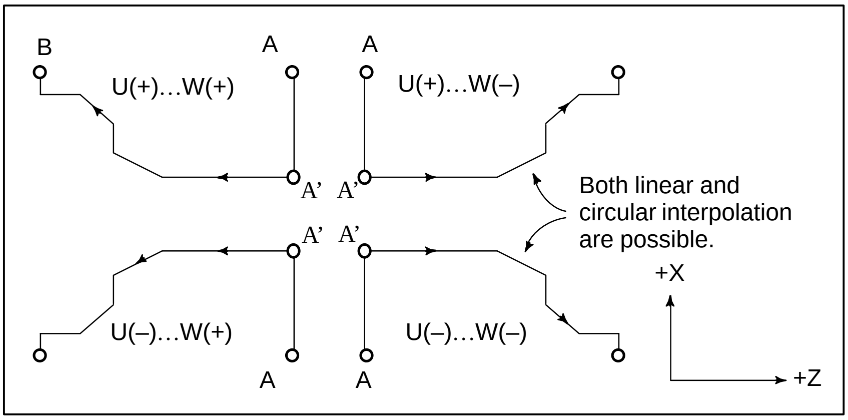
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ; (ns)

```
N(ns)  . . . . .
. . . . . . . . . . .
. . . . . . .  F_
. . . . . . .  S_
. . . . . . .  T_
N(nf) . . . . . . ;
```

A block between sequence numbers ns and nf specifies the target figure between A and B.

Δd	:	Depth of cut. It is specified without sign. The cutting direction is determined by the direction of AA’.
e	:	Clearance
ns	:	Sequence number of the first block of target figure blocks
nf	:	Sequence number of the end block of the target figure blocks
Δu	:	Distance and direction of finishing allowance along X axis
Δw	:	Distance and direction of finishing allowance along Z axis
f, s, t	:	The F, S, and T specified by a block between ns and nf are ignored during cycle operation. Those specified by the block of G71 or before are effective.

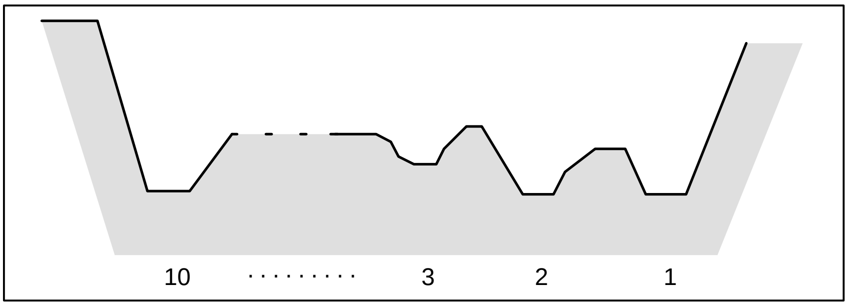
F, S, and T in the blocks of move commands from A to B are ignored and those specified in the same block as G71 or before are effective.
G96s (constant surface speed control on) and G97s (constant surface speed control off) in the blocks of move commands from A to B are ignored. A G96 or a G97 commanded in the same block as G71 or before is effective. The following four patterns are given depending on the sign of ΔU and ΔW as in the figure below. All of these cutting cycles are made parallel to Z axis.



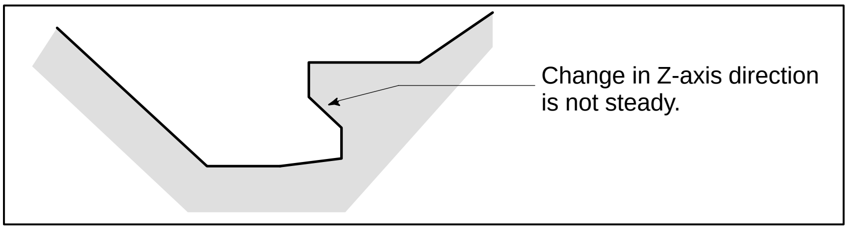
For the path from A to A’, the block of sequence number ns specifies a command including G00 or G01. For the path A’ to B, increase or decrease must be steady in both X-axis and Z-axis directions.
If the command for the path from A to A’ is G00, the cutting along the path is performed in the G00 mode. If the command for the path from A to A’ is G01, the cutting is performed in the G01 mode.

- **Type II**

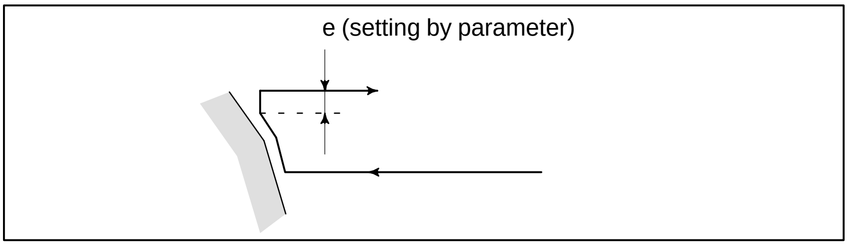
Type II differs from Type I in the following point.
Increase in X-axis direction does not need to be steady. Up to 10 pockets are allowed.



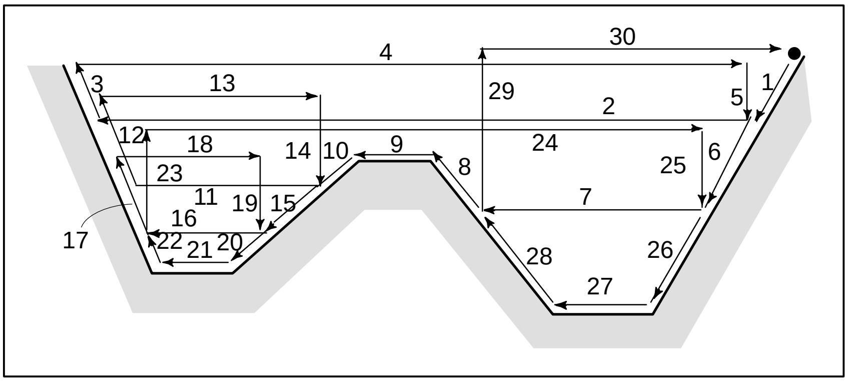
In Z-axis direction, however, increase or decrease must be steady. The following figure is not allowed for machining.



The first cutting does not need to be vertical. Any profile is allowed as far as the change in Z-axis direction is steady.
For clearance after turning, chamfering is performed along the workpiece profile.



The following figure shows an example of a cutting path when there are two pockets.



The offset of tool tip R is not added to the finishing allowance Δu and Δw. It is assumed to be zero for cutting. Generally $\Delta w=0$ is specified. Otherwise, the tool catches into a side wall. The two axes X(U) and Z(W) are specified in the first block of the repeat part. If there is no movement in Z-axis direction, W0 is specified.
This function is effective only in memory mode.

- **Use of Types I and II**

Type I:
Used when only one axis is specified in the first block (ns block) in the repeat part.

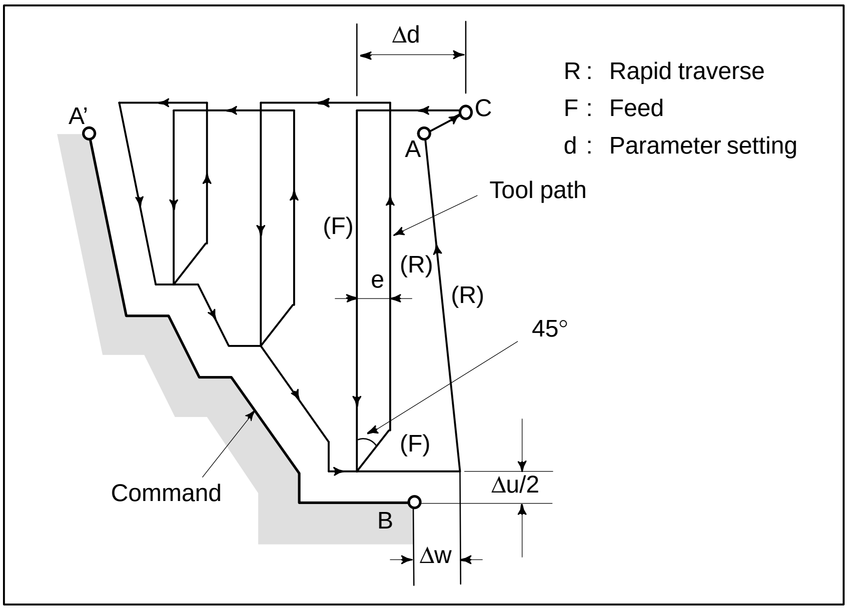
Type II
Used when two axes are specified in the first block in the repeat part.

Example)

Type I	Type II
G71 10.0 R5.0 ;	G71 10.0 R5.0 ;
G71 P100 Q200 … ;	G71 P100 Q200 … ;
N100 X(U)_ ;	N100 X(U)_ Z(W)_ ;
:	:
B200… ;	B200… ;

13.5.2 Stock Removal in Facing (G72)

As shown in the figure below, this cycle is the same as G71 except that cutting is made parallel to X-axis.



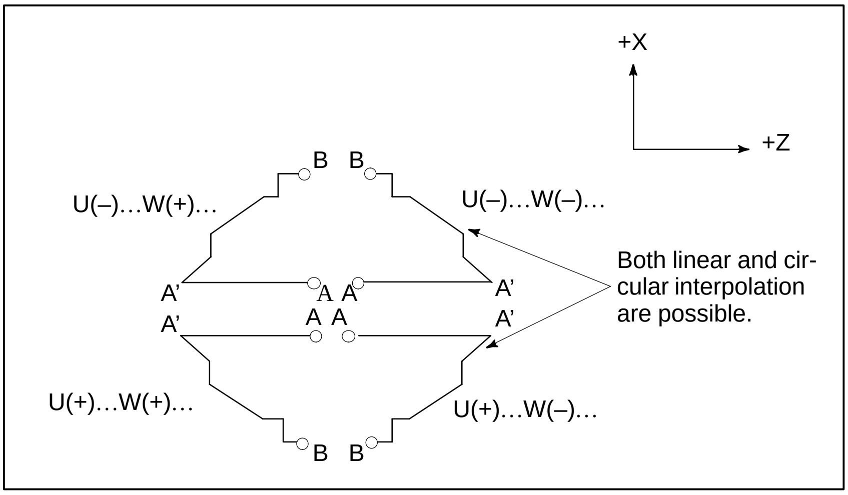
Format

G72 W(Δd) R(e) ;

G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

Δd, e, ns, nf, Δu, Δw, f, s, and t are the same as those in G71.

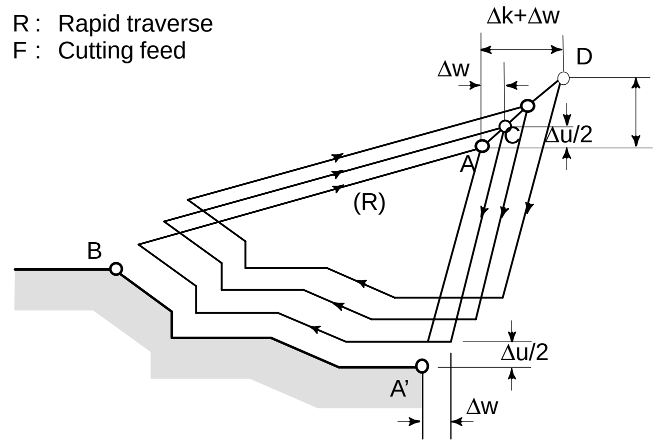
For the shape to be cut by G72, the following four patterns are considered. Any of them is cut by repetition of operation parallel to the X axis of the tool. The signs of ΔU and ΔW are as follows:



This function is effective only in memory mode.

13.5.3 Pattern Repeating (G73)

This function permits cutting a fixed cutting pattern repeatedly with the position being displaced bit by bit. By this cutting cycle, it is possible to efficiently cut the work whose rough shape has already been made by rough machining, forging, or casting, etc.



Pattern to be specified by the program
Point A→ Point A'→ Point B

Format

G73 U(Δi) W(Δk) R(d) ;

G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

```
N(ns) . . . . .
. . . . . . . . . . .
. . . . . . . F_
. . . . . . . S_
. . . . . . . T_
N(nf) . . . . . . ;
```

The move commands for the target figure from A to A' then to B are specified by blocks between sequence numbers ns and nf.

Δi	:	Distance and direction of the clearance along X axis (radius programming)
Δk	:	Distance and direction of the clearance along Z axis
d	:	Number of divisions which is equal to the number of times that rough cutting is performed
ns	:	Sequence number of the first block of target figure blocks
nf	:	Sequence number of the end block of target figure blocks
Δu	:	Distance and direction of the finishing allowance along X axis (diameter or radius programming)
Δw	:	Distance and direction of the finishing allowance along Z axis
f, s, t	:	F, S, and T codes

CAUTION

F, S, and T specified by any block between ns and nf are ignored. Those specified by the block of G73 or before are effective.

This function is available for only memory mode.

13.5.4 Finishing Cycle (G70)

After rough machining with G71, G72 or G73 the following command actuates finishing.

Format

G70 P(ns) Q(nf) ;

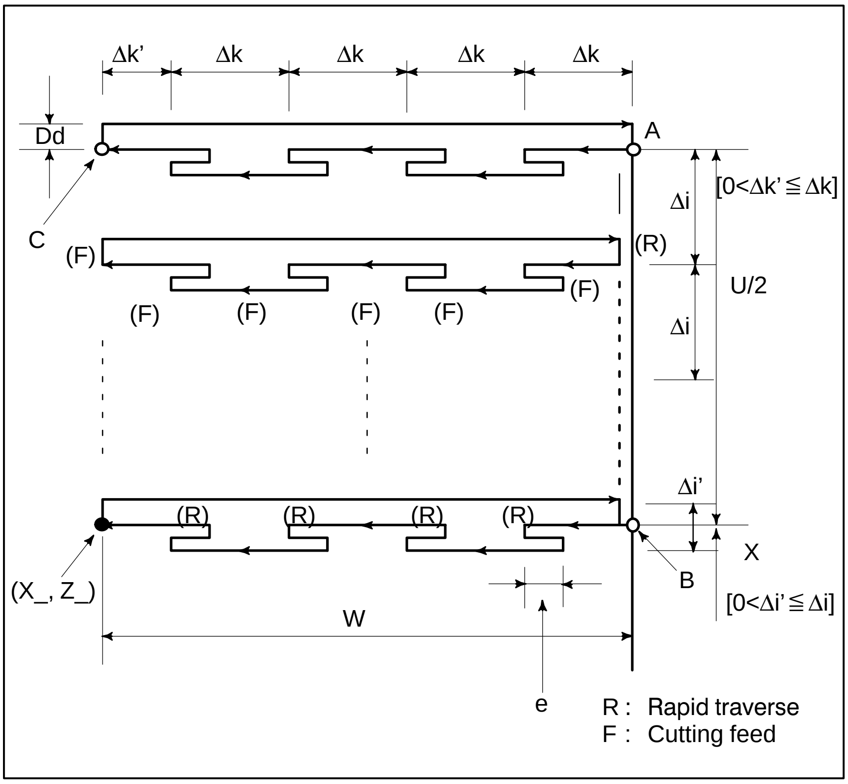
P : Sequence number of cycle start (ns)
Q : Sequence number of cycle end (nf)

NOTE
F, S, and T codes specified in the block of G71, G72 or G73 are ignored. But F, S, and T codes specified in the blocks from sequence numbers (ns) to (nf) become effective

The function is effective only in memory mode.

13.5.5 Peck Drilling in Z-axis (G74)

The following command permits operation as seen in the figure below. Chip breaking is possible in this cycle. Also if both X(U) and P are omitted, the machining is done only in the Z-axis resulting in peck drilling.



Format

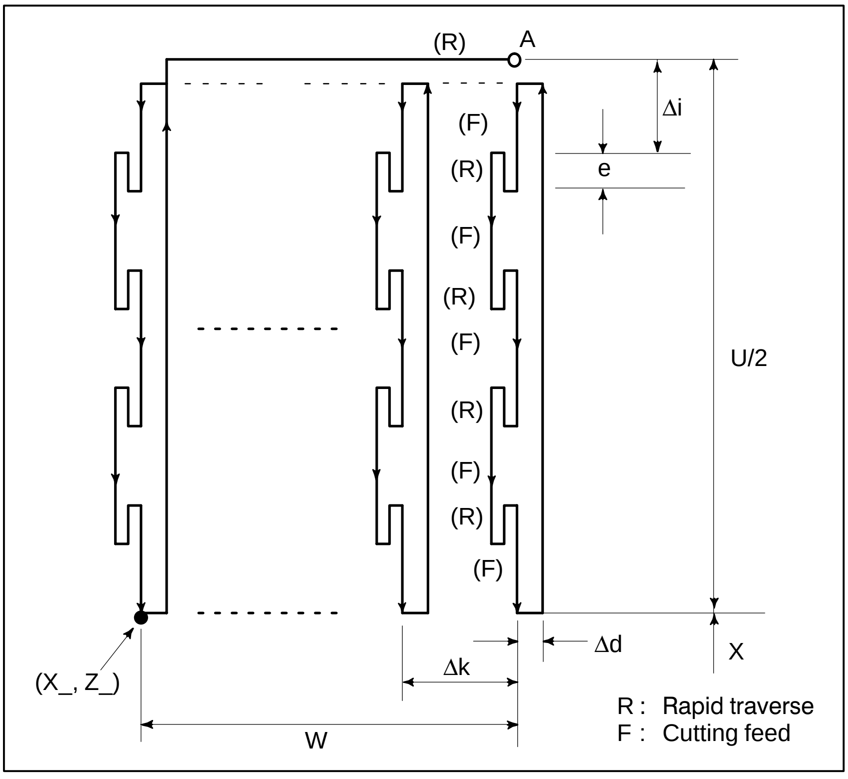
G74 R(e) ;

G74 $\left\{ \begin{array}{l} X\_\ Z\_ \\ U\_W\_ \end{array} \right\}$ P(Δi) Q(Δk) U(Δd) F(f) ;

e	:	Amount of return
X	:	X-axis coordinate of point B
U	:	Increment for A → B (for G code system A)
Z	:	Z-axis coordinate of point C
W	:	Increment for A → C (for G code system A)
Δi	:	Movement amount in X-axis direction (without sign)
Δk	:	Depth of cut in Z-axis direction (without sign)
Δd	:	Clearance amount at the cutting bottom Usually a positive integer is specified. If X(U) and Δi are omitted, however, the sign indicating the direction is added.
f	:	Feedrate

13.5.6 Grooving in X-axis (G75)

The following tape command permits operation as seen in the figure below. This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle. Grooving in the X-axis (in this case, Z, W and Q are omitted) is possible.



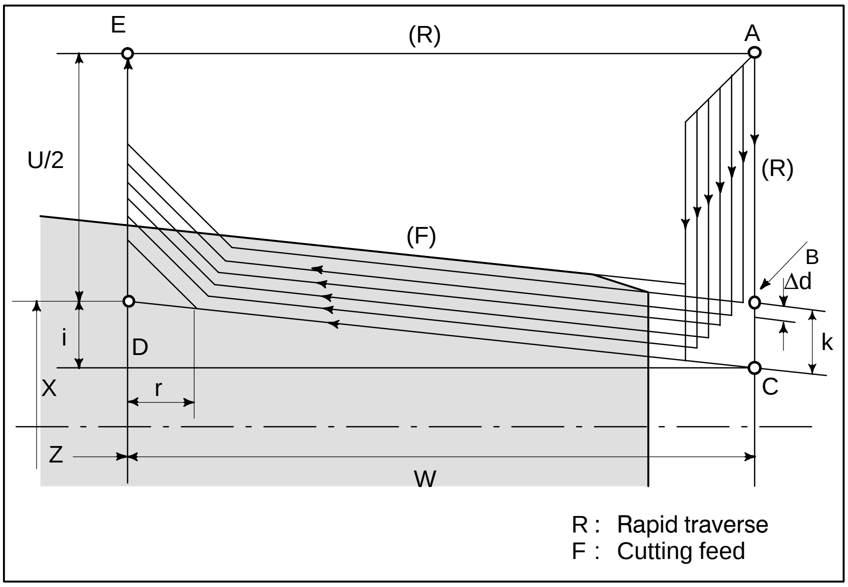
Format

G75 R(e) ;

G75 { X_ Z_ / U_W_ } P(Δi) Q(Δk) R(Δd) F(f) ;

13.5.7 Thread Cutting Cycle (G76)

A thread cutting cycle as shown below can be made.



Format

G76 P(m)(r)(a) Q(Δd min) R(d) ;

G76 {X_ Z_ / U_W_} R(i) P(k) Q(Δd) F(ℓ) ;

m : Number of final finishing repeats 1 to 99

r : Screw finishing (chamfering amount)

a : Tool tip angle (thread angle)
One of the six angles, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected as a 2-digit number. m, r, and a are specified in address P at the same time.

Example)
When m = 2, r = 1.2ℓ, and a = 60 , they are specified as follows:

$\underset{m}{\underline{P02}}\ \underset{r}{\underline{12}}\ \underset{a}{\underline{60}}$

Δbmin: Minimum depth of cut

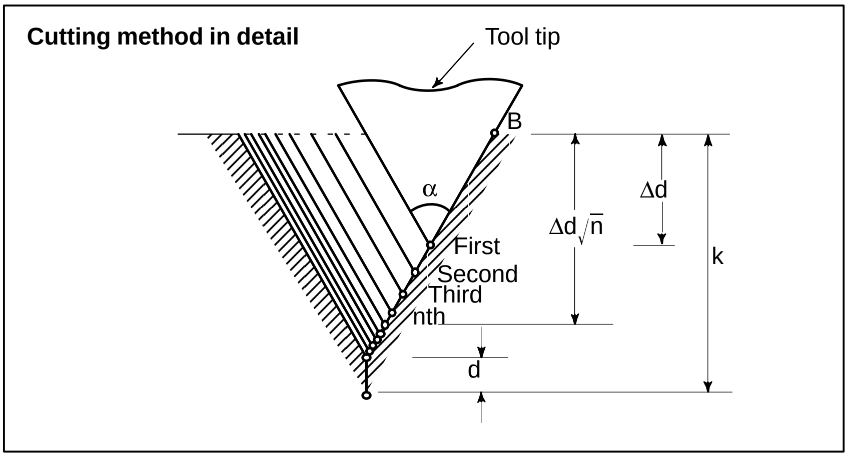
d : Finishing allowance

i : Difference in thread radius
Straight threading for i = 0

k : Height of the thread (The distance in X-axis direction is specified with a radius value.)

Δd : Depth of first cut (specified with a radius value)

ℓ : Screw lead (same as threading of G32)



NOTE
Thread chamfering can be inhibited by entering the chamfering signal.

13.6 CANNED CYCLES FOR DRILLING (G80 - G89)

T series

The canned cycles for drilling enable one block including the G function to specify the machining which is usually specified by several blocks. Programming is then simplified.
The canned cycles for drilling conform to JIS B 6314.

Canned cycles

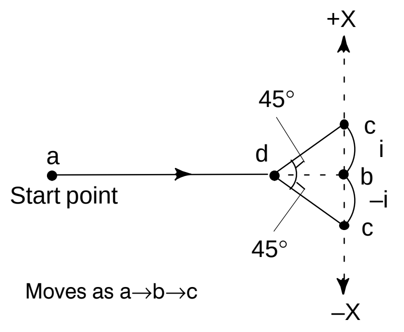
G code	Drilling axis	Drilling	Operation at hole bottom	Clearance	Use
G80					Cancel
G83	Z axis	Cutting feed Intermittent feed	Dwell	Rapid traverse	Deep drilling
G84	Z axis	Cutting feed	Spindle reverse	Cutting feed	Tapping
G85	Z axis	Cutting feed	Dwell	Cutting feed	Boring
G87	X axis	Cutting feed Intermittent feed	Dwell	Rapid traverse	Deep drilling
G88	X axis	Cutting feed	Spindle reverse	Cutting feed	Tapping
G89	X axis	Cutting feed	Dwell	Cutting feed	Boring

13.7 CHAMFERING AND CORNER R

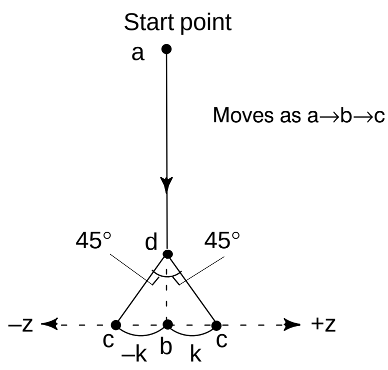
T series

A chamfer or corner are can be inserted between two blocks which intersect at a right angle as follows. An amount of chamfering or corner are specifies by address I, K, or R.

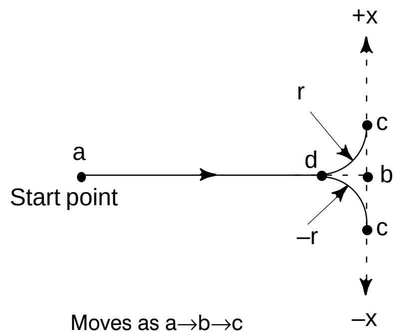
- **Chamfering**
  Z → X

Command	Tool movement
G01 Z(W) I(C) ±i ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

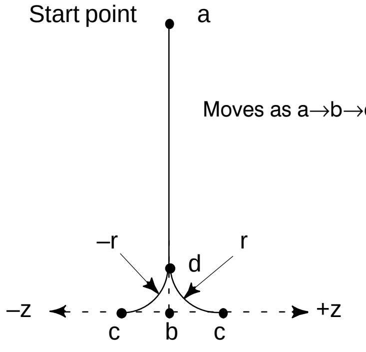
- **Chamfering**
  X → Z

Command	Tool movement
G01 X(U) K(C) ±k ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

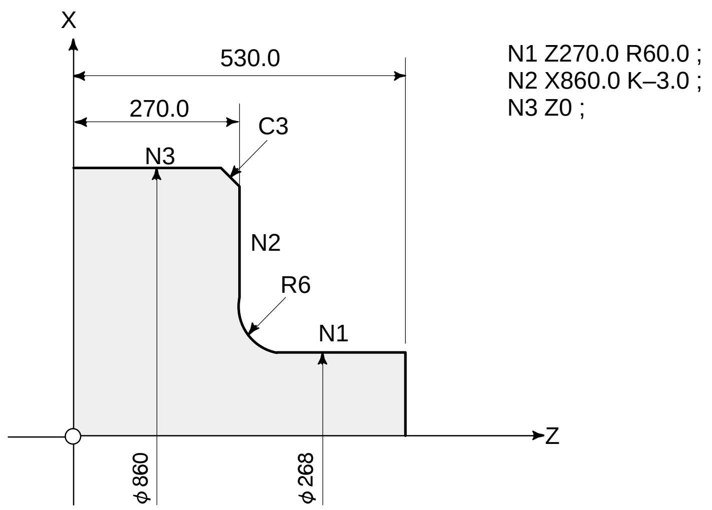
- **Corner R**
  Z → X

Command	Tool movement
G01 Z(W) R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

- **Corner R**
  X → Z

Command	Tool movement
G01 X(U) R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

CAUTION
If C is not used as an axis name, C can be used for a chamfer address instead of I or K.



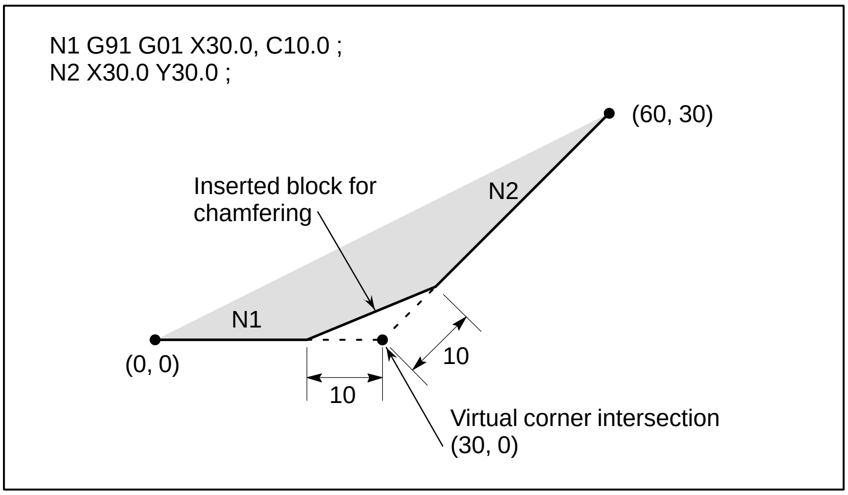
13.8 OPTIONAL ANGLE CHAMFERING/ CORNER ROUNDING

M series

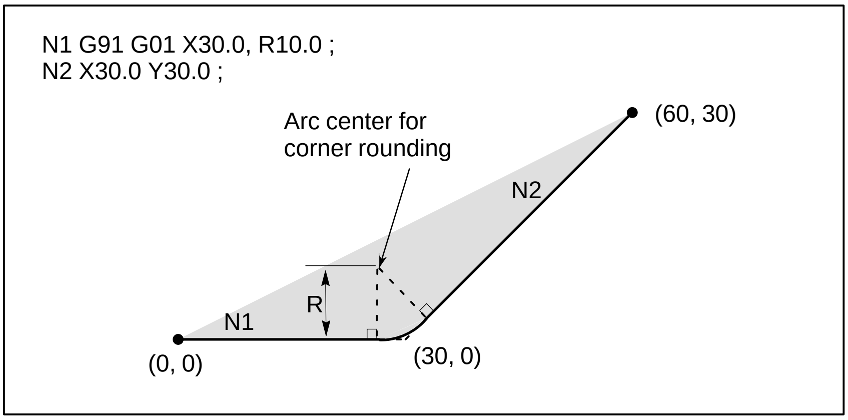
The block for chamfering or corner rounding can be inserted automatically between two optional linear interpolations, or between the linear interpolation and circular interpolation, or between two circular interpolations.

Specifying “**,C_**” inserts the block for chamfering, and specifying “**,R_**” inserts the block for corner rounding. They must be specified at the end of the block which specifies the linear interpolation (B01) or circular interpolation (G02 or G03).

The numeric following C specifies the distance between the virtual corner intersection and the chamfering start or end point. See the figure below.



The numeric following R specifies the radius value of corner rounding. See the figure below.



13.9 DIRECT DRAWING DIMENSIONS PROGRAMMING

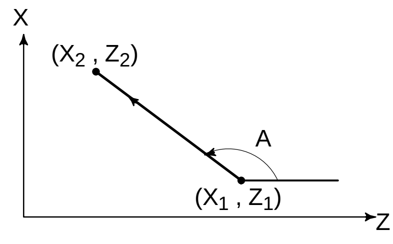
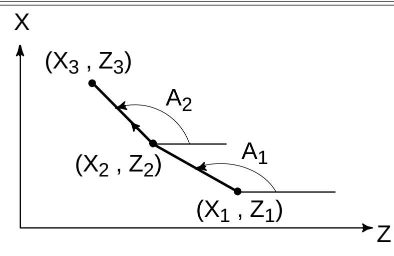
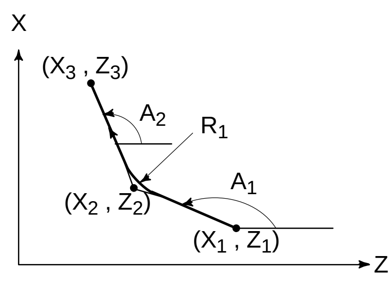
T series

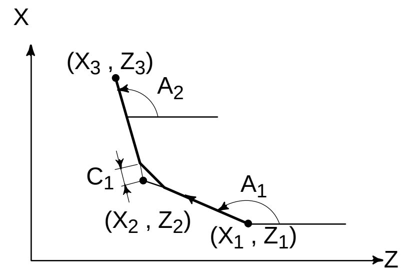
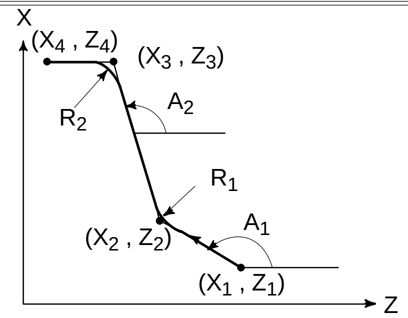
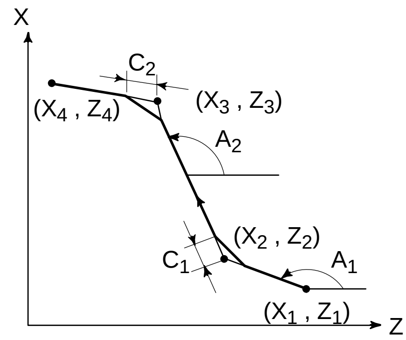
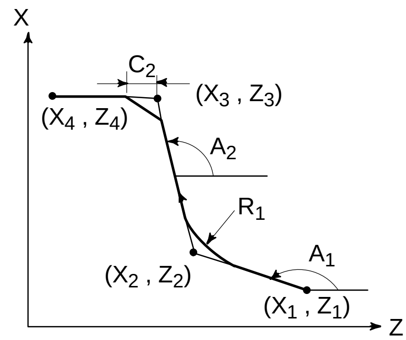
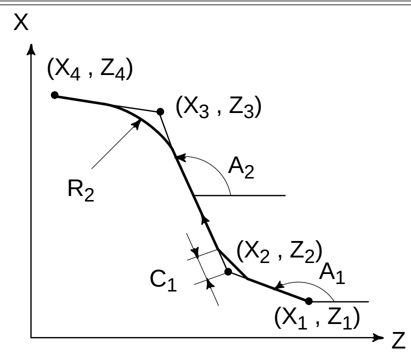
Angles of straight lines, chamfering values, corner rounding values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an arbitrary angle. The straight line angle, chamfering value, or corner rounding must be specified with a comma as follows:

,A_
,C_
,R_

> **NOTE**
> When A or C is not used as an axis name, the line angle, chamfering value, or corner rounding can be specified in the parameter without comma as follows:
> **A_**
> **C_**
> **R_**

Command list

	Command	Movement of tool
1	X_2_ (Z_2_), A_ ;	
2	, A_1_ ; X_3_ Z_3_, A_2_ ;	
3	X_2_ Z_2_, R_1_ ; X_3_ Z_3_ ; or , A_1_, R_1_ ; X_3_ Z_3_, A_2_ ;	

<table>
<tr><th></th><th>Command</th><th>Movement of tool</th></tr>
<tr><td>4</td><td>$X_2\_\ Z_2\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_\ ;$
or
$,\ A_1\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_,\ A_2\_\ ;$</td><td>
</td></tr>
<tr><td>5</td><td>$X_2\_\ Z_2\_,\ R_1\_\ ;$
$X_3\_\ Z_3\_,\ R_2\_\ ;$
$X_4\_\ Z_4\_\ ;$
or
$,\ A_1\_,\ R_1\_\ ;$
$X_3\_Z_3\_,\ A_2\_\ R_2\_\ ;$
$X_4\_\ Z_4\_\ ;$</td><td>
</td></tr>
<tr><td>6</td><td>$X_2\_\ Z_2\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_,\ C_2\_\ ;$
$X_4\_\ Z_4\_\ ;$
or
$,\ A_1\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_,\ A_2\_\ C_2\_\ ;$
$X_4\_\ Z_4\_\ ;$</td><td>
</td></tr>
<tr><td>7</td><td>$X_2\_\ Z_2\_,\ R_1\_\ ;$
$X_3\_\ Z_3\_,\ C_2\_\ ;$
$X_4\_\ Z_4\_\ ;$
or
$,\ A_1\_,\ R_1\_\ ;$
$X_3\_\ Z_3\_\ ,\ A_2\_\ C_2\_\ ;$
$X_4\_\ Z_4\_\ ;$</td><td>
</td></tr>
<tr><td>8</td><td>$X_2\_\ Z_2\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_,\ R_2\_\ ;$
$X_4\_\ Z_4\_\ ;$
or
$,\ A_1\_,\ C_1\_\ ;$
$X_3\_\ Z_3\_,\ A_2\_\ R_2\_\ ;$
$X_4\_\ Z_4\_\ ;$</td><td>
</td></tr>
</table>

13.10 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

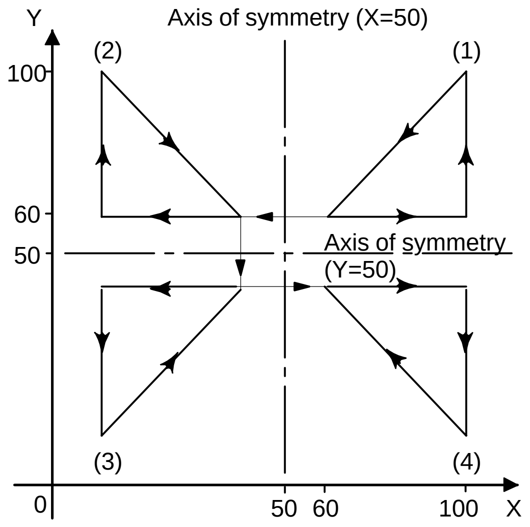
M series

Mirror image can be commanded on each axis by programming. Ordinary mirror image (commanded by remote switch or setting) comes after the programmable mirror image is applied.

- **Setting of programmable mirror image**
  G51.1 X_ Y_ Z_ ;
  is commanded and mirror image is commanded to each axis (as if mirror was set on the axis).
- **Programmable mirror image cancel**
  G50.1 X_ Y_ Z_ ;
  is commanded and the programmable mirror image is canceled.

> **CAUTION**
> If mirror image is specified only for one axis on the specified plane, the operation of the commands is as follows:
> - Arc command: The rotation direction is reversed.
> - Cutter compensation: The offset direction is reversed.
> - Coordinate rotation: The rotation angle is reversed.

When shape of the workpiece is symmetric to an axis, a program for machining the whole part can be prepared by programming a part of the workpiece using programmable mirror image and sub program.

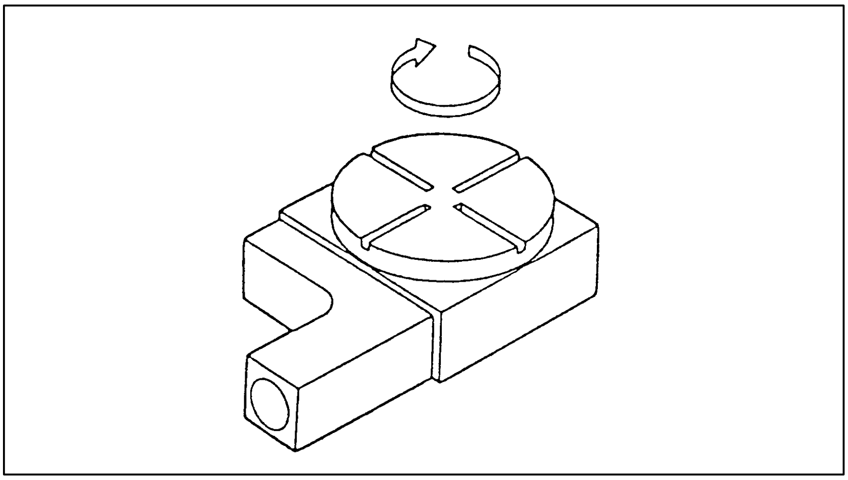



```
Main program
N10 G00 G90 ;
N20 M98 P9000 ;
N30 G51.1 X50.0 ;
N40 M98 P9000 ;
N50 G51.1 Y50.0 ;
N60 M98 P9000 ;
N70 G50.1 X0 ;
   (Cancel only X–axis)
N80 M98 P9000 ;
N90 G50.1 Y0 ;

Sub program
O9000 ;
G00 G90 X60.0 Y60.0 ;
G01 X100.0 F100 ;
G01 Y100.0 ;
G01 X60.0 Y60.0 ;
M99
```

13.11 INDEX TABLE INDEXING

M series

The index table on the machining center is indexed by using the fourth axis as an indexing axis.
To command for indexing, an indexing angle is only to be specified following a programmed axis (arbitrary 1 axis of A, B, C as the rotation axis) assigned for indexing. It is not necessary to command the exclusive M code in order to clamp or unclamp the table and therefore programming will become easy.

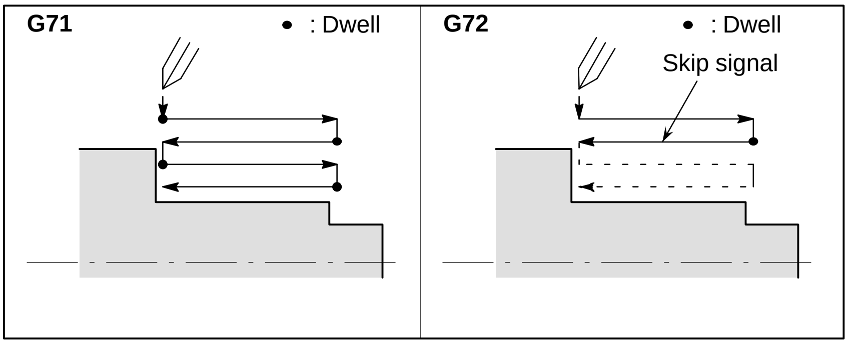


13.12 CANNED CYCLES FOR CYLINDRICAL GRINDING

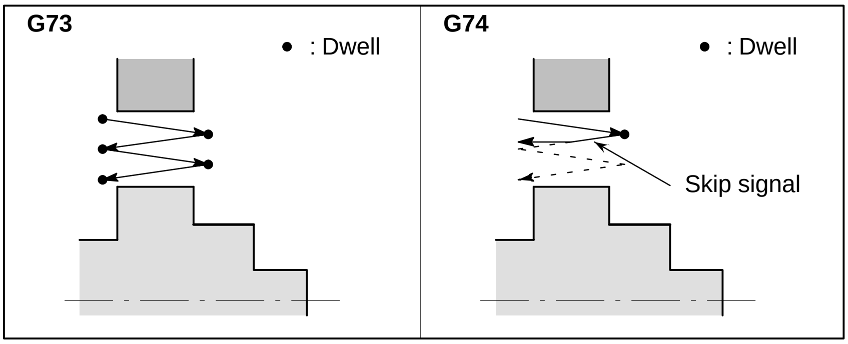
T series

The repetitive machining specific to grinding can be specified by one block. Since four types of canned cycles are provided for grinding, programming is simplified.

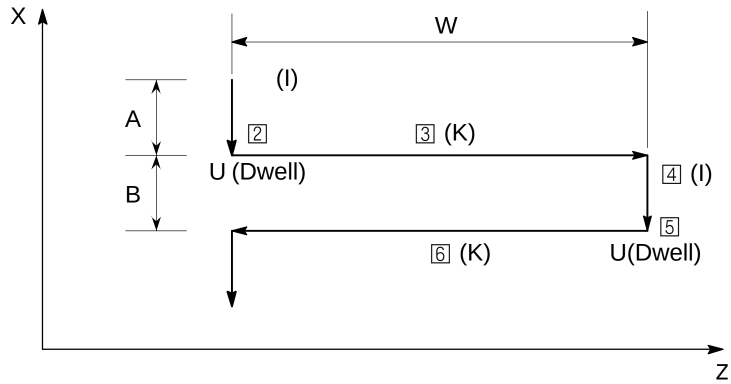
Traverse grinding cycle (G71, G72)



Oscillating grinding cycle (73, G74)



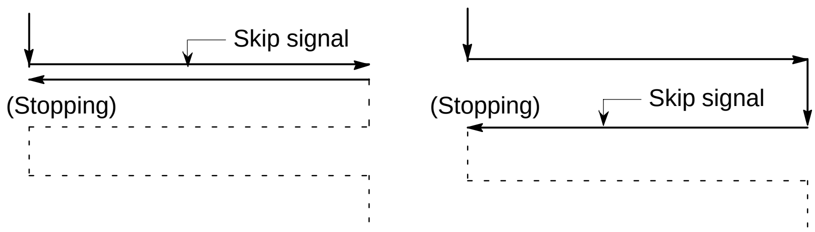
13.12.1 Traverse Grinding Cycle (G71)



G71 A_ B_ W_ U_ I_ K_ H_ ;

A	:	The first cutting depth
B	:	The second cutting depth
W	:	Grinding range
U	:	Dwell time Maximum command time 9999.999 sec.
I	:	Feed rate of A and B
K	:	Feed rate of W
H	:	Repetition frequency Setting value 1-9999

13.12.2 Traverse Direct Gauge Grinding Cycle (G72)

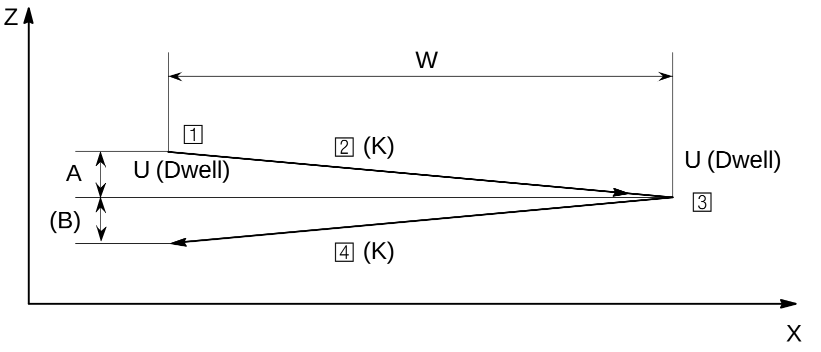


G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

P : Gauge number (1-4)

If the option of the multi-step skip is employed, gauge number can be specified. The specifying means of the gauge number is the same as the multi-step skip. If the option of the multi-step skip is not employed, a conventional skip signal becomes effective. Commands other than gauge number are similar to G71.

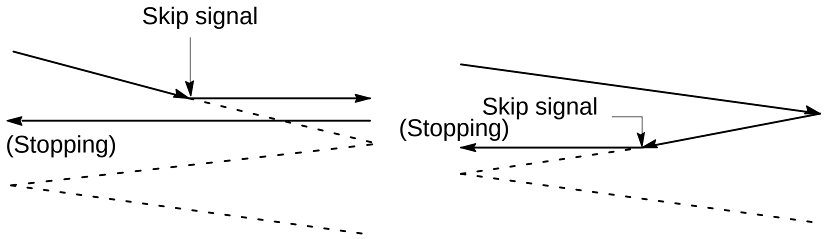
13.12.3 Oscillation Grinding Cycle (G73)



G73 A_ B_ W_ U_ K_ H_ ;

A : Cutting depth
B : Cutting depth
W: Grinding range
U : Dwell time
K : Feed rate
H : Repetition frequency Setting value 1-9999

13.12.4 Oscillation Direct Gauge Grinding Cycle (G74)



G74 P_ A_ (B_) W_ U_ K_ H_ ;

P : Gauge number (1-4)

If the option of the multi-step skip is employed, gauge number can be specified. The specifying means of the gauge number is the same as the multi-step skip. If the option of the multi-step skip is not employed, a convectional skip signal becomes effective. Commands other than gauge number are similar to G73.

13.13 SURFACE GRINDING CANNED CYCLE

M series

In the surface grinding canned cycle, repeated cutting peculiar to grinding machining normally commanded by a number of blocks, is simply programmed by commanding one block which includes the G function. There are the following 4 types of grinding canned cycle.

- **Plunge grinding cycle G75**
- **Plunge direct grinding cycle G77**
- **Continuous feed plane grinding cycle G78**
- **Intermittent feed plane grinding cycle G79**

13.13.1 Plunge Grinding Cycle (G75)

The plunge grinding cycle is possible by the following command.

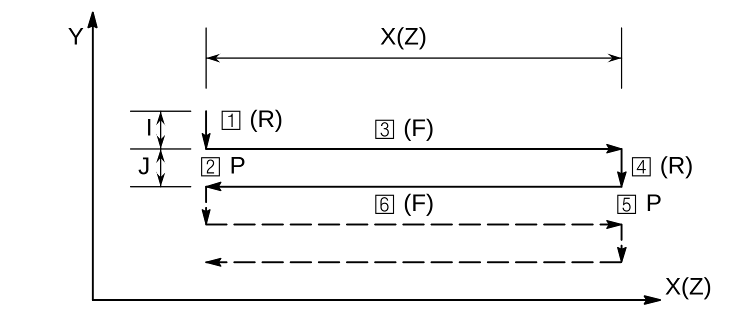
Format

G75 I_ J_ K_ X(Z)_ R_ F_ P_ L_ ;

I	:	The first cutting depth (Cutting direction is by command coding.)
J	:	The second cutting depth (Cutting direction is by command coding.)
K	:	Total cutting depth
X(Z)	:	Grinding range (Grinding direction is by command coding.)
R	:	Feed rate of I and J
F	:	Feed rate of X(Z)
P	:	Dwell time
L	:	Grindstone wear compensation number (Note 1)

Note 1) L is specified when performing continuous dressing.
Note 2) X(Z), I, J and K commands are all incremental commands.

Explanations



The plunge grinding cycle is made up from the following sequence of 6 operations.
The operations from 1 up to 6 are repeated until the grindstone cutting amount reaches the total cutting amount specified by address K.

1 **Grindstone cutting :**
Cuts in Y axis direction by cutting feed only the amount specified by the first cutting depth 1. The feed rate becomes the rate specified by R.

2 **Dwell :**
Performs dwell for only the time specified by P.

3 **Grinding :**
Shift by cutting feed only the amount specified by X (or Z) in the X axis direction (or Z axis direction). The feed rate becomes the rate specified by F.

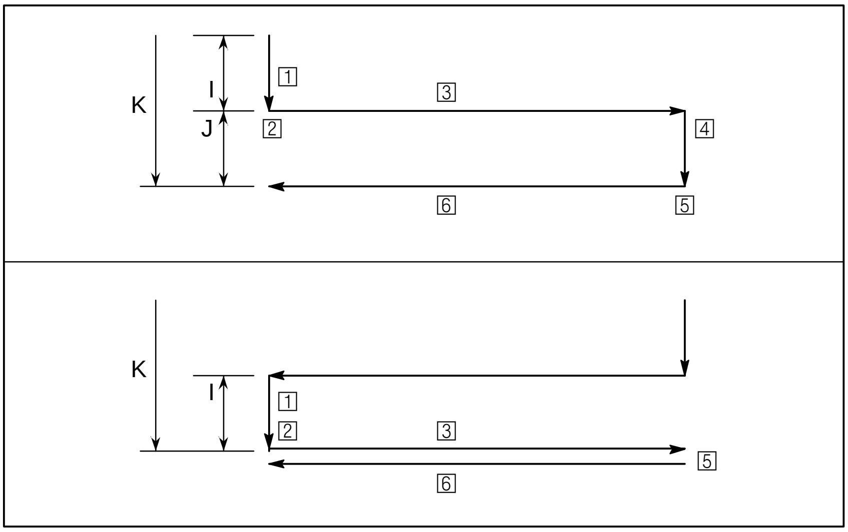
$\boxed{4}$ **Grindstone cutting :**
Cuts in Y axis direction by cutting feed only the amount specified by the second cutting depth J. The feed rate becomes the rate specified by R.

$\boxed{5}$ **Dwell :**
Performs dwell for only the time specified by P.

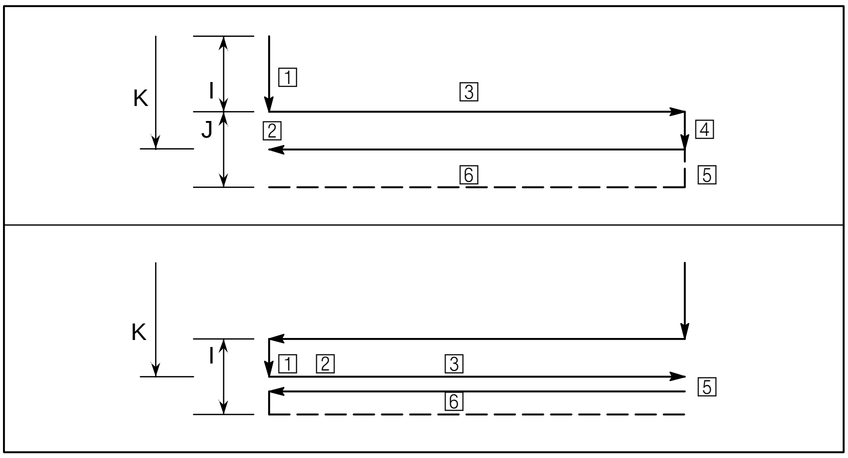
$\boxed{6}$ **Grinding (return direction) :**
Sent at rate specified by F in the reverse direction only the amount specified by X (or Z).

In case of single block, the operations from $\boxed{1}$ to $\boxed{6}$ are performed by one cycle start. When cutting by 1 or J, in the case where the total cutting depth is reached, the cycle finishes after the following sequence of operations (up to $\boxed{6}$) has been executed. The cutting depth in this case reaches the total cutting depth position.

- **When total cutting depth is reached by the cutting operation of I or J**



- **When total cutting depth is reached in the middle of cutting of I or J**



13.13.2 Plunge Direct Grinding Cycle (G77)

The plunge direct grinding cycle is possible by the following command.

Format

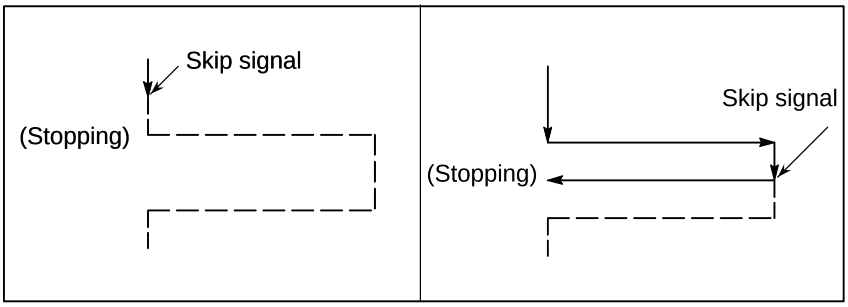
G77 I_ J_ K_ X(Z)_ R_ F_ P_ L_ ;

The command method is the same as the G75 case except for the G code. Further, even for the operation, the same sequence of 6 operations as the G75 case is repeated.
G77 differs from G75 as follows: Inputting a skip signal during a cycle can terminate the cycle after stopping (or terminating) the current operation sequence.
The following shows the operation at skip signal input for each operation sequence.

Explanations

- **Case of during operation sequence 1 and 4 (when I and J shift)**

Cutting immediately stops and returns to X(Z) coordinate at cycle start.

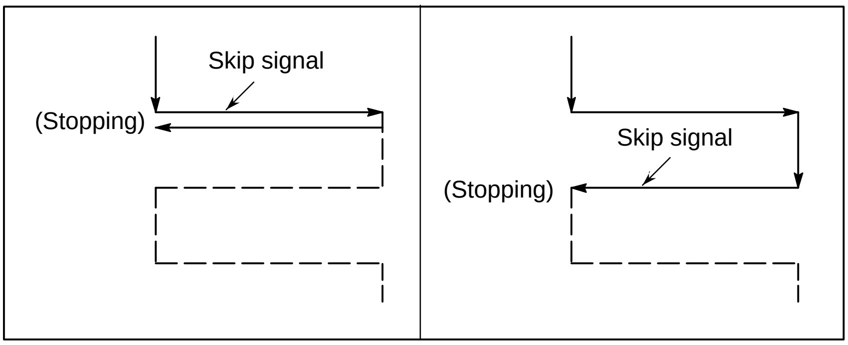


- **Case of during operation sequence 2 and 5 (during dwell)**

Dwell immediately stops and returns to X(Z) coordinate at cycle start.

- **Case of during operation sequence 3 and 6 (when X(Z) shifts)**

After shift of X(Z) has finished, returns to X(Z) coordinate at cycle start.



13.13.3 Continuous Feed Plane Grinding Cycle (G78)

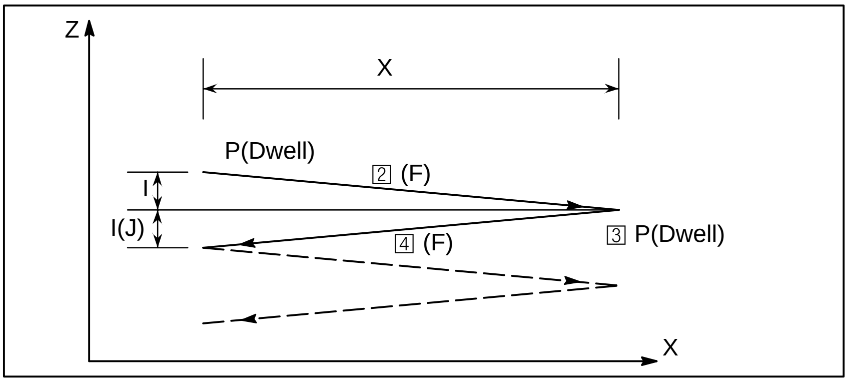
The continuous feed plane grinding cycle is possible by the following command.

Format

G78 I_ (J)_ K_ X_ R_ F_ P_ L_ ;

I : Cutting depth (Cutting direction is by command coding.)
J : Cutting depth (Cutting direction is by command coding.)
K : Total cutting depth
X : Grinding range (Grinding direction is by command coding.)
F : Feed rate
P : Dwell time
L : Grindstone wear compensation number (Note 1)

Note 1) L is specified when performing continuous dressing.
Note 2) X, I, J and K commands are all incremental commands.



The continuous feed plane grinding cycle is made up from the following sequence of 4 operations.
The operations from 1 up to 4 are repeated until the grindstone cutting depth reaches the total cutting depth specified by address K.

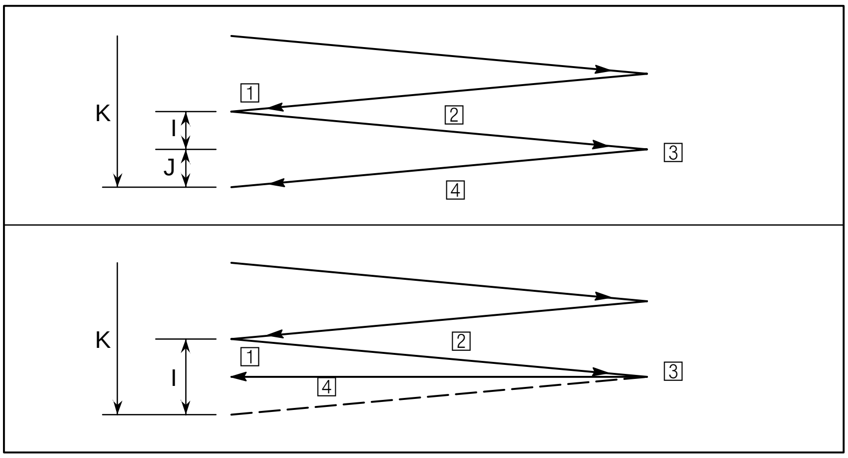
1 Dwell
2 Grinding
3 Dwell
4 Grinding (return direction)

In case of single block, the operation from 1 to 4 are performed by one cycle start.

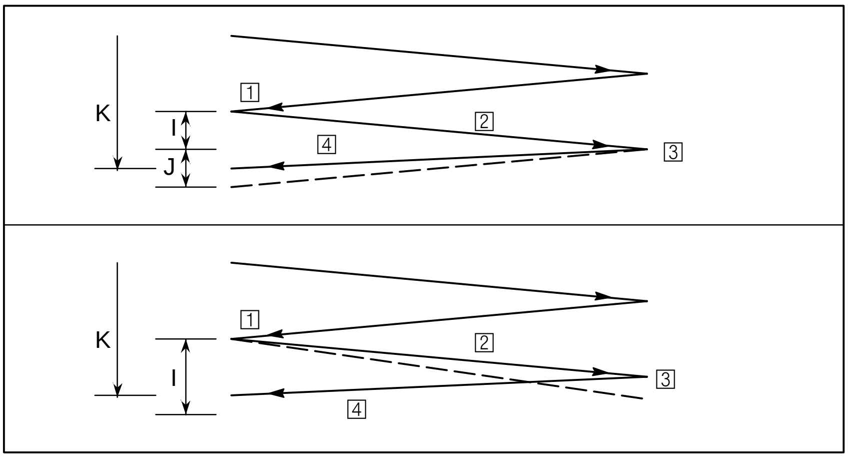
NOTE
When J is not commanded, it is regarded as J=1.
Further, the J command effective only at the specified block.
It does not remain as modal information. (Irrespective of “J” of G75, G77, and G79)

When cutting by I or J, in the case the total cutting depth is reached, the cycle finishes after the following sequence of operations (up to 4) has been executed. The cutting depth in this case reaches the total cutting depth position.

- **When total cutting depth is reached by cutting operation of I or J.**



- **When the total cutting depth is reached in the middle of cutting of I or J.**



13.13.4 Intermittent Feed Plane Grinding Cycle (G79)

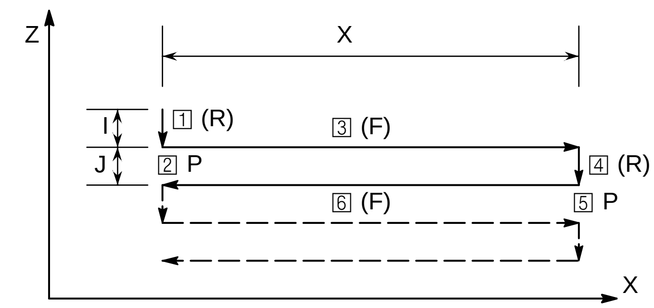
The intermittent feed plane grinding cycle is possible by the following command.

Format

G79 I_ J_ K_ X_ R_ F_ P_ L_ ;

I : The first cutting depth (Cutting direction is by command coding.)
J : The second cutting depth (Cutting direction is by command coding.)
K : Total cutting depth
X : Grinding range (Grinding direction is by command coding.)
R : Feed rate of I and J
F : Feed rate of X
P : Dwell time
L : Grindstone wear compensation number (Note 1)

Note 1) L is specified when performing continuous dressing.
Note 2) I, J, K and X commands are all incremental commands.



The intermittent feed plane grinding cycle is made up from the following sequence of 6 operations.
The operations from 1 up to 6 are repeated until the grindstone cutting depth reaches the total cutting depth specified by address K.

1 **Grindstone cutting :**
Cuts in Z axis direction by cutting feed only the amount specified by the first cutting depth I. The feed rate becomes the rate specified by R.

2 **Dwell :**
Performs dwell for only the time specified by P.

3 **Grinding :**
Shifts by cutting feed only the amount specified by X in the X axis direction. The feed rate becomes the rate specified by F.

4 **Grindstone cutting :**
Cuts in Z axis direction by cutting feed only the amount specified by the second cutting depth J. The feed rate becomes the rate specified by R.

5 **Dwell :**
Performs dwell for only the time specified by P.

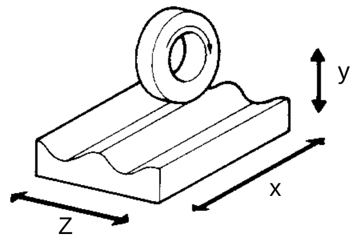
6 **Grinding (return direction) :**
Sent at rate specified by F in the reverse direction only the amount specified by X.

In the case of a single block, the operations from 1 to 6 are performed by one cycle start.

13.14 INFEED CONTROL

M series

Controls cutting a certain fixed amount along the programmed figure for input of external signals at the swing end point.



Format

G161 R_ ;

Figure program

G160 ;

G161R_	:	Commands the operation mode and start of start of figure program. Further, specifies the cutting depth by address R.
Figure program	:	Programs the workpiece figure in the Y-Z plane by either linear interpolation (G01) or by circular interpolation (G02, G03). Multiple blocks can be commanded.
G160	:	Commands cancelling of operation mode (ending of figure program).

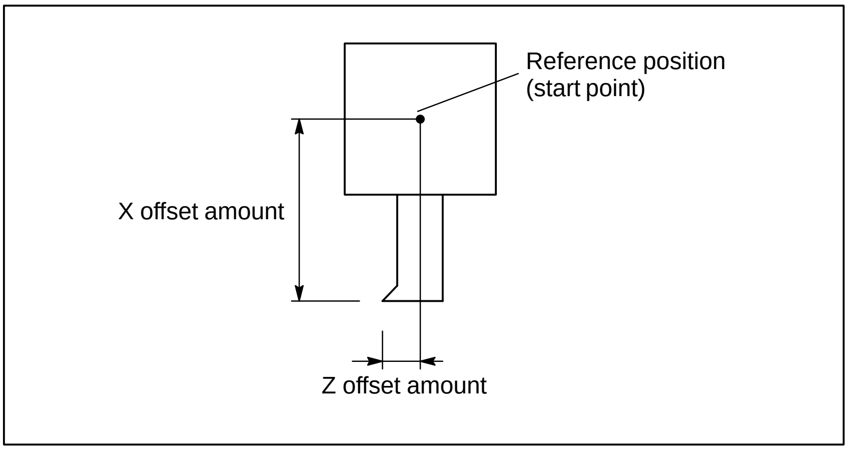
14 TOOL COMPENSATION FUNCTION

14.1 TOOL OFFSET

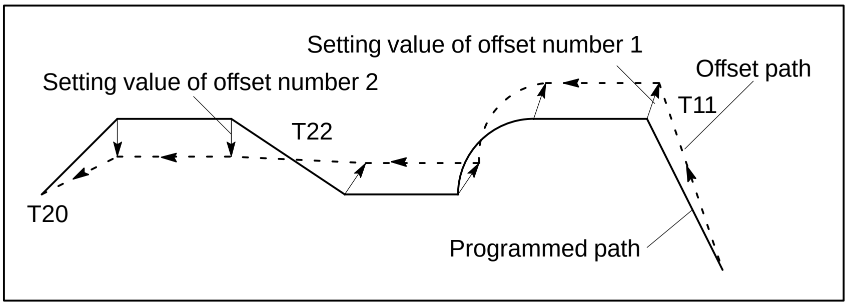
T series

14.1.1 Tool Offset (T Code)

By using this function, shift amount between the reference position assumed when programming and the actual tool position when machining, can be set as tool offset amount, thus allowing workpiece machining according to the programmed size without changing the program.



The tool offset can be commanded to X, Y, and Z axes.



Explanations

- **Offset number**

The offset number is specified in the last one or two digit of the T code. Use parameters to select offset number digits (one or two).

❏ When offset number is specified with one digit

T ○○○○○○○☆

Offset number

Tool number

❏ When offset number is specified with two digits

T ○○○○○○☆☆

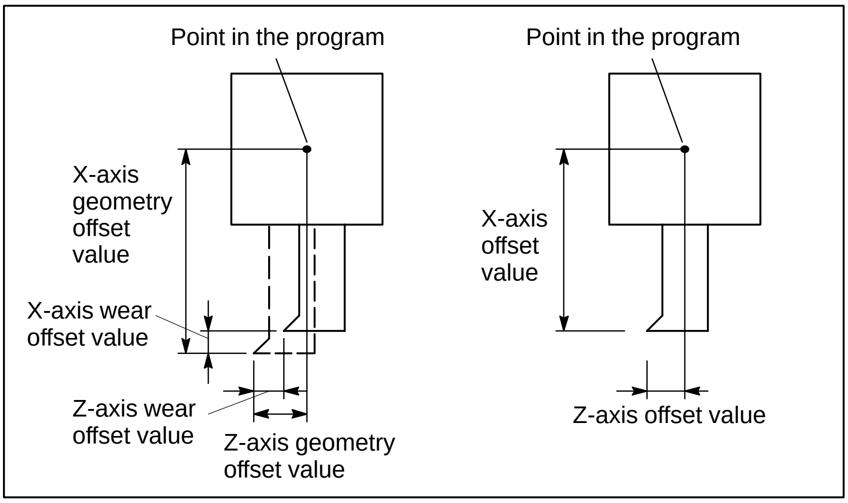
Offset number

Tool number

When the offset number is specified, the corresponding offset amount is selected, and tool offset starts.
When 0 is selected as offset number, the tool offset is canceled.

14.1.2 Tool Geometry Compensation and Tool Wear Compensation

The tool geometry compensation function compensates the tool figure or tool mounting position. The tool wear compensation function compensates the wear of a tool tip. These compensation amounts (offset values) can be set separately. There are two types of geometry compensation: So–called geometry compensation and the second geometry compensation that allows the user to specify whether to use the direction directed by the PMC. The second geometry compensation is used to compensate for the difference in tool mounting position or selected position. If distinction between them is not necessary, the total value of them is set as a tool position offset value.



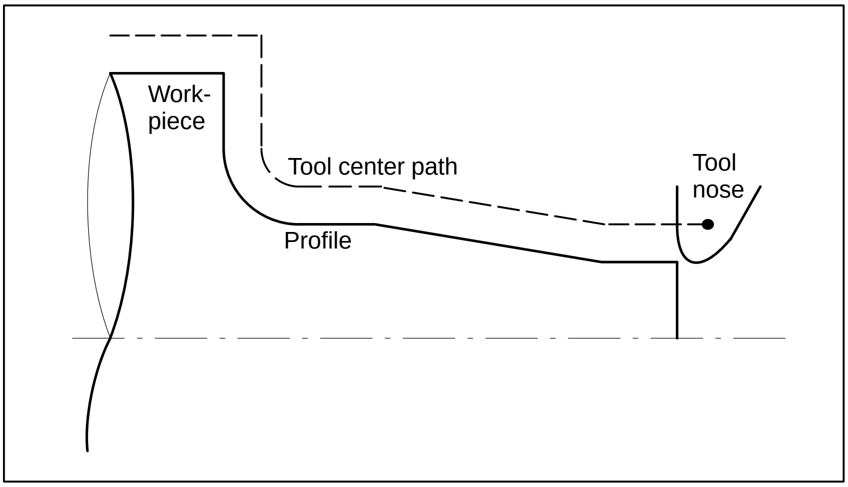
Tool geometry compensation and tool wear compensation not distinguished

Tool geometry compensation distinguished from tool wear compensation

14.2 TOOL NOSE RADIUS COMPENSATION (G40, G41, G42)

T series

With this function, the programmed tool path can be offset when actually machining, for value of the tool radius set in the CNC.
By programming machining pattern using this function (measuring cutter radius for actual cutting, and setting the value in the CNC as offset value), the tool can machine the programmed pattern, via the offset path. There is not need to change the program even when tool radius changes; just change the offset value.

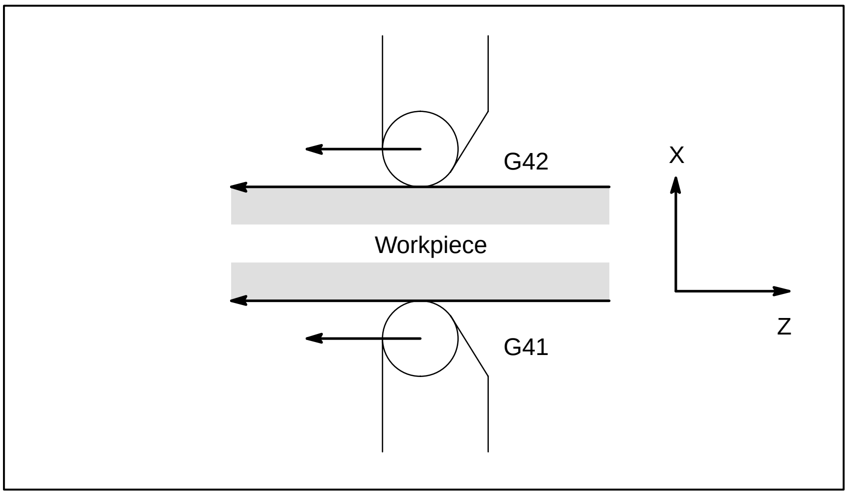


Cross points of line and line, arc and arc, line and arc is automatically calculated in the CNC to obtain offset actual tool path. So, Programming becomes simple, because it is only necessary to program the machining pattern.

Explanations

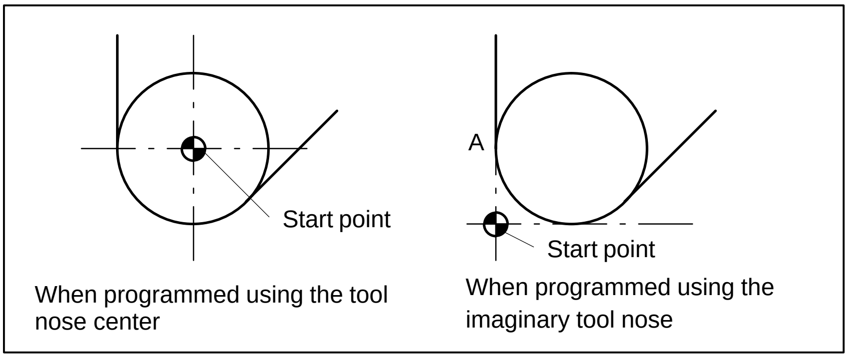
- **Tool nose radius compensation and its cancellation (G40, G41, G42)**

G40 : Tool nose radius compensation cancel
G41 : Tool nose radius compensation left
G42 : Tool nose radius compensation right
G41 and G42 are commands for tool nose radius compensation mode. The tool is offset to the left forward in the tool movement in G42 and right forward in G42. Tool nose radius compensation is cancelled with G40.

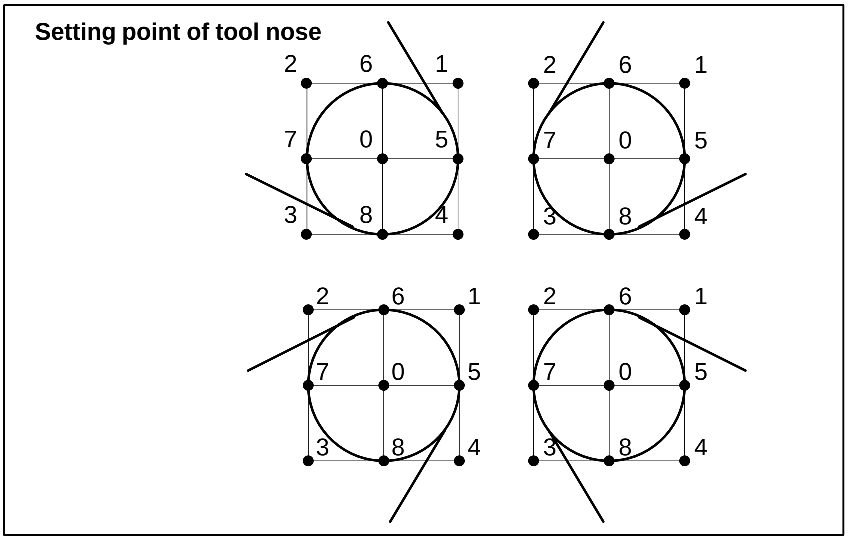


- **Imaginary tool nose**

The tool nose at position A in the following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose center to the start point than the imaginary tool nose. (Note) Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.



The position relationship when the tool is set to the start point is shown in the following figure. The point of tool nose for start point or reference point i set in offset memory same as tool nose radius compensation amount.



- **Tool nose radius compensation amount and assignment of imaginary tool nose point (T code)**

Tool nose radius compensation amount and imaginary tool nose point can be set in the tool nose radius compensation memory.
When the last one or two digits of T code is commanded as offset number, corresponding tool nose radius compensation amount and imaginary tool nose point in the tool compensation memory is applied as the tool nose radius compensation amount and imaginary tool nose point for cutter radius compensation.

• Plane selection (G17, G18, G19)

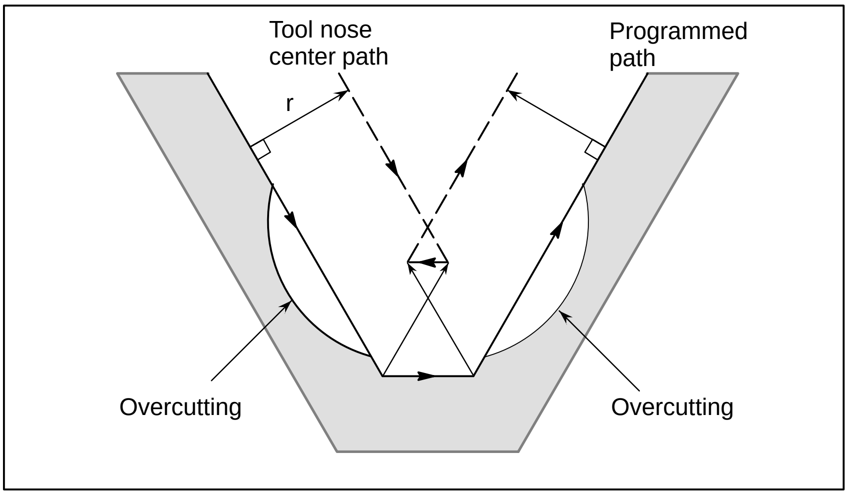
Cutter radius compensation is done on XY, ZX, YZ planes and on parallel axes of X, Y, Z axes.
Plane to perform tool nose radius compensation is selected with G17, G18, G19.

G17 :	Xp-Yp plane	Xp :	X axis or the parallel axis
G18 :	Zp-Xp plane	Yp :	Y axis or the parallel axis
G19 :	Yp-Zp plane	Zp :	Z axis or the parallel axis

Parameters are used to set which parallel axis of the X, Y, Z axes is to be the additional axis.

• Interference check

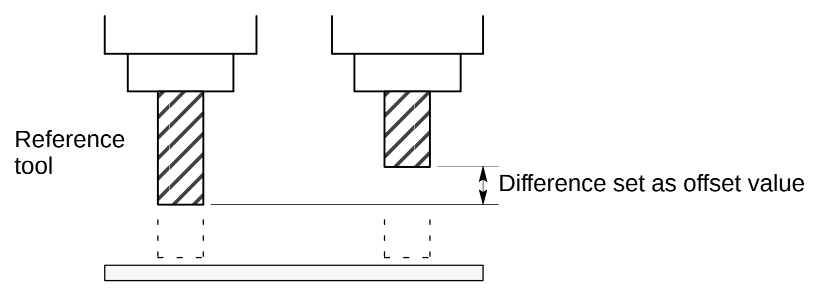
Tool overcutting is called ’interference’. This function checks whether interference occurs, if tool nose radius compensation is performed.



14.3 TOOL LENGTH COMPENSATION (G43, G44, G49)

M series

By setting the difference between tool length assumed when programming and the actual tool length as offsets, workpiece can be machined according to the size commanded by the program, without changing the program.



Explanations

- **Tool length compensation and its cancellation (G43, G44, G49)**

G43 : Tool length compensation +
G44 : Tool length compensation –
G49 : Tool length compensation cancel

In G43 mode, the tool is offset to the + direction for the preset tool length offset amount. In G44 mode, it is offset to the - direction for the preset tool length offset amount. G49 cancels tool length compensation.

- **Tool length compensation axis**

Tool length compensation can be performed for three types of axes. Compensation for the Z axis is tool length compensation A. That for the axis vertical to the selected plane is tool length compensation B. That for the axis specified by the G43 or G44 block is tool length compensation C. Which compensation to perform can be selected by a parameter.

- **Assignment of offset amount (H code)**

The offset amount can be set in the tool length compensation memory. By specifying an offset number with the H code, offset amount loaded in corresponding tool length compensation memory is used as tool length compensation amount.

Format

Tool length compensation A

$$\left\{\begin{matrix}\mathbf{G43}\\\mathbf{G44}\end{matrix}\right\}\ \mathbf{H\_\ ;}$$

Tool length compensation B

$$\left\{\begin{matrix}\mathbf{G17}\\\mathbf{G18}\\\mathbf{G19}\end{matrix}\right\}\left\{\begin{matrix}\mathbf{G43}\\\mathbf{G44}\end{matrix}\right\}\ \mathbf{H\_\ ;}$$

Tool length compensation C

$$\left\{\begin{matrix}\mathbf{G43}\\\mathbf{G44}\end{matrix}\right\}\ \alpha\_\ \mathbf{H\_\ ;}\ (\alpha\text{: arbitrary 1 axis})$$

14.4 TOOL OFFSET (G45, G46, G47, G48)

M series

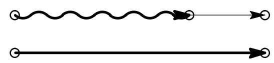
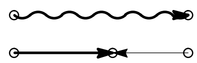
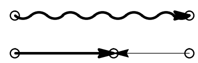
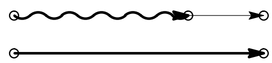
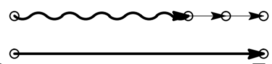
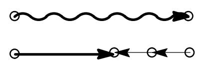
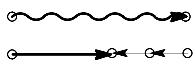
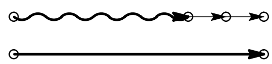
The programmed tool movement can be expanded or reduced for offset amount preset in the tool length compensation memory, by using this function.

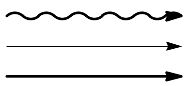
Explanations

- **G45, G46, G47, G48**

G45: Tool offset expansion
G46: Tool offset reduction
G47: Tool offset double expansion
G48: Tool offset double reduction

By commanding G45 - G48, expansion, reduction, double expansion, double reduction to axis move commanded in the program can be performed for the offset amount preset in the tool length compensation memory. The same offset amount is applied to all move command axes in the same block as G45 - G48.

G code	Tool offset value is positive	Tool offset value is negative
G45	 Start point End point	 Start point End point
G46	 Start point End point	 Start point End point
G47	 Start point End point	 Start point End point
G48	 Start point End point	 Start point End point


Movement distance
Tool offset value
Actual movement distance

- **Assignment of offset amount (D code)**

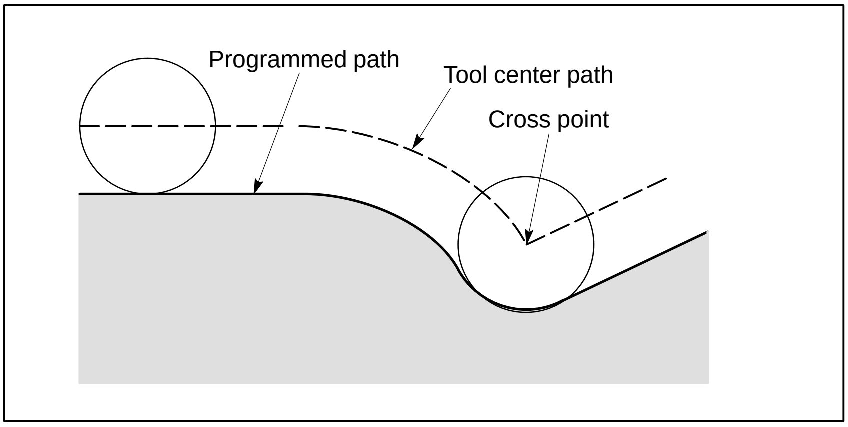
The offset amount can be set in the tool length compensation memory. By commanding an offset number with the D code, offset amount corresponding to the number in the tool length compensation memory is used as tool offset amount.

14.5 CUTTER COMPENSATION

M series

14.5.1 Cutter Compensation C (G40 - G42)

With this function, the programmed tool path can be offset when actually machining, for value of the tool radius set in the CNC.
By measuring cutting radius for actual cutting, and setting the value in the NC as offset value, the tool can machine the programmed pattern, via the offset path. There is no need to change the program even when tool radius changes; just change the offset value.



Cross points of line and line, arc and arc, line and arc is automatically calculated in the CNC to obtain offset actual tool path. So, programming becomes simple, because it is only necessary to program the machining pattern.

- **Cutter compensation and its cancellation (G40, G41, G42)**

G40 : Cutter radius compensation cancel
G41 : Cutter radius compensation left
G42 : Cutter radius compensation right
G41 and G42 are commands for cutter radius compensation mode. The cutter is offset to the left forward in the cutter movement in G42 and right forward in G42. Cutter radius compensation is cancelled with G40.

- **Assignment of offset amount (D code)**

The offset amount can be set in the cutter radius compensation memory. When the D code is commanded as an offset number, corresponding offset amount in the tool compensation memory is applied as the offset amount for cutter radius compensation.
The offset can be specified with an H code when the parameter is set accordingly.

- **Plane selection (G17, G18, G19)**

Cutter radius compensation is done on XY, ZX, YZ planes and on parallel axes of X, Y, Z axes.
Plane to perform cutter radius compensation is selected with G17, G18, G19.

G17 : Xp-Yp plane
G18 : Zp-Xp plane
G19 : Yp-Zp plane

where

Xp : X axis or its parallel axis
Yp : Y axis or its parallel axis
Zp : Z axis or its parallel axis

Parameters are used to set which parallel axis of the X, Y, Z axes is to be the additional axis.

Plane to perform cutter radius compensation is decided in the axis address commanded in the G17, G18, or G19 block.

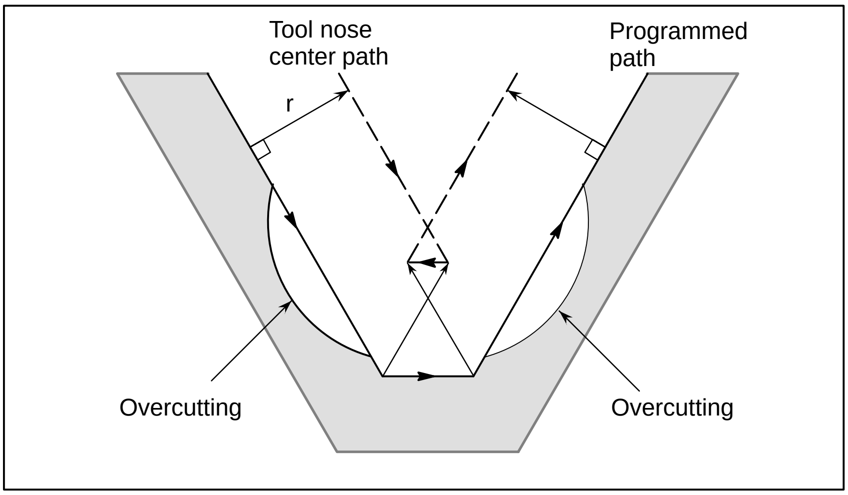
Example)
(U, V, W axes are parallel axes of X, Y, Z axes respectively)

G17 X_ ; XY plane
G17 U_ W_ ; UV plane
G19 Y_ W_ ; YW plane

If axis address of Xp, Yp, or Zp was omitted, compensation plane is decided regarding that X, Y, or Z was omitted.

- **Interference check**

Tool overcutting is called 'interference'. This function checks whether interference occurs, if cutter radius compensation is performed.



14.6 TOOL COMPENSATION MEMORY

14.6.1 Tool Compensation Memory

M series

Tool compensation memory C is provided as tool compensation amount memory.
Tool offset amount range which can be set is as follows:

Increment system	Geometry compensation		Tool wear compensation	
	Metric input	Inch input	Metric input	Inch input
IS-B	±999.999mm	±99.9999inch	±99.999mm	±9.9999inch
IS-C	±999.9999mm	±99.9999inch	±99.9999mm	±9.99999inch

Explanations

- **Tool compensation memory C**

Memory for geometry compensation as well as tool wear compensation is prepared separately in tool compensation memory C. Geometry compensation and tool wear compensation can thus be set separately. Separate memories are prepared for cutter radius compensation (for D code) and for tool length compensation (for H code).

Example

Offset number	For D code		For H code	
	Geometry compensation	Wear compensation	Geometry compensation	Wear compensation
001	10.0	0.1	100.0	0.1
002	20.0	0.2	300.0	0.3

14.6.2 Tool Offset Amount Memory

T series

Wear compensation memory and geometry compensation memory are provided as tool compensation amount memory.
Tool offset amount range which can be set is as follows:

Increment system	Tool compensation value	
	Metric input (mm)	Inch input (inch)
IS-B	–999.999 to +999.999	–99.9999 to +99.9999
IS-C	–999.9999 to +999.9999	–99.99999 to +99.99999

The maximum wear compensation value can, however, be modified using a parameter.

The number of digits used to specify a tool geometry/wear compensation value can be expanded by selecting the option which enables seven–digit tool offset specification. When this option is used, tool compensation values can be specified using up to seven digits for IS–B and eight digits for IS–C. The valid data range for tool compensation values will thus be as listed in the following table.

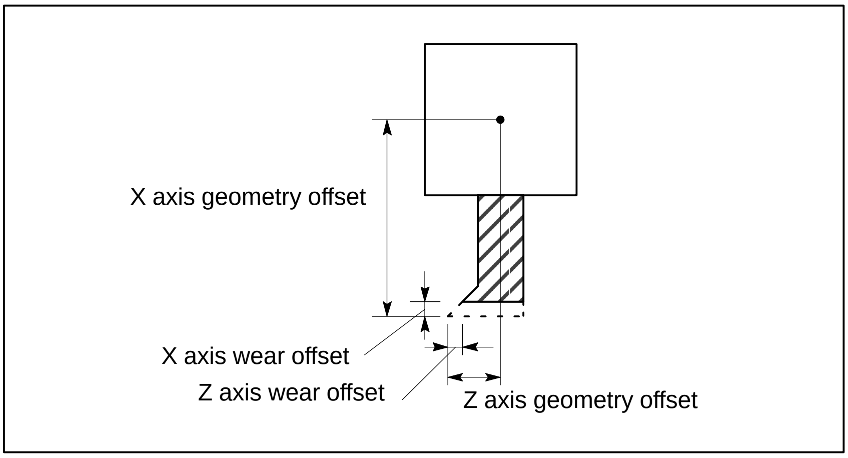
Increment system	Tool compensation value	
	Metric input (mm)	Inch input (inch)
IS-B	0 to ±9999.999	0 to ±999.9999
IS-C	0 to ±9999.9999 (0 to ±4000.0000)	0 to ±999.99999 (0 to ±160.00000)

NOTE

The range enclosed in parentheses applies when automatic inch/metric conversion is enabled.

- **Tool geometry/wear compensation**

Memory for geometry offset and tool wear offset is prepared separately. Geometry offset and tool wear offset can thus be set separately.



Offset number	X axis offset value		Z axis offset value		Tool nose radius compensation value		Imaginary tool nose number
	Geometry offset	Wear offset	Geometry offset	Wear offset	Geometry offset	Wear offset	
01	10.0	0.0	100.0	0.1	0.4	0.1	3
02	20.2	0.2	150.0	0.3	0.5	0.2	2
03	30.4	0.4	200.0	0.5	1.2	–0.2	8

14.7 NUMBER OF TOOL OFFSETS

14.7.1 Number of Tool Offsets

M series

- **400 tool offsets**
  Offset numbers (D code/H code) 0 - 400 can be used.
  D00 - D400 or H00 - H400

14.7.2 Number of Tool Offsets

T series

- **64 tool offsets**
  Offset numbers 0 - 64 can be used.

14.8 CHANGING OF TOOL OFFSET AMOUNT (PROGRAMMABLE DATA INPUT) (G10)

Tool offset amount can be set/changed with the G10 command.
When G10 is commanded in absolute input (G90), the commanded offset amount becomes the new tool offset amount. When G10 is commanded in incremental input (G91), the current tool offset amount plus the commanded offset amount is the new tool offset amount.

Format (M series)

- **Tool compensation memory C**

Setting/changing of geometry offset amount for H code

G10 L10 P_ R_ ;

Setting/changing of geometry offset amount for D code

G10 L12 P_ R_ ;

Setting/changing of tool wear offset amount for H code

G10 L11 P_ R_ ;

Setting/changing of tool ware offset amount for D code

G10 L13 P_ R_ ;

NOTE
L1 may be used instead of L11 for the compatibility with the conventional CNC's format.

Format (T series)

G10 P_ X_ Z_ R_ Q_ ;

or

G10 P_ U_ W_ C_ Q_ ;

P : Offset number
1–64 :Tool wear offset number
10000+(1–64) : Tool geometry offset number+10000
X : Offset value on X axis (absolute)
Z : Offset value on Z axis (absolute)
U : Offset value on X axis (incremental)
W : Offset value on Z axis (incremental)
R : Tool nose radius offset value (absolute)
R : Tool nose radius offset value (incremental)
Q : Imaginary tool nose number

In an absolute command, the values specified in addresses X, Z, and R are set as the offset value corresponding to the offset number specified by address P. In an incremental command, the value specified in addresses U, W, and C is added to the current offset value corresponding to the offset number.

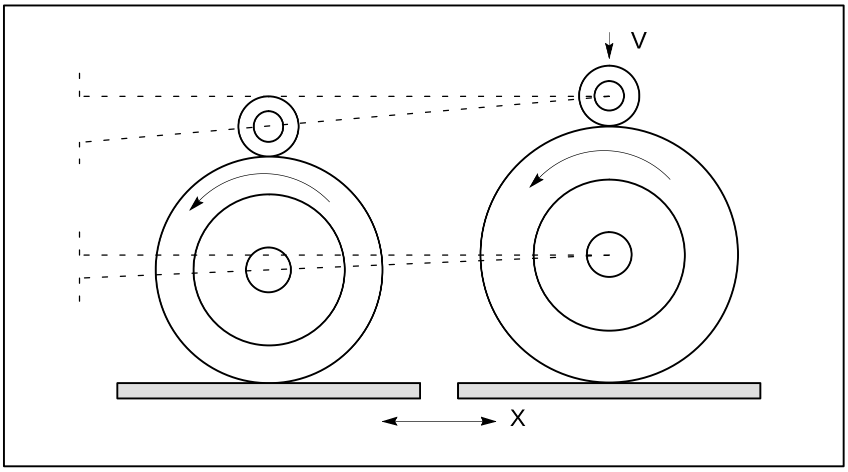
NOTE

1 Addresses X, Z, U, and W can be specified in the same block.
2 Use of this command in a program allows the tool to advance little by little. This command can also be used input offset values one at a time from a tape by specifying this command successively instead of inputting these values one at a time from the MDI unit.

14.9 GRINDING-WHEEL WEAR COMPENSATION BY CONTINUOUS DRESSING

M series

The grinding-wheel cutting and dresser cutting are compensated continuously during grinding in the canned cycles for surface grinding (G75, and G77 to G79). They are compensated according to the amount of continuous dressing.



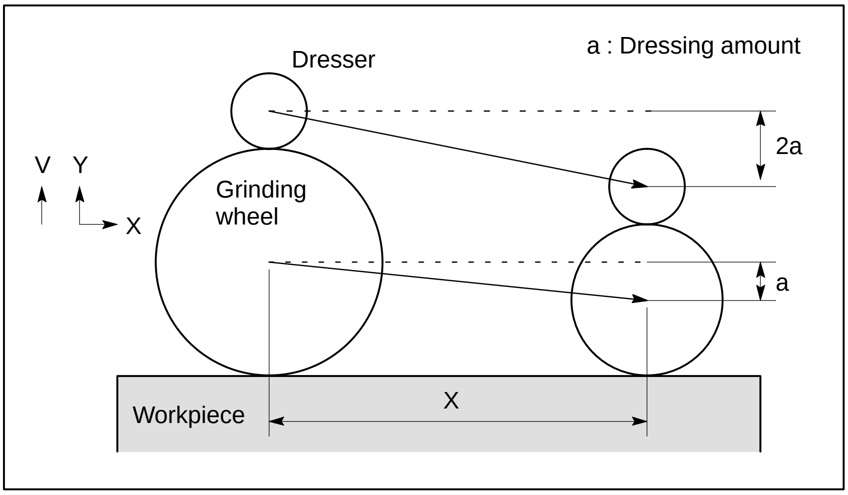
Explanations

• Specification

The offset number (grinding-wheel wear compensation number) is specified by address L in the block for the canned cycles for surface grinding. The compensation amount set in the offset memory corresponding to the specified number is the dressing amount.

• Compensation

Compensation is performed for each cutting operation (each X-axis movement) in the canned cycles for grinding. Along with X-axis movement, compensation is performed both in Y-axis direction (grinding-wheel cutting) and that in V-axis direction (dresser cutting). That is, compensation is performed for interpolation for the three coordinates simultaneously. The Y-axis movement amount (compensation amount) is the specified dressing amount. The V-axis movement amount is twice the specified dressing amount (diameter).



15 ACCURACY COMPENSATION FUNCTION

15.1 STORED PITCH ERROR COMPENSATION

The errors caused by machine position, as pitch error of the feed screw, can be compensated. This function is for better machining precision. As the offset data are stored in the memory as parameters, compensations of dogs and settings can be omitted. Offset intervals are set constant by parameters (per axis).

This function can perform compensation as described below.

- Compensation period: Constant period for each axis (parameter settings (for each axis))
- Compensation points: Total of 1,024 points
  Optional distribution to each axis (parameter settings (for each axis))
- Amount of compensation: The following compensation pulse is output at each compensation point.

  Compensation pulse = (–7 to +7) × (compensation magnification)

  Unit: Same as the detection unit

  –7 to +7: Setting for each compensation point (setting within the range of –7 to +7)

  Compensation magnification: 1 to 100 times
  Constant magnification for each axis (parameter settings (for each axis))

15.2 BI–DIRECTIONAL PITCH ERROR COMPENSATION

In bi–directional pitch error compensation, different pitch error compensation amounts can be set for travel in the positive direction and that in the negative direction, so that pitch error compensation can be performed differently in the two directions, in contrast to stored pitch error compensation, which does not distinguish between the directions of travel. In addition, when the direction of travel is reversed, the compensation amount is automatically calculated from the pitch error compensation data to perform compensation in the same way as in backlash compensation. This reduces the difference between the paths in the positive and negative directions.

15.3 BACKLASH COMPENSATION

This function is used to compensate lost motions proper to the machine system. Offset amounts come in a range of 0 to ±9999 pulses per axis, and is set as parameters in detection unit.

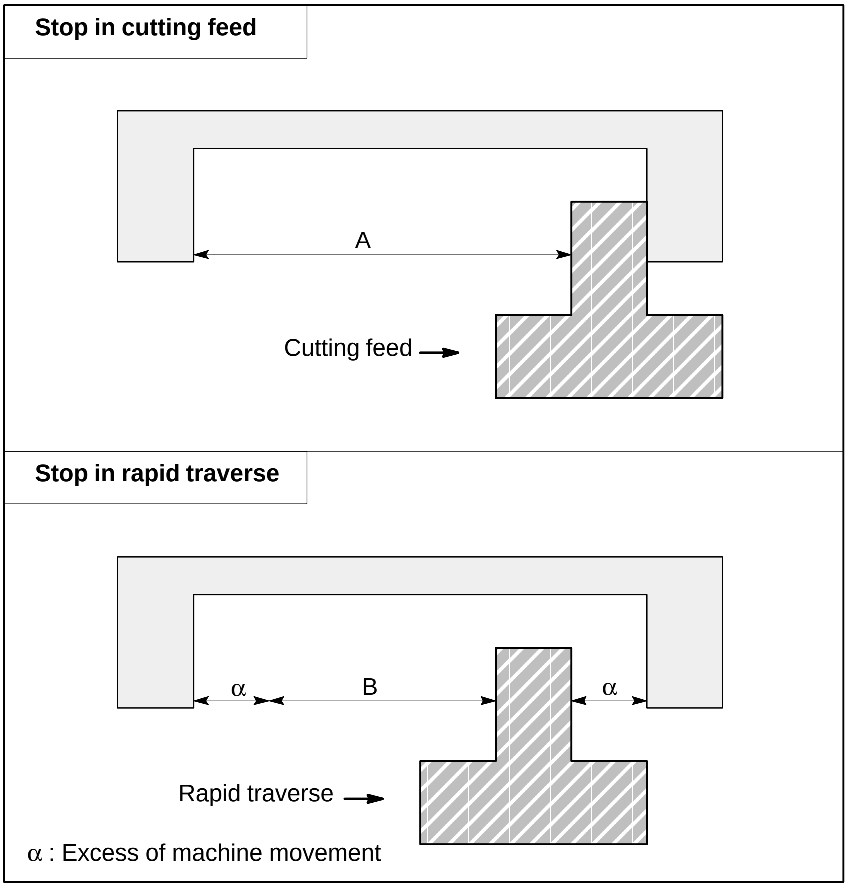
15.4 BACKLASH COMPENSATION FOR EACH RAPID TRAVERSE AND CUTTING FEED

Since different backlash compensation values can be used for cutting feed and rapid traverse, the machining precision is improved.
The following table shows backlash amounts according to the feedrate and movement direction. In the table, the backlash compensation amount for cutting feed is A, and that for rapid traverse is B. A and B are set in parameters.

	Cutting feed ↓ Cutting feed	Rapid traverse ↓ Rapid traverse	Rapid traverse ↓ Cutting feed	Cutting feed ↓ Rapid traverse
Movement in same direction	0	0	$\pm\alpha$	$\pm(-\alpha)$
Movement in opposite directions	±A	±B	$\pm(B+\alpha)$	$\pm(B+\alpha)$

*1) $\alpha=(A-B)/2$

*2) Sings (+ and –) indicate directions.



15.5 PROGRAMMABLE PARAMETER INPUT (G10, G11)

Parameters and pitch errors data can be set by programs. Therefore, following uses can be done example.

- Parameter setting such as pitch errors compensation data, etc. When the attachment is replaced.
- Parameters such as max. Cutting speed and cutting feed time constant can be changed according to the machining conditions.

Format

```
G10 L50 ;
N_ R_ ;       Input of parameters except axis type
N_ P_ R_ ;    Input of axis type parameters
   :
G11 ;
```

G10 L50	:	Parameter input mode
G11	:	Parameter input mode cancel
N_	:	Parameter No. (or pitch error data No.+10000)
P_	:	Axis No. (in the case of axis type parameter)
R_	:	Parameter setting value (or pitch error data)

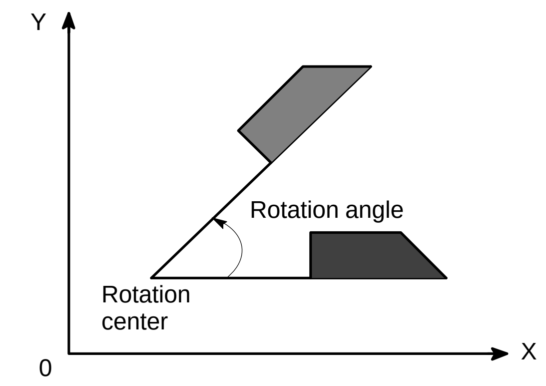
NOTE
Some parameters cannot be set.

16 COORDINATE SYSTEM CONVERSION

16.1 COORDINATE SYSTEM ROTATION (G68, G69)

M series

Patterns specified by the program can be rotated. For example, by using this function, when the attached workpiece comes in a position which is somewhat rotated from the machine coordinates, the position can be compensated by the rotation instruction.
If a pattern is similar to that made by rotating a programmed figure, the program for the pattern can be created by calling the program for the figure as a sub program, and rotating the coordinates in the program. This function reduces the programming time and program length.



Format

M series

$\left\{\begin{array}{l}\textbf{G17}\\ \textbf{G18}\\ \textbf{G19}\end{array}\right\}$ **G68** α_ β_ **R_ ;**

⋮ Coordinate system rotation mode (Coordinate system rotates.)

G69 ; Coordinate system rotation cancel

α, β : Coordinate valne of rotation center specify two axes from X, Y, Z axes of G17, G18, G19.(These are always absolute values.)

R : Rotation (+ for the counterclockwise direction. Specified in absolute value. It can be also specified in incremental values according to the parameter setting.)

By this command, commands thereafter are rotated in the angle commanded by R, with the point commanded by α, β as the rotation center. Rotation angle is commanded in 0.001 x units in a range of:

$-360000 \leqq R \leqq 360000$

The rotation plane is the plane selected (G17, G18, G19) when G68 was commanded.
G17, G18 and G19 may not be commanded in the same block as G68.
When α, β is omitted, the point where G68 was commanded becomes the rotation center.

G69; Cancels the coordinate system rotation.

16.2 SCALING (G50, G51)

M series

Scaling can be commanded to figures commanded in the machining programs.

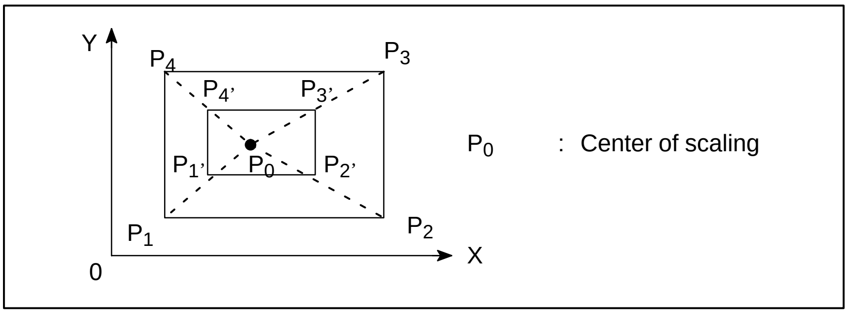
Format

When each axis is scaling of the same magnification	
Format	**Sign explanation**
G51 X_ Y_ Z_ P_ ; Scaling start ⋮ Scaling is effective. (Scaling mode) **G50 ;** Scaling cancellation	X_Y_Z_ : Absolute command of center coordinate value of scaling P_ : Magnification of scaling

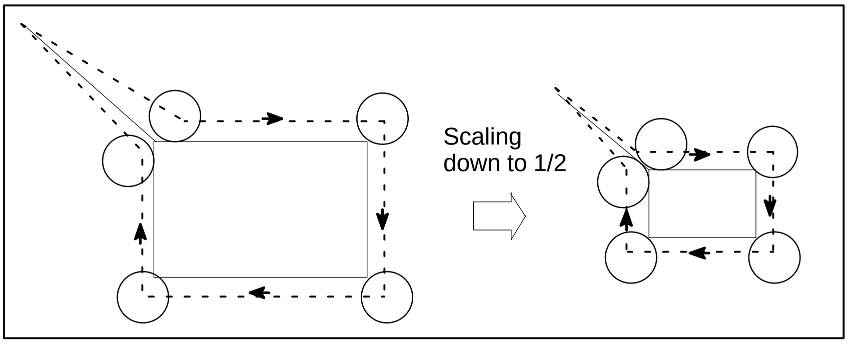
By this command, scaling of the magnification specified by P is commanded with the point commanded by X, Y, Z as its center. G50 cancels to scaling mode.

G50 : Scaling mode cancel
G51 : Scaling mode command

Commandable magnification is as follows :
0.00001 - 9.99999 times or 0.001 - 999.999



If P was not commanded, the magnification set by parameters is applied. When X, Y, Z are omitted, the point where G51 was commanded becomes the center of scaling.
Scaling cannot be done to offset amounts such as tool length compensation, cutter radius compensation, or tool offset.

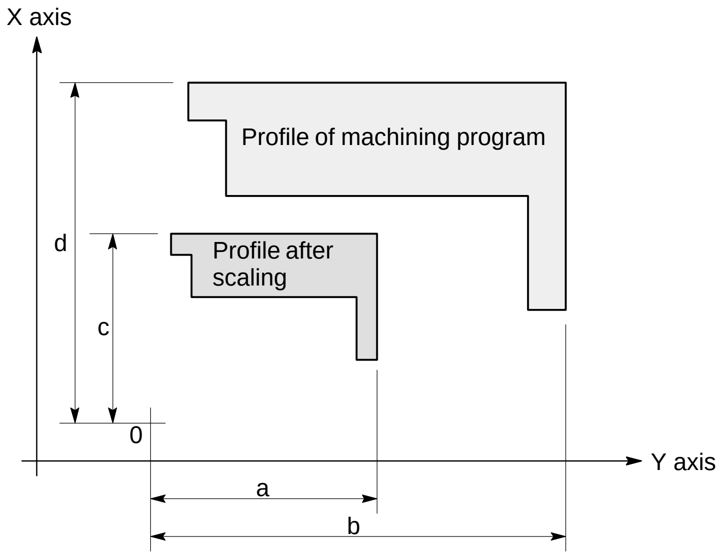


A scaling magnification can be set for each axis or for all axes in common. A parameter can specify whether it should be set for each axis or for all axes.

Format

Scaling of each axis (Mirror image)	
Format	**Sign explanation**
G51 X_ Y_ Z_ I_ J_ K_ ; Scaling start ⋮ Scaling is effective. (Scaling mode) **G50 ;** Scaling cancellation	X_Y_Z_ : Absolute command of center coordinate value of scaling I_J_K_ : Magnification of scaling of X axis, Y axis, and Z axis (Unit 0.001 or 0.00001 is selected according to the parameter.) The magnification which can be instructed is as follows. ±0.00001–±9.99999 or ±0.001–±999. 999

If magnifications I, J, or K are not specified, the magnification of each axis set by a parameter is used.

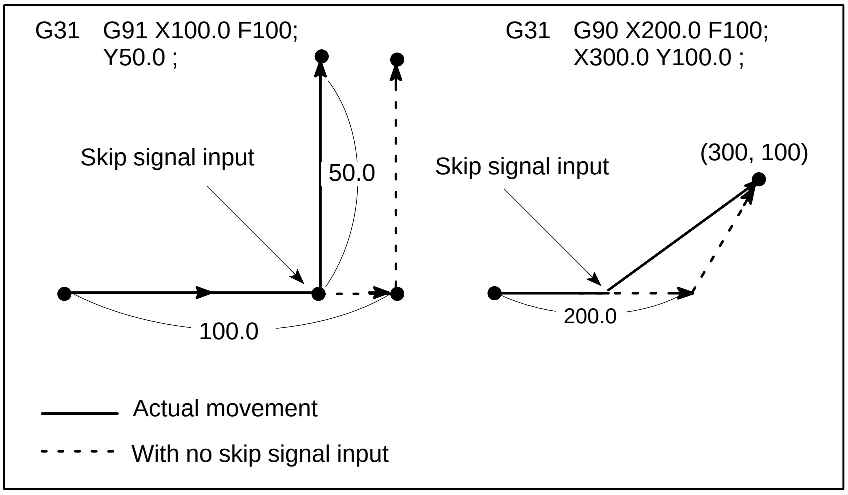


a/b : X-axis scaling magnification
c/d : Y-axis scaling magnification
o : Center of scaling

17 MEASUREMENT FUNCTIONS

17.1 SKIP FUNCTION (G31)

By commanding axis move after G31, linear interpolation can be commanded like in G01. If an external skip signal is input during this command, the remainder of this command is cancelled, and program skips to the next block.
G31 is a one-shot command and is valid for the commanded block only.



Coordinate value when skip signal is on, is stored in the system variables #5061 - #5068 of the customer macro, so this function can also be read with the customer macro function.

#506n : ntn axis skip signal position (n=1–8)

As the skip function can be used when move amount is not clear, this function can be used for:

- Constant feed in grinding machines
- Tool measurement with tactile sensor.

17.2 MULTI-STEP SKIP FUNCTION (G31 P1 - G31 P4)

n blocks with either of P1 to P4 following G31 commanded, the coordinate value where skip signals (4 types) were input is stored in the custom macro variables, and at the same time, the remaining movement of the block is skipped. It is also possible to skip the remaining dwell with the skip signal by parameter, in a block where: G04 is commanded (dwell).
Parameters decide which skip command or dwell command is valid to which of the four skip signals. The skip signal is not necessarily unique to a single skip command or dwell command; it is also possible to set a skip signal to multiple skip command or dwell commands.

17.3 HIGH-SPEED SKIP SIGNAL INPUT

Delay and error of skip signal input is 0–2 msec at the NC side (not considering those at the PMC side).
This high-speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement. This signal is connected directly to the NC; not via the PMC.

17.4 TORQUE LIMIT SKIP (G31 P99, G31 P98)

T series

With the motor torque limited (for example, by a torque limit command, issued through the PMC window), a move command following G31 P99 (or G31 P98) can cause the same type of cutting feed as with G01 (linear interpolation).
Skip operation is performed when the motor torque reaches the limit, when the tool is pushed back for example, during cutting feed.
For details of how to use this function, refer to the manuals supplied by the machine tool builder.

Format

G31 P99 IP _ F_ ;

G31 P98 IP _ F_ ;

G31 : One–shot G code (G code effective only in the block in which it is issued)
P99 : Skip operation is performed when the motor torque reaches the limit or the skip signal is input.
P98 : Skip operation is performed only when the motor torque reaches the limit (regardless of the skip signal).

17.5 TOOL LENGTH AUTOMATIC MEASUREMENT (G37)

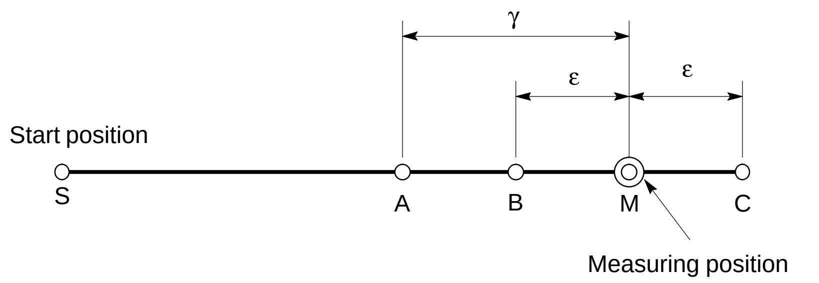
M series

Difference between the coordinate value of tool when tool end has reached the measuring position and coordinate value of the measuring position is automatically measured, calculated, and added to the currently set tool offset amount by CNC system. The machine must be equipped with measuring devices, for example tactile sensor, so that a signal is sent when the tool end has reached the measuring position.
Measuring position coordinate value is commanded as follows:

Format

G37 α_ **;**

α : The measuring position is commanded in by either X, Y, or Z.



γ : Deceleration point (parameter)
ε : Allowable measuring range (parameter)

The tool is moved from the start position to the deceleration point A in rapid traverse, tool speed is decelerated to the measurement speed preset by parameter, and moved on till the measuring position reach signal is output. In case measuring position reach signal is not output in the allowable measuring range (from point B to C), and alarm arises.
(New offset amount) = (Old offset amount) + (Measuring position reach signal detected position) - (measuring position)

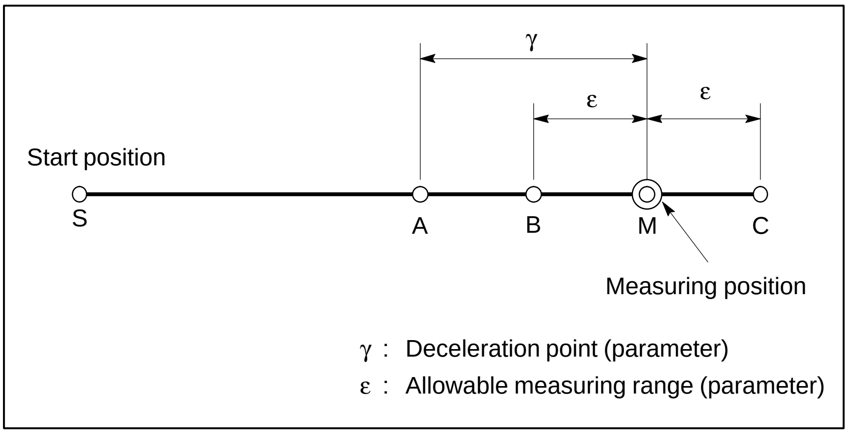
17.6 AUTOMATIC TOOL OFFSET (G37, G36)

T series

Difference between the coordinate value of tool when tool end has reached the measuring position and coordinate value of the measuring position is automatically measured, calculated, and added to the currently set tool offset amount by CNC system. The machine must be equipped with measuring devices, for example tactile sensor, so that a signal is sent when the tool end has reached the measuring position.
Measuring position coordinate value is commanded as follows:

Format

G36 X_ ;
or
G37 Z_ ;

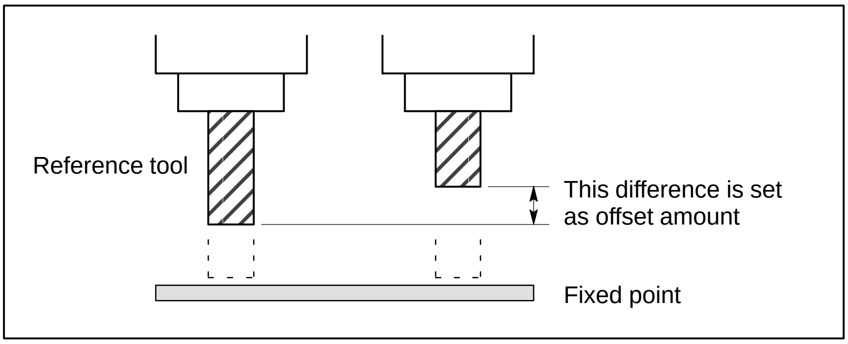


The tool is moved from the start position to the deceleration point A in rapid traverse, tool speed is decelerated to the measurement speed preset by parameter, and moved on till the measuring position reach signal is output. In case measuring position reach signal is not output in the allowable measuring range (from point B to C), and alarm arises.
(New offset amount) = (Old offset amount) + (Measuring position reach signal detected position) - (measuring position)

17.7 TOOL LENGTH MEASUREMENT

M series

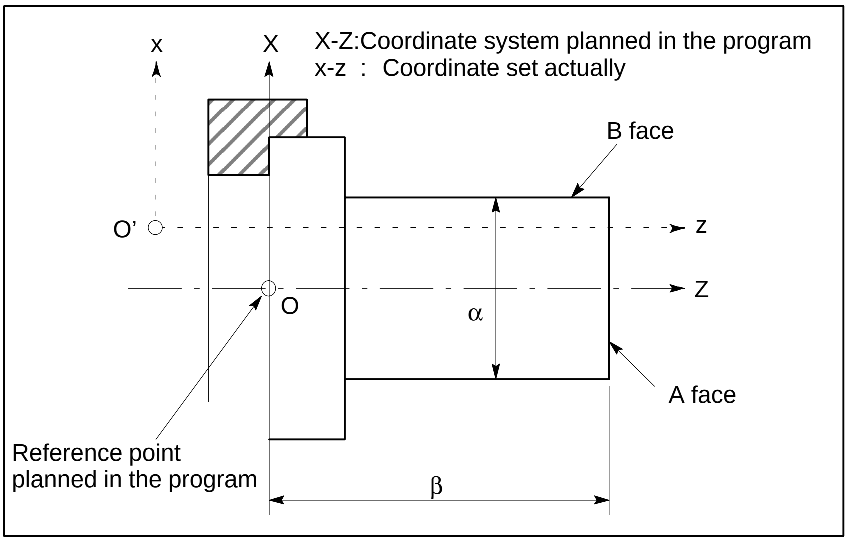
The value displayed as a relative position can be set in the offset memory as an offset value by a soft key.
Call offset value display screen. Relative positions are also displayed on this screen. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by hand. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts.



17.8 DIRECT INPUT OF TOOL OFFSET VALUE MEASURED/DIRECT INPUT OF WORKPIECE COORDINATE SYSTEM SHIFT AMOUNT

T series

This is a function of setting an offset value by key-inputting a workpiece diameter manually cut and measured from the MDI keyboard.
First the workpiece is cut in the longitudinal or in the cross direction manually. When a button on the machine operator's panel is pressed upon completion of the cutting, the work coordinate value at that time is recorded. Then, withdraw the tool, stop the spindle, and measure the diameter if the cutting was on the longitudinal direction or distance from the standard face if it was on the facing. (The standard face is made as Z = 0.) When the measured value is entered into the offset number desired plus 100, NC inputs the difference between the input measured value and the coordinate value recorded in NC, as the offset value of the offset number.
The work coordinate system can be shifted using the technique of directly inputting the measured value for offset. This technique is used when the coordinate system planned in the program does not match with the coordinate system set by the G92 command or by the automatic coordinate system setting.
The procedures are the same as those for direct input for offset, except a difference of using the standard tool.



Cut A or B face and measure β or α. Direct input the measured value.

17.9 DIRECT INPUT OF TOOL OFFSET VALUE MEASURED B

T series

By installing the touch sensor and by manually making the tool contact the touch sensor, it is possible to set the offset amount of that tool automatically in the tool offset amount memory. It is also possible to set the work coordinate system shift amount automatically.
In addition, a tool compensation value measured value direct input B function for two–spindle lathes is provided so that the tool compensation value measured value direct input B function can be used for both spindles of a one–turret two–spindle lathe.

Explanations

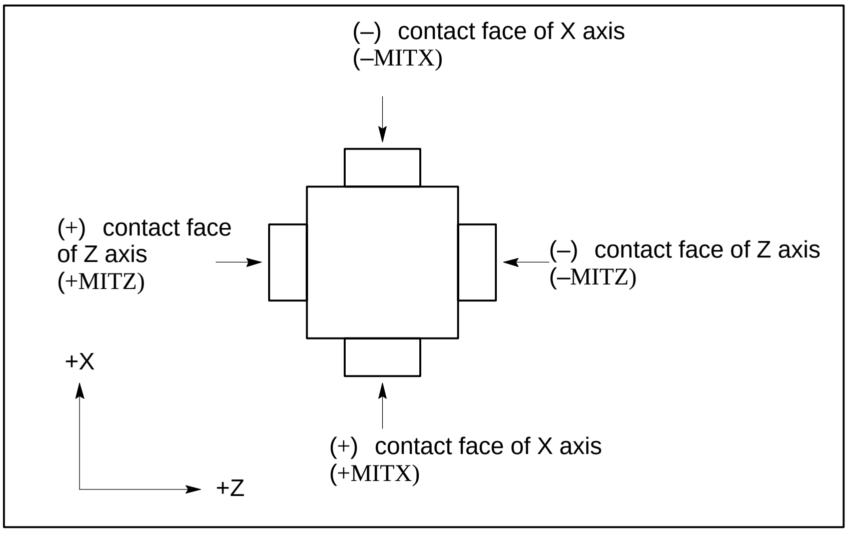
- **Touch sensor**

As the touch sensor detection mode, either four–contact input mode or single–contact input mode can be selected.

1) Four–contact input (when bit 3 (TS1) of parameter No. 5004 is set to 0)
The touch sensor has contact faces in two directions along each axis, and outputs four signals when a touch is detected. These signals are input to the CNC as tool compensation value writing signals (+MIT1, +MIT2, –MIT1, and –MIT2).
When any of these input signals is input, the CNC stops feed in the corresponding direction along the corresponding axis.

2) Single–contact input (when bit 3 (TS1) of parameter No. 5004 is set to 1)
The touch sensor outputs one signal when a touch by a single–contact input is detected. This signal is input to the CNC as the tool compensation value writing signal (+MIT1).
Then, the CNC determines the two directions along each axis automatically and stops feed in the corresponding direction along the corresponding axis.

Signal	Parameter	
	TS1=0	TS1=1
+MIT1	Contact the (+) contact face of the X–axis (Contact in the X+ direction)	The two directions along each axis is determined automatically.
–MIT1	Contact the (–) contact face of the X–axis (Contact in the X– direction)	Not used
+MIT2	Contact the (+) contact face of the Z–axis (Contact in the Z+ direction)	Not used
–MIT2	Contact the (–) contact face of the Z–axis (Contact in the Z– direction)	Not used



- **Setting method**

❑ **Setting of tool compensation value**

Previously set the distance from the measurement reference position (a particular point on the machine) to the measuring position (the touch sensor contact face) to the parameter as the reference value.
As the tool of which the offset amount is to be measured is selected and is positioned at the measuring position (contact the touch sensor), the contact detection signal (tool compensation value write signal) from the touch sensor is received, and the difference between the machine coordinate value at that time (= the distance from the measured tool nose tip position at the machine reference position (machine zero point) to the measuring position) and the reference value (parameter value) is set in the tool offset amount memory as the tool geometry offset amount of that tool. The corresponding tool wear offset amount becomes zero.

[**Tool offset amount to be set**] = [**Machine coordinate value when tool compensation value write signal turns "1"**] + [**Reference value (value of parameter) corresponding to tool compensation value write signal**]

The tool offset amount to be set depends on how to determine the measurement reference position.

Supplement : When single–contact input (when bit 3 (TS1) of parameter No. 5004 is set to 1) is set for touch detection in the touch sensor

When receiving the touch detection signal (tool compensation value writing signal +MIT1) from the touch sensor, the CNC determines the two directions along each axis automatically according to the number of pulses stored for the movement along each axis that has been made until the input of the signal. (Set the number of interpolation cycles of stored pulses in parameter No. 5021 in advance.)

After determining the two directions along each axis automatically, the CNC applies axis interlock to the corresponding axis direction to stop feed operation, and the calculated tool compensation value is stored in tool compensation memory.

When stored pulses show various directions, when the servo power is shut down (the servo off state), or when no pulse is stored because no axis movement has taken place, the direction cannot be determined, so P/S alarm No. 5195 is generated.

Also when the tool moves along two axes (the X–axis and Z–axis), the P/S alarm is generated. So, the tool must be moved along just one axis.

If the P/S alarm is generated, the tool compensation value is not set, and the two axes and four directions are interlocked.

NOTE

1 Pulses used for automatic determination are stored while the tool compensation value writing mode select signal GOSQM <G039#7> is 1 in manual mode. These pulses are lost when:
   a. A mode other than manual mode is set.
   b. The tool compensation value writing mode select signal GOSOM <G039#7> is driven to 0.
   c. The tool compensation value is set upon reception of the touch detection signal from the touch sensor, or P/S alarm No. 5195 is generated.
   d. The servo off state is entered. In this case, the pulses stored for the axis for which the servo off state is entered are lost.
   e. Axis movement is performed. In this case, pulses stored for the other axis along which no movement takes place are lost.

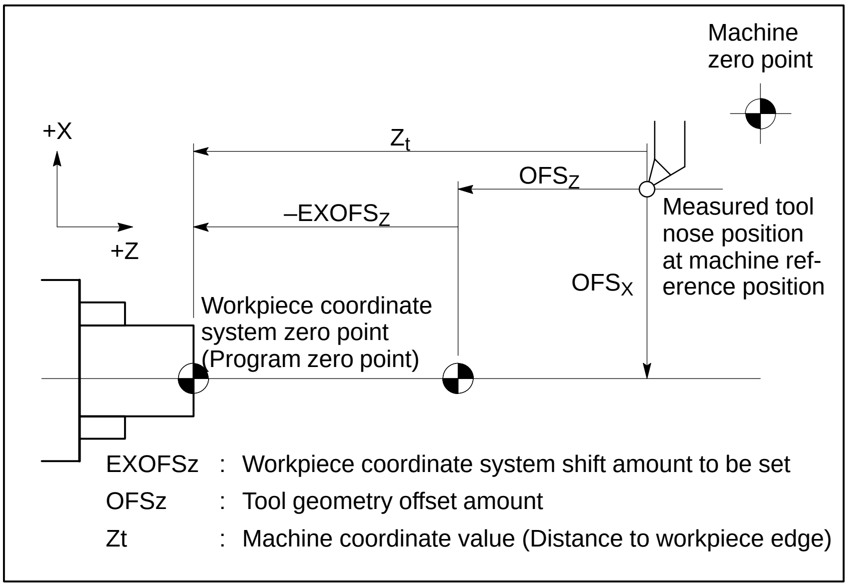
2 Axis interlock applied in the axis direction determined automatically and two–axis four–direction interlock applied due to a P/S alarm are released when a mode other than manual mode is set or when the tool compensation value writing mode select signal GOSQM <G039#7> is driven to 0. This type of interlock is not released by a reset operation.

❑ Setting of workpiece coordinate system shift amount

The workpiece coordinate system shift amount along the Z axis is to be set as follows. When the tool touches the end face of the workpiece, the touch detection signal (workpiece coordinate system shift write signal) is output. This signal is used to set the workpiece coordinate system shift, calculated by subtracting the tool geometry compensation value (shift of coordinate system due to tool geometry compensation) from the current machine coordinate (distance between the end face of the workpiece and the tip of the measurement tool when it is at the machine reference position (machine zero point)). In this case the tool geometry offset amount corresponding to the tool must be programmed previously.

[Workpiece coordinate system shift amount along Z axis to be set (EXOFSz)] = **[Tool geometry offset amount of Z axis of corresponding tool (OFSz)]** + **[Machine coordinate value of Z axis (Zt)]**

By the above procedure the workpiece coordinate system with the workpiece edge (sensor contact point) being taken as the workpiece coordinate system zero point of the Z axis (the program zero point) is set when the tool is selected by the program command (T code).



17.10 TOOL OFFSET VALUE COUNT INPUT

T series

By manipulating soft keys, a position value displayed on the relative position display can be set to the offset memory.
Call offset value display screen on the screen. Relative positions are also displayed on this screen. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by hand. The relative position display at this point shows difference between the reference tool and the tool measured and the relative position display value is then set as offset amounts.

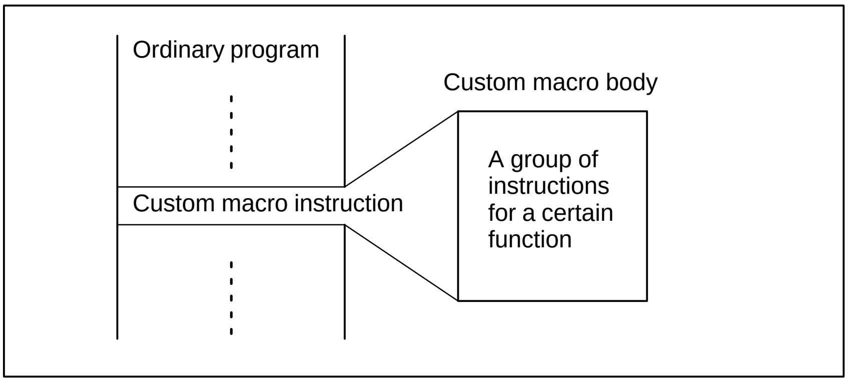
17.11 DIRECT INPUT OF WORKPIECE ORIGIN OFFSET VALUE MEASURED

By directly entering the measured deviation of the actual coordinate system from a programmed workpiece coordinate system, the workpiece zero point offset at the cursor is automatically set so that a command value matches the actual measurement.

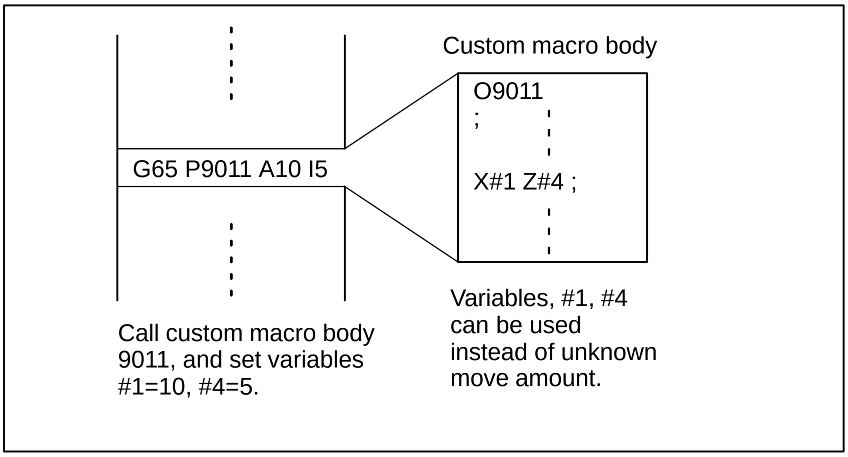
18 CUSTOM MACRO

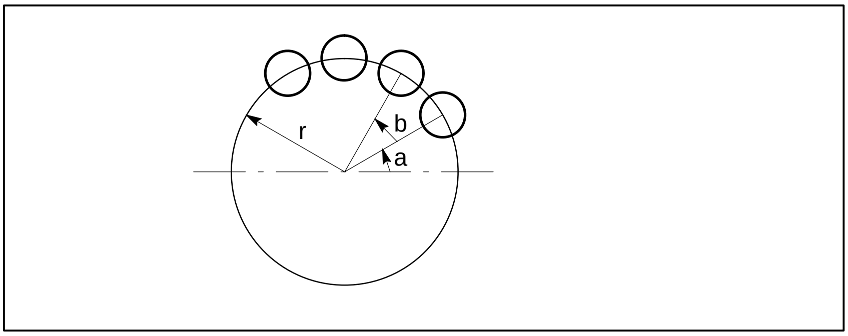
18.1 CUSTOM MACRO

A function covering a group of instructions is stored in the memory like the sub program. The stored function is represented by one instruction and is executed by simply writing the represented instruction. The group of instructions registered is called the custom macro body, and the representative instruction, the custom macro instruction.



The programmer need not remember all the instructions in the custom macro body. He needs only to remember the representative, custom macro instruction.
The greatest feature in custom macro is that variables can be used in the custom macro body. Operation between the variables can be done, and actual values can be set in the variables by custom macro instructions.





Bolt hole circle as shown above can be programmed easily. Program a custom macro body of a bolt hole circle; once the custom macro body is stored, operation can be performed as if the CNC itself has a bolt hole circle function. The programmer need only to remember the following command, and the bolt hole circle can be called any time.

Format

G65 Pp Rr Aa Bb Kk ;

p : Macro number of the bolt hole circle
r : Radius
a : Initial angle
b : Angle between holes
k : Number of holes

With this function, the CNC can be graded up by the user himself. Custom macro bodies may be offered to the users by the machine tool builder, but the users still can make custom macro himself.
The following functions can be used for programming the custom macro body.

Explanations

• Use of Variable

Variables: #1 (i=1, 2, 3,.......)
Quotation of variables: F#33 (#33: speed expressed by variables)

• Operation between variables

Various operation can be done between variables and constants.
The following operands, and functions can be used:
+ (sum), – (difference), * (product), / (quotient), OR (logical sum), XOR (exclusive logical sum), AND (logical product), SIN (sine), COS (cosine), TAN (tangent), ATAN (arc tangent), SQRT (square roots), ABS (absolute value), BIN (conversion from BCD to binary), BCD (conversion from binary to BCD), FIX (truncation below decimal point), FUP (raise fractions below decimal point), ROUND (round)
Example : #5 = SIN [[#2 + #4] ✻ 3.14 + #4] ✻ ABS (#10)

• Control command

Program flow in the custom macro body is controlled by the following command.

❑ **If [<conditional expression>]GOTO n (n = sequence number)**
When <conditional expression> is satisfied, the next execution is done from block with sequence number n.
When <conditional expression> is not satisfied, the next block is executed.
When the [<IF conditional expression>] is committed, it executes from block with n unconditionally.
The following <conditional expressions> are available:
#j EQ #k whether #j = #k
#j NE #k whether #j = #k
#j GT #k whether #j > #k
#j LT #k whether #j < #k
#j GE #k whether #j ≧ #k
#j LE #k whether #j ≦ #k

❑ **IF[<conditional expression>]THEN**
If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

❑ **WHILE (<conditional expression>) DO m (m = 1, 2, 3)**
:
END m

While <conditional expression> is satisfied, blocks from DO m to END m is repeated.
When <conditional expression> is no more satisfied, it is executed from the block next to
END m block.

```
Example
   #120 = 1 ;
   WHILE [#120 LE 10] DO 1 ;
   ┌─────────────────────┐
   │                     │
   │                     │  Repeated 10 times.
   │#120=#120+1 ;        │
   └─────────────────────┘
    END
```

- **Format of custom macro body**

The format is the same as the sub program.

```
O Macro number ;

        Custom macro body

M99 ;
```

- **Custom macro instruction**

❑ **Simple call**
G65 P (macro number) L (times to repeat) <argument assignment> ;

A value is set to a variable by <argument assignment>.
Write the actual value after the address.
Example A5.0E3.2M13.4
There is a regulation on which address (A - Z) corresponds to which variable number.

❑ **Modal call A**
G66 P (macro number) L (times to repeat) <argument assignment> ;

Each time a move command is executed, the specified custom macro body is called. This can be canceled by G67.
This function is useful when drilling cycles are programmed as custom macro bodies.

❑ Macro call by G codes

The macro can also be called by the parameter-set G codes. Instead of commanding:

N_ G65 P○○○○ <argument assignment> ;

macro can be called just by commanding:

N_ Gxx <argument assignment> ;.

G code for calling the macro, and macro program number **** to be called, are coupled together and set as parameter.

Maximum ten G codes from G01 to G9999 can be used for macro call (G00 cannot be used).

The G code macro call cannot be used in the macro which was called by a G code. It also cannot be used in sub programs called by sub program call with M codes or T codes.

❑ Macro call by M code

Custom macros can be called by pre-determined M codes which are set by parameters.

The following command

N_ G65 P○○○○ <Argument assignment> ;

is equivalent to the following command:

N_ Mxx <Argument assignment> ;

The correspondence between M codes (Mxx) and program number (delta delta delta delta) of a macro shall be set by a parameter.

Signal MF and M code are not sent out the same as the subprogram call by M code.

Also when this M code is specified in a program called a macro calling G code or a subprogram calling M or T code, the M code is regarded as a normal M code.

Up to ten M codes from M01 to M99999999 can be used for custom macro calling M codes.

❑ Sub program call by M code

An M code can be set by parameter to call a sub program. Instead of commanding:

N_ G_ X_ Y_ … M98 P○○○○ ; ,

the same operation can be performed simply by commanding:

N_ G_ X_ Y_ … Mxx ;.

As for M98, M codes are not transmitted.

The M code XX for calling the sub program and the sub program number delta delta delta delta to be called are coupled together and set by parameter.

Maximum ten M codes from M01 to M99999999 can be used for macro call.

Arguments cannot be transmitted. It also cannot be commanded in the same block as the block with M98 command.

When these M codes are commanded in macro called by G code or in subprogram called by M code or T code, they are regarded as ordinary M codes.

❑ **Sub program call by T code**

By setting parameter, sub program can be called by T codes. When commanded:

N_ G_ X_ Y_ … **Tt ;** ,

the same operation is done as when commanded:

#149 = t;

N_ G_ X_ Y_ … **M98 P9000;** .

The T type code t is stored as arguments of common variable #149. This command cannot be done in the same block with a sub program calling M code, or with M98 command. The T code is not output.

When T code is commanded in macros called by G code, or in sub programs called by M codes or T codes, the T code is treated as ordinary T codes.

- **Types of variables**

Variables are divided into local variables, common variables, and system variables, according to their variable numbers. Each type has different use and nature.

❑ **Local variables #1 – #33**

Local variables are variables used locally in the macro. Accordingly, in case of multiples calls (calling macro B from macro A), the local variable used in macro A is never destroyed by being used in macro B.

❑ **Common variables #100 – #199, #500 – #999**

Compared with local variables used locally in a macro, common variables are common throughout the main program, each sub program called from the main program, and each macro. The common variable #1 used in a certain macro is the same as the common variable #i used in other macros. Therefore, a common variable #1 calculated in a macro can be used in any other macros.

Common variables #100 to #199 are cleared when power is turned off, but common variables #500 to #999 are not cleared after power is turned off.

❑ **System variables**

A variable with a certain variable number has a certain value. If the variable number is changed, the certain value is also changed.

The certain value are the following:

◇ 16 points DI (for read only)

◇ 48 points DO (for output only)

◇ Tool offset amount, workpiece origin offset amount

◇ Position information (actual position, skip position, block end position, etc.)

◇ Modal information (F code, G code for each group, etc.)

◇ Alarm message (Set alarm number and alarm message, and the CNC is set in an alarm status. The alarm number and message is displayed.)

◇ A date (year, month, day) and time (hour, minute, second) are indicated.

◇ Clock (Time can be known. A time can also be preset.)

◇ Single block stop, Miscellaneous function end wait hold

◇ Feed hold, Feed rate override, Exact stop inhibition

◇ The number of machining parts is indicated. It can be preset.

- **External output commands**

Value of variables or characters can be output to external devices via the reader/puncher interface with custom macro command. Results in measurement is output using custom macro.

- **Limitations**

❑ **Usable variables**
See “Types of variables”.

❑ **Usable variable values**
Maximum : $\pm 10^{47}$
Minimum : $\pm 10^{-29}$

❑ **Constants usable in <expression>**
Maximum : ±99999999
Minimum : ±0.0000001
Decimal point allowed

❑ **Arithmetic precision**
8-digit decimal number

❑ **Macro call nesting**
Maximum 4 folds.

❑ **Repeated ID numbers**
1 - 3

❑ **( ) nesting**
Maximum 5 folds.

❑ **Sub program call nesting**
8 folds (including macro call nesting)

18.2 ADDITION OF CUSTOM MACRO COMMON VARIABLES

The range of common variables can be used to #100 to #199, and #500 to #999.

18.3 INTERRUPTION TYPE CUSTOM MACRO

When custom macro interruption signal is input during automatic operation, the block currently under execution is interrupted and the specified custom macro is activated. After execution of this custom macro, it returns to the interrupted block and continues execution of the remaining commands.

```
  :
M96P_ ;
  :  When custom macro interruption signal is input between M96 block
     and M97 block, custom macro specified by P is activated.
M97;
  :
```

This function enables implementation of an application of detecting a broken tool, entering a custom macro interrupt signal, executing a tool change cycle using the custom macro, and then restarting machining after the tool change.

18.4 PATTERN DATA INPUT

With this function, custom macro interruption signal can be input on detection of tool break, tool change cycle can be executed by custom macro, and machining is continued.

This function simplifies program creation for CNC machining. Instead of programming in the NC format, the program can be created by selecting a menu and entering data according to the menu displayed on the CRT screen. A menu is provided for each type of drilling such as boring and tapping. A programmer can select data necessary for actual machining from these menus. Machining data such as hole position and hole depth is also provided in menus. The programmer can create a program simply by entering data from the menus.

This function is basically executed by the custom macro created by a machine tool builder. What menus and machining data to prepare totally depends on a machine tool builder. Therefore, a machine tool builder can incorporate their own know-how into this function.

```
MENU:HOLE PATTERN                          O9505 N0001

   1. TAPPING
   2. DRILLING
   3. BORING
   4. POCKET
   5. BOLT HOLE
   6. LINEANGLE
   7. GRID
   8. PECK
   9.
  10.

>_                                         S    0 T0000
EDIT **** *** ***       15:56:32
[ MACRO ][  MENU  ][  OPR  ][        ][(OPRT) ]
```

Pattern menu display

5 is selected

```
VAR. : BOLT HOLE                           O9505 N0001
 NO.   NAME           DATA             COMMENT
  500 TOOL            0.000
  501 KIJUN X         0.000          *BOLT HOLE
  502 KIJUN Y         0.000               CIRCLE*
  503 RADIUS          0.000          SET PATTERN
  504 S. ANGL         0.000          DATA TO VAR.
  505 HOLES NO.       0.000          NO.500-505.
  506                 0.000
  507                 0.000
ACTUAL POSITION (RELATIVE)
      X     0.000       Y     0.000
      Z     0.000
>_                                         S    0 T0000
 MDI **** *** ***       15:56:32
[OFFSET ][SETING ][       ][        ][(OPRT) ]
```

Pattern data display

18.5 MACRO EXECUTOR FUNCTION

There are two types of NC programs; those which, once created, are scarcely changed, and those which are changed for each machining type. The former are programs created by the custom macro, and the latter are machining programs. If programs of these types are executed simultaneously, a battery may run out or the custom macro may be destroyed by error operation.

Such problems can be solved by this function. The custom macro created by a machine tool builder is converted to an execute-form program, be cataloged in the Flash ROM module, and be executed.

Features

- ❑ Since the program is cataloged after converted to an execute-form program, the execution speed is high. The machining time is then reduced, and the precision is improved.
- ❑ Since the program is cataloged in Flash ROM, there is no problem of battery extinction or custom macro destruction by error operation. The reliability is improved.
- ❑ Since the cataloged program is not displayed on a program screen, the know-how of the machine tool builder is protected.
- ❑ Since the custom macro is cataloged in Flash ROM, the program edit memory can be used efficiently.
- ❑ The user can call the macro easily without knowing the cataloged program. A custom macro can be created and executed in the program edit memory as usual.
- ❑ An original screen can be created by using the graphic display or selecting screens by the soft key. The machine tool builder can extend the control function by using such functions as machining program creation and edit control, reader/punch interface control, and PMC data read/write functions.

NOTE

To use the macro executor function for graphics display, the graphics function is required.

19 SERIES 10/11 TAPE FORMAT

19.1 SERIES–10/11 TAPE FORMAT

Memory operation of a program created for the following function in the Series 10/11 tape format can be performed based on the setting parameter.

- Equal–lead threading (G33) (T series) ... (G32 with G code system A)
- Subprogram call (M98)
- Canned cycle (G77, G78, G79) (T series) ... (G90, G92, and G94 with G code system A)
- Multiple repetitive canned cycle (G71 to G76) (T series)
- Canned cycle for drilling (G80 to G85) (T series)
- Canned cycle (G73, G74, G76, G80 to G89) (M series)

NOTE

Address and value specification range

The restrictions imposed on the Series 0*i* format are also imposed on the value specification range for the basic addresses. When a specified value exceeds the range of the Series 0*i* format, a P/S alarm is issued. The use of addresses may be restricted in some cases.

20 FUNCTIONS FOR HIGH SPEED CUTTING

20.1 ADVANCED PREVIEW CONTROL (G08)

T series

This function is designed for high–speed precise machining. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase as the feedrate becomes higher can be suppressed.
The tool can then follow specified values accurately and errors in the machining profile can be reduced.
This function becomes effective when advanced preview control mode is entered.

Format

G08 P_ ;

P1 : Turn on advanced preview control mode.
P0 : Turn off advanced preview control mode.

In advanced preview control mode, the following functions are available :

- Linear acceleration/deceleration before interpolation
- Automatic corner deceleration function

20.2 AI ADVANCED PREVIEW CONTROL (G05.1)

M series

By taking full advantage of high–precision contour control using a RISC processor, this function enables high–speed high–precision machining without the need for special hardware.

The function enables look–ahead linear acceleration/deceleration before interpolation of up to 15 blocks. This results in smooth acceleration/deceleration over many blocks, as well as high–speed machining.

Format

G05.1 Q_ ;

Q1 : Start AI advanced preview control mode
Q0 : End AI advanced preview control mode
G05.1 A block for specifying G05.1 must not contain any other command.
AI advanced preview control mode can also be canceled by a reset.

20.3 AI CONTOUR CONTROL (G05.1)

M series

By taking full advantage of high–precision contour control using a RISC processor, this function enables high–speed high–precision machining without the need for special hardware.

The function enables look–ahead linear acceleration/deceleration before interpolation of up to 15 blocks. This results in smooth acceleration/deceleration over many blocks, as well as high–speed machining.

Format

G05.1 Q_ ;

Q1 : Start AI contour control mode
Q0 : End AI contour control mode
G05.1 A block for specifying G05.1 must not contain any other command.
AI contour control mode can also be canceled by a reset.

21 AXIS CONTROL

21.1 FOLLOW UP FUNCTION

Normally, the machine is controlled to move to a commanded position. However, when the follow up function is applied, actual position in the CNC is revised according to the move of the machine.
Follow up function is activated when:
- Emergency stop is on

Because machine movement during the emergency stop is reported, the actual position of the machine is reflected in the CNC. Therefore, machining can be resumed after the emergency stop has been deactivated, without performing the reference point return again.
However, when a trouble has generated in the position detection system, the system cannot follow up correctly. So present position in CNC does not become correct value.
By input signal (follow up signal) from PMC follow up function can also be applied to:
- Servo off

status. It is also valid in cases when the machine is moved with a mechanical handle.

21.2 MECHANICAL HANDLE FEED

It is possible to move the machine by hand, using the mechanical handle installed on the machine; not by the NC (servo motor).
Move distance by the mechanical handle is followed up and actual position in The NC is revised. The mechanical handle feed is done by inputting the servo off signal of the axis fed. It is necessary, however, to specify following up of the movement in the servo off status with the follow up signal.

21.3 SERVO OFF

Servo on/off control per axis is possible by input signals from PMC. This function is generally used with the machine clamp.

21.4 MIRROR IMAGE

The MDI-commanded or the program-commanded move direction of each axis can be reversed and executed.
Mirror image is set by MDI setting or by input signals from PMC.
Mirror image can be applied to each axis.

21.5 CONTROL AXIS DETACH

It is possible to detach or attach rotary tables and attachments with this function. Switch control axis detach signal according to whether the rotary tables and attachments are attached or detached. When this signal is on, the corresponding axis is excluded from the control axes, so the servo alarm applied to the axis are ignored. The axis is automatically regarded as being interlocked. This signal is not only accepted when power turned is on, so automatic change of attachments is possible any time with this function.
The same switching as with this signal can also be performed with the MDI setting.

21.6 SIMPLE SYNCHRONOUS CONTROL

An input signal from PMC can be used to select whether simple synchronization control is performed. During simple synchronization control, the move command for the master axis is issued to the two motors of the master and slave axes for synchronization control of the two axes. However, there are no functions for performing synchronization compensation so that the positional deviations of the master and slave axes are equalized by constantly monitoring these positional deviations. Even during simple synchronization control, pitch error compensation and backlash compensation for the master axis are performed separately from those for the slave axis.
The following shows the differences in simple synchronization control between the M series and the T series.

For M series

- Operation modes that allow simple synchronization control

  Automatic operation, Jog feed, Manual handle feed,
  Incremental feed, and Manual reference position return
- Number of synchronized pairs

  Up to four (depending on the number of control axes)
- Other functions

  Synchronization error check (positional deviation/machine coordinates)
  Synchronization (During power–up of the CNC, the departure generated during power–down of the CNC is automatically compensated. However, the absolute–position detectors are required for the master and slave axes.)
  Automatic grid positioning
  Torque difference alarm

For T series

- Operation modes that allow simple synchronization control

  Automatic operation only (disabled during manual operation)
- Number of synchronized pairs

  One
- Other functions

  None

21.7 NORMAL DIRECTION CONTROL (G40.1,G41.1,G42.1)

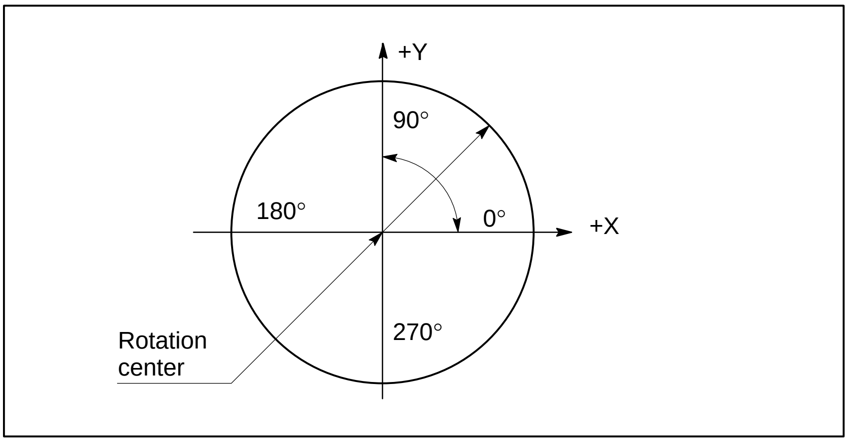
M series

The rotation axis (C axis) can be controlled by commanding the G41.1 or G42.1 so that the tool constantly faces the direction perpendicular to the advancing direction during cutting.

Format

G40.1	:	Normal direction control cancellation mode (No normal direction control can be performed.)
G41.1	:	Normal direction control left side on (Control is made to allow facing perpendicular to advancing direction to the left)
G42.1	:	Normal direction control right side on (Control is made to allow facing perpendicular to advancing direction to the right)

In the normal direction control, control is made so that the tool may be perpendicular to the advancing direction on the X-Y plane.
With the angle of C axis, the +X direction is defined to be 0 degrees viewed from the rotation center of C axis. Then, the +Y direction, –X direction, and –Y direction are defined to be 90, 180, and 270 degrees, respectively.
When shifting to the normal direction control mode from the cancellation mode, the C axis becomes perpendicular to the advancing direction where the G41.1 or G42.1 is at the starting point of commanded block.



Between blocks, the traveling of C axis is automatically inserted so that the C axis faces the normal direction at the starting point of each block according to the change of traveling direction.
Normal direction control is performed for the path after compensation during the cutter compensation mode. The feed rate of rotation of C axis inserted at the starting point of each block becomes the federate set by parameters. However, when dry run is valid, the feed rate is set to the dry run rate. Also, in the case of rapid traverse (G00), it becomes the rapid traverse rate. In the case of circular command, the C axis is allowed to be rotated first so that the C axis faces perpendicular to the circular starting point. At this time, the C axis is controlled so that it constantly faces the normal direction along with the move of circular command.

NOTE

The rotation of C axis during normal direction control is controlled at short distance so that 180 degrees or less may result.

21.8 POLYGON TURNING (G50.2, G51.2)

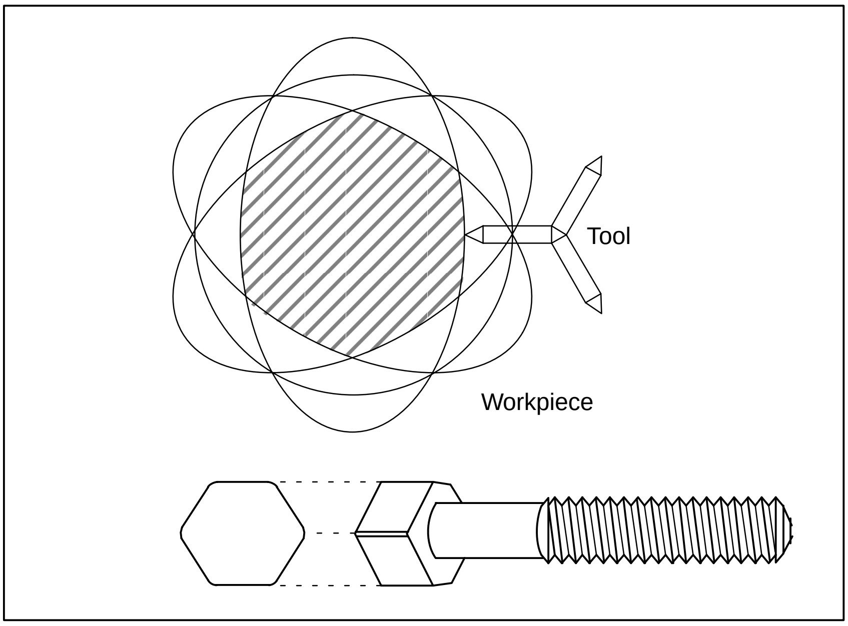
T series

A polygonal figure can be machined by turning the workpiece and tool at a certain ratio.

- Rotation ratio of the workpiece and tool
- Number of tool teeth

The polygon can be a quadrilateral or hexagon according to the above machining conditions.
Compared with the machining performed by using C and X axes with polar-coordinate compensation, this machining requires shorter time. It, however, cannot form a precise figure of a polygon. Generally, this method is used for machining of square or hexagonal bolt heads or hexagonal nuts.

Example)
Rotation ratio of the workpiece and tool : 1 : 2
Number of teeth : Three at every 120° (for a hexagon)



The rotation of the tool for polygon turning is controlled by the CNC control axis. Hereafter, the rotation axis of this tool is called B axis.
Command G51.2 controls the B axis so that the ratio of the tool speed to the speed of the workpiece (specified by the S command beforehand) attached to the spindle becomes the specified value.
The synchronization between the spindle and B axis is canceled by the command below.
G50.2 ;

Format

G51.2 P_ Q_ ;

P and Q :	Rotation ratio of spindle to B axis
	Command range : Integer value of 1 to 9 for both P and Q
	When the value of Q is positive, the rotation direction of B axis is in positive direction.
	When the value of Q is negative, the rotation direction of B axis is in negative direction.
(Example)	When the rotation ratio of spindle to B axis is equal to 1 : 2 and the rotation direction of B axis is positive direction G51.2 P1 Q2 ;
G50.2 ;	Cancel

When synchronous start is commanded by the G51.2, one rotation signal from the position coder mounted in the spindle is detected and the rotation of B axis is controlled while being synchronous with the rate of spindle in response to the rotation ratio (P : Q). Namely, control is made so that the ratio of spindle to B axis is P : Q. This relationship continues until the synchronous cancellation command (G50.2 or reset) are carried out. The direction of rotation of B axis is determined by the symbol Q and is not affected by the direction of rotation of position coder.
When the G50.2 is commanded, the synchronization of the spindle and B axis is canceled and the B axis is stopped.

21.9 AXIS CONTROL WITH PMC

The PMC can directly control any given axis, independently of the CNC. In other words, moving the tool along axes that are not controlled by the CNC is possible by entering commands, such as those specifying movimg distance and feedrate, from the PMC. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

Whether the CNC or PMC controls an axis is determined by the input signal provided for that particular axis.

The PMC can directly control the following operations :

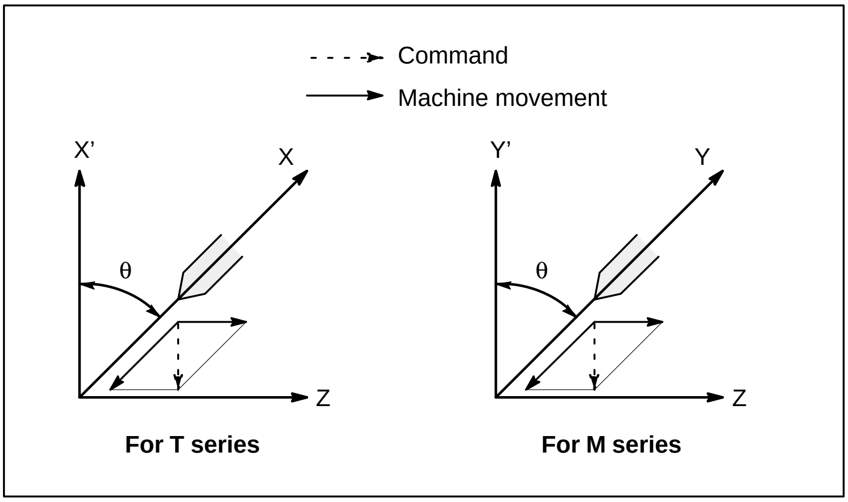
(1) Rapid traverse with moving distance specified
(2) Cutting feed–feed per minute, with moving distance specified
(3) Cutting feed–feed per revolution, with moving distance specified
(4) Skip–feed per minute, with moving distance specified
(5) Dwell
(6) Continuons feed
(7) Reference position return
(8) 1st reference position return
(9) 2nd reference position return
(10) 3rd reference position return
(11) 4th reference position return
(12) External pulse synchronization–Main spindle
(13) External pulse synchronization–first manual handle
(14) External pulse synchronization–second manual handle
(15) External pulse synchronization–third manual handle (for M series only)
(16) Feedrate control
(17) Auxiliary function, Auxliary function 2, Auxliary function 3
(18) Selection of the machine coordinate system
(19) Torque control command

The PMC is provided with four paths to control these operations using input and output signals.

By issuing commands through these four paths, the PMC can simultaneously control multiple axes separately. Use parameter to determine which path controls which axis. Commands may be issued through one path to two or more axes, thus allowing the PMC to control multiple axes using one path.

21.10 ANGULAR AXIS CONTROL

For T series, even if the X axis is not vertical to the Z axis (for T series, the Y axis not vertical to the Z axis), they are assumed to form a orthogonal coordinate system, simplifying programming. The movement of each axis is automatically controlled according to the slant angle.

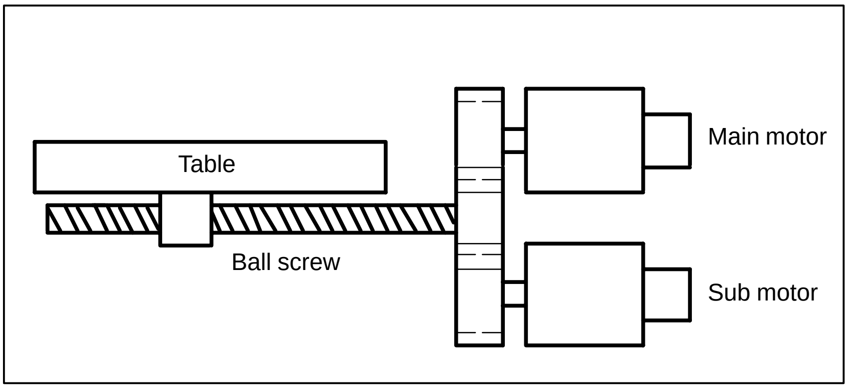


21.11 ARBITRARY ANGULAR AXIS CONTROL

With arbitrary angular axis control, however, any axes can be specified as the angular and perpendicular axes, by specifying parameters accordingly.

21.12 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The sub motor is used only to produce torque. With this tandem control function, the torque produced can be doubled.



Example of operation

In general, the CNC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.

21.13 TANDEM DISTURBANCE ELIMINATION CONTROL

This function can be used to suppress vibrations caused by interference between the main axis and the sub axis in position tandem (quick synchronization) control.

NOTE

1 The function can only be used for two–axis quick synchronization control. It cannot be used for multiaxis quick synchronization control.
2 In a servo axis arrangement, the main axis must be placed on an odd–numbered axis and the sub axis must be placed on an even–numbered axis adjacent to the main axis.
3 The function cannot be used for a mechanism in which a mechanical connection between two axes can be released.

21.14 TEMPORARY ABSOLUTE COORDINATE SETTING

In a closed loop system equipped with a built–in absolute position detector (serial pulse coder) and incremental linear scale, a coordinate system is established at power–up by using absolute position data from the built–in absolute position detector. The subsequent position control is performed using incremental data of the linear scale. Since the position immediately after power–up is a temporary position, manual reference position return is needed to obtain a correct position.
When this function is used, a stroke limit check is enabled even before reference position return, although the position at power–up is an approximate position. Note that this function does not use the incremental linear scale as an absolute position detector.
This function is optional.

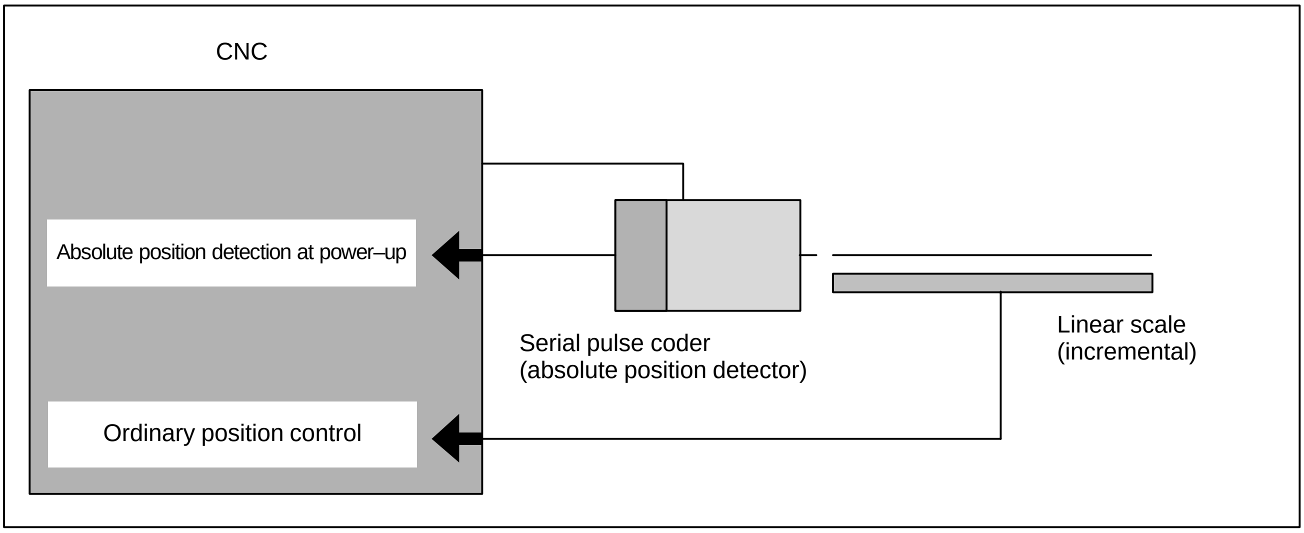


Fig. 21.14 System using the temporary coordinate setting function

22 MANUAL OPERATION

22.1 JOG FEED

- **Jog feed**
  Each axis can be moved in the + or - direction for the time the button is pressed. Feed rate is the parameter set speed with override of:
  0 - 655.34%, 0.01% step.
  The parameter set speed can be set to each axis.
- **Manual rapid feed**
  Each axis can be fed in a rapid feed to the + or - direction for the time the button is pressed.
  Rapid traverse override is also possible.

22.2 INCREMENTAL FEED

Specified move amount can be positioned to the + or - direction with the button.
Move amount of:
(least command increment) x (magnification)
can be specified. The feed rate is that of manual feed.
The possible magnifications to be specified are as follows.
×1, ×10, ×100, ×1000.

Increment system	Metric input	Inch input
IS–B	0.001, 0.01, 0.1, 1.0	0.0001, 0.001, 0.01, 0.1
IS–C	0.0001, 0.001, 0.01, 0.1	0.00001, 0.0001, 0.001, 0.01

22.3 MANUAL HANDLE FEED (1ST)

By rotating the manual pulse generator, the axis can be moved for the equivalent distance. Manual handle feed is controlled 1 axis at a time. The manual pulse generator generates 100 pulses per rotation. Move amount per pulse can be specified from the following magnifications:
×1, ×10, ×M, ×N.
N is parameter set values of 0 - 1000. M is parameter set values of 1–127.
Move distance is :
(Least command increment) x (magnification)

Increment system	Metric input	Inch input
IS–B	0.001, 0.01, M/1000, N/1000 mm	0.0001, 0.001, M/10000, N/10000 inch
IS–C	0.0001, 0.001, M/10000, N/1000 mm	0.00001, 0.0001, M/100000, N/100000 inch

22.4 MANUAL HANDLE FEED (2ND, 3RD) (T SERIES: 2ND)

A 2nd, as well as 3rd manual pulse generator can be rotated to move the axis for the equivalent distance. Manual handle feed of 3 axes (for T series, 2 axes) can be done at a time. Multiplier is common to 1st, 2nd and 3rd manual pulse generators.

22.5 JOG AND HANDLE SIMULTANEOUS MODE

Although manual handle feed is usually enabled only in the manual handle-feed mode, it can also be performed in the manual continuous-feed mode by setting the corresponding parameters. However, manual continuous-feed and manual handle-feed cannot be performed simultaneously. Manual handle-feed can be performed only when manual continuous-feed is in progress (i.e., an axis is moving).

22.6 MANUAL PER–REVOLUTION FEED

T series

The feedrates in jog feed and incremental feed can be specified by inputting either feed distance per minute or feed distance per rotation.

1 Specification of feed distance per minute or feed distance per rotation is selected by setting the corresponding parameter.
2 During manual rapid traverse, feed distance per minute is always specified.

22.7 MANUAL ABSOLUTE ON/OFF

When tool is moved by manual operation, whether to add the move distance to the absolute coordinate value in the workpiece coordinate system is selected depending on the input signal *ABSM.
When tool is moved by manual operation when *ABSM is set to 0, the move distance is added to the absolute coordinate value.
When tool is moved by manual operation when *ABSM is set to 1, the move distance is ignored, and is not added to the absolute coordinate value. In this case, the work coordinates is shifted for the amount tool was moved by manual operation.

23 AUTOMATIC OPERATION

23.1 OPERATION MODE

23.1.1 DNC Operation

The part program can be read and executed block by block from the input device connected to the reader/puncher interface.

23.1.2 Memory Operation

Program registered in the memory can be executed.

23.1.3 MDI Operation

Multiple blocks can be input and executed by the MDI unit.

23.2 SELECTION OF EXECUTION PROGRAMS

23.2.1 Program Number Search

Program number currently in need can be searched from the programs registered in memory operating the MDI.

23.2.2 Sequence Number Search

The sequence number of the program on the currently selected memory can be searched using the MDI unit.
When executing the program from half-way (not from the head) of the program, specify the sequence number of the half-way program, and the program can be executed from the half-way block by sequence number search.

23.2.3 Rewind

After program execution has ended, the program in the memory or the tape reader can be reminded to the program head, with this reset & rewind signal on. (When a portable tape reader with reels is in use)

23.2.4 External Workpiece Number Search

By specifying workpiece numbers of 01 - 15 externally (from the machine side, etc.), program corresponding to the workpiece number can be selected.
The workpiece number equals the program number. For example when workpiece number 12 is specified, program, O0012 is selected.

23.2.5 Expanded External Workpiece Number Search

The external workpiece number search function has been expanded. A workpiece number 0001 to 9999 can be specified to select a program (O0001 to O9999) corresponding to the workpiece number.

23.3 ACTIVATION OF AUTOMATIC OPERATION

23.3.1 Cycle Start

Set operation mode to memory operation, MDI operation, or DNC operation, press the cycle start button, and automatic operation starts.

23.4 EXECUTION OF AUTOMATIC OPERATION

23.4.1 Buffer Register

Buffer register in CNC equivalent to one block is available for program read and control of CNC command operation intervals caused by preprocess time.

23.5 AUTOMATIC OPERATION STOP

23.5.1 Program Stop (M00, M01)

Automatic operation is stopped after executing the M00 (program stop) commanded block. When the optional stop switch on the operator's panel is turned on, the M01 (optional stop) commanded block is executed and the automatic operation stops.
The automatic operation can be restarted by the cycle start button.

23.5.2 Program End (M02, M30)

The CNC is reset after executing the M02 (end of program) or M30 (end of tape) commanded block.

23.5.3 Sequence Number Comparison and Stop

During program operation, when the block with a preset sequence number appears, operation stops after execution of the block, to a single block stop status. The sequence number can be set by the operator through the MDI panel. This function is useful for program check, etc., because program can be stopped at optional block without changing the program.

23.5.4 Feed Hold

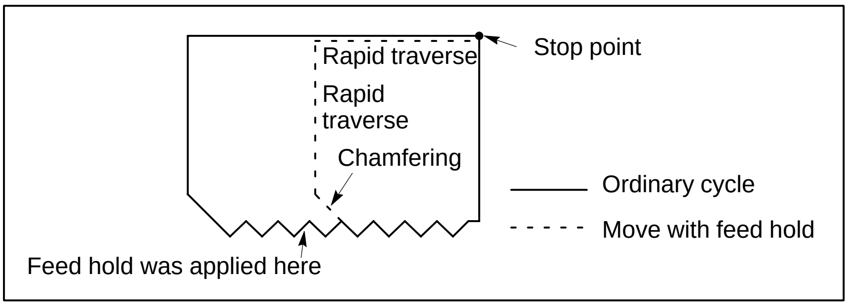
The CNC can be brought to an automatic operation hold status by pressing the feed hold button on the operator's panel. When feed hold is commanded during motion, it decelerates to a stop.
Automatic operation can be restarted by the cycle start button.

23.5.5 Thread Cutting Cycle Retract

T series

When feed hold is commanded during thread cutting cycle by G76 or G78, the tool rapidly relieves to the cycle start point, like in the final chamfering of the thread cutting cycle. Thread cutting cycle restarts by cycle start command.

Without this function, if feed hold is commanded during thread cutting, it returns and stops at the position where thread cutting circle was started after thread cutting is ended.



23.5.6 Reset

The automatic operation can be ended in a reset status by the reset button on the MDI panel or by the external reset signal, etc. When reset is commanded during motion, it decelerates to a stop.

23.6 RESTART OF AUTOMATIC OPERATION

23.6.1 Program Restart

This function allows program restart by specifying the desired sequence number, for example after tool break and change, or when machining is restarted after holidays. The NC memorizes the modal status from the beginning of the program to the sequence number.
If there are M codes necessary to be output, output the M code by the MDI, press the start button, the tool automatically moves to the start position, and the program execution restarts.

23.6.2 Manual Intervention and Return

In cases such as when tool movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the tool: When automatic operation is restarted, this function returns the tool to the position where manual intervention was started.
To use the conventional program restart function and tool withdrawal and return function, the switches on the operator's panel must be used in conjunction with the MDI keys. This function does not require such operations.

23.7 MANUAL INTERRUPTION DURING AUTOMATIC OPERATION

23.7.1 Handle Interruption

During automatic operation, tool can be adjusted by the manual pulse generator without changing the mode. The pulse from the manual pulse generator is added to the automatic operation command and the tool is moved for the recommended pulses.
The work coordinate system thereafter is shifted for the pulse commanded value. Movement commanded by handle interruption can be displayed.

23.8 SCHEDULING FUNCTION

Any of the files (programs) stored on a FANUC Handy File, a FANUC Program File Mate, a FANUC FLOPPY CASSETTE can be selected and executed.

- A list of the files stored on the Floppy Cassette can be displayed.
- Files can be executed in an arbitrary order and executed an arbitrary number of times by specifying file numbers in a desired order along with their repeat counts.

File list screen

```
FILE DIRECTORY                          F0004 N00020
 CURRENT SELECTED:O0002
   NO.  FILE NAME             (METER)  VOL
  0000  SCHEDULE
  0001  PARAMETER                 46.1
  0002  ALL.PROGRAM               12.3
  0003  O0001                      1.9
  0004  O0002                      1.9
  0005  O0003                      1.9
  0006  O0004                      1.9
  0007  O0005                      1.9
  0008  O0010                      1.9


 RMT **** *** ***          09:36:48
[ SELECT ][      ][       ][       ][       ]
```

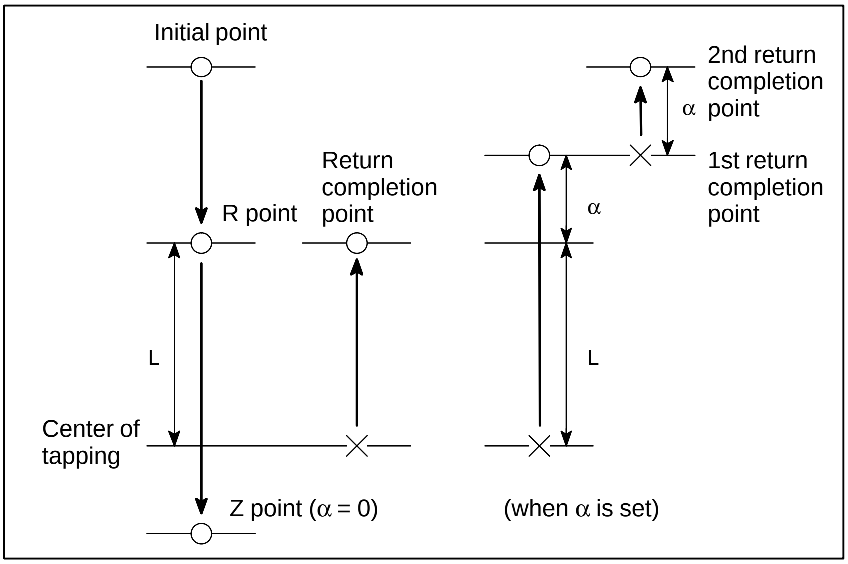
Schedule screen (for specifying file numbers and repeat counts)

```
FILE DIRECTORY                          F0000 N00020
 ORDER    FILE NO.      REQ.REP  CUR.REP
   01       0001            2        0
   02       0007           25        0
   03       0008            6        0
   04       0011         9999        0
   05       0012         LOOP        0
   06
   07
   08
   09
   10

>_
 RMT **** *** ***          09:36:48
[ PRGRM ][     ][  DIR  ][ SCHDUL ][ (OPRT) ]
```

23.9 RETRACTION FOR RIGID TAPPING

M series

When rigid tapping is stopped, either by an emergency stop or by a reset, the tap may cut into the workpiece. The tap can subsequently be drawn out by using a PMC signal. This function automatically stores information relating to the tapping executed most recently. When a tap return signal is input, only the rigid tapping cycle return operation is executed, based on the stored information. The tap is pulled toward the R point. When a return value α is set in a corresponding parameter, the pulling distance can be increased by α.



23.9.1 Retraction for Rigid Tapping by Specifying G30

Instead of signal input, a G30 command can be used to start rigid tapping return. This is made possible by parameter setting.

Format

G30 P99 M29 S_ ;

M29 : A parameter–set M code that specifies rigid tapping
S_ : Specify S used when rigid tapping is specified. (Optional)

NOTE

1 When use of G30 is selected, rigid tapping return cannot be performed by signal input.
2 The rigid tapping return command is a one–shot command.

24 PROGRAM TEST FUNCTIONS

24.1 ALL-AXES MACHINE LOCK

In machine lock condition, the machine does not move, but the position display is updated as if the machine were moving. Machine lock is valid even in the middle of a block.

24.2 MACHINE LOCK ON EACH AXIS

Machine lock can be commanded per axis.

24.3 AUXILIARY FUNCTION LOCK

This function inhibits transmitting of M, S, T, B function code signals and strobe signals to PMC. Miscellaneous functions M00, M01, M02, and M30 are executed even when miscellaneous function lock is applied, allowing the code signal, strobe signal, and decode signal to be transmitted normally.

24.4 DRY RUN

In the dry run mode, the tool moves at the speed obtained by multiplying the dry run speed by the override value for manual feeding, regardless of the specified cutting federate. The dry run speed is specified in the corresponding parameter. However, the rapid traverse command (G00) and rapid traverse override value are effective.
Dry run can also be commanded to rapid feed command (G00) by parameter setting.

24.5 SINGLE BLOCK

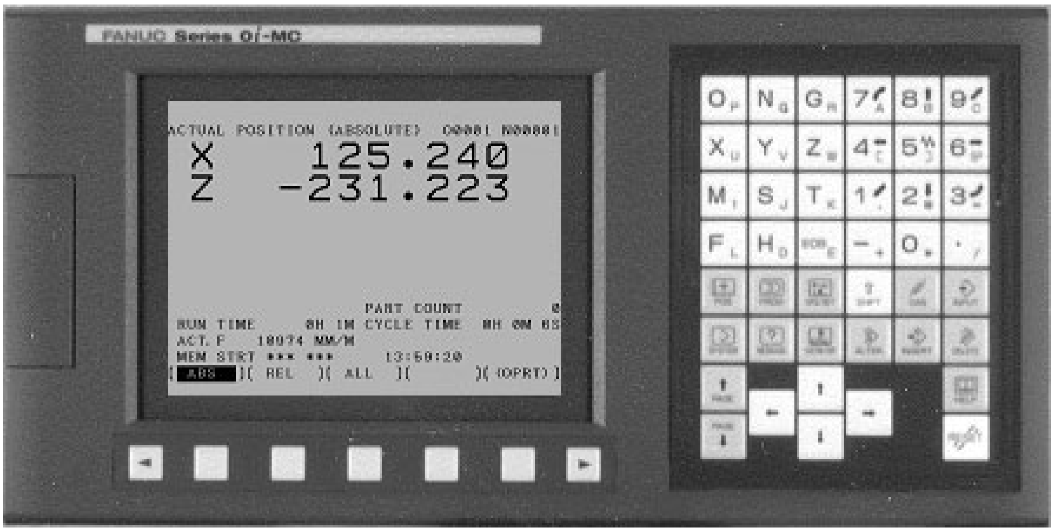
The program can be executed block by block under automatic operation.

25 SETTING AND DISPLAY UNIT

25.1 SETTING AND DISPLAY UNIT

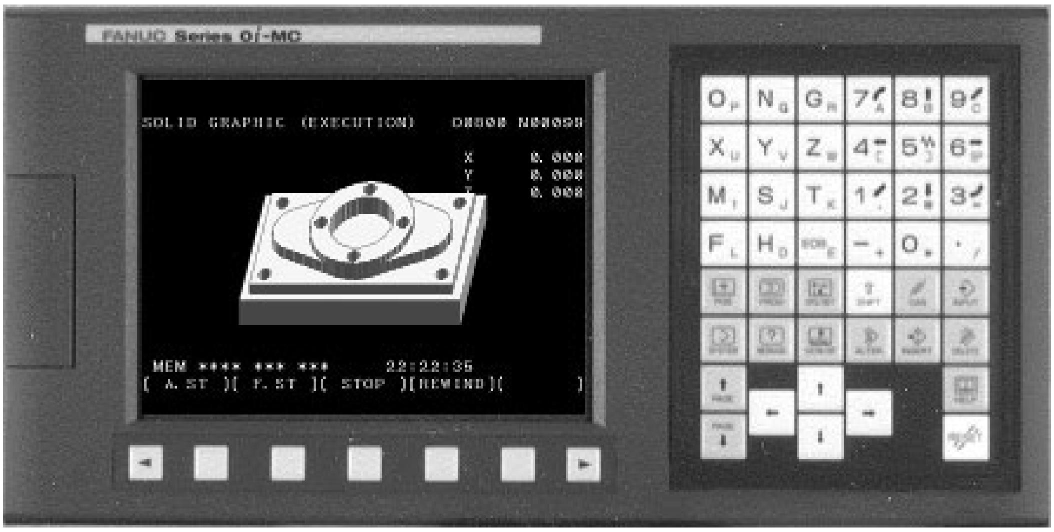
The setting and display units are shown in Subsections II–25.1.1 to II–25.1.4.

25.1.1 7.2″ Monochrome LCD/MDI Unit (Horizontal Type)



NOTE
The MDI varies between the T series and the M series.

25.1.2 8.4″ Color LCD/MDI Unit (Horizontal Type)

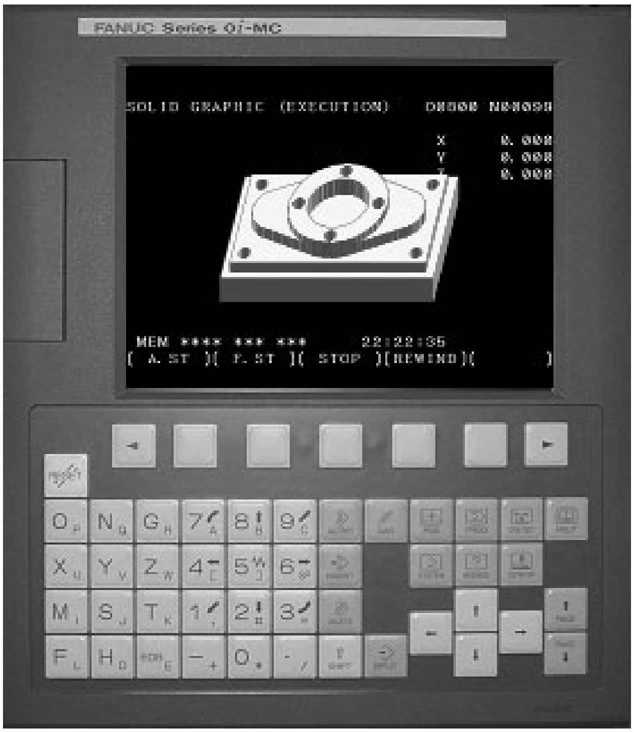


NOTE
The MDI varies between the T series and the M series.

25.1.3 7.2″ Monochrome LCD/MDI Unit (Vertical Type)

NOTE
The MDI varies between the T series and the M series.

25.1.4
8.4″ Color LCD/MDI Unit (Vertical Type)



NOTE
The MDI varies between the T series and the M series.

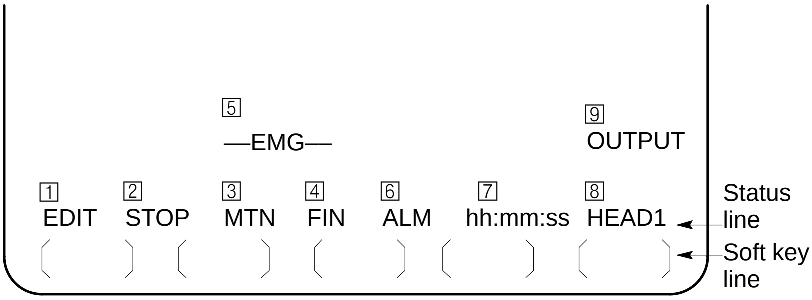
26 DISPLAYING AND SETTING DATA

26.1 DISPLAY

The following data are displayed.

- **Indication of statuses and tool post names**

The status of the control unit is indicated on the screen. Statuses include the state when an alarm is being activated or when the system is in the edit mode. The status line is displayed right above the soft key line.



1 Operation mode (MDI, MEM, RMT, EDIT, HND, TJOG, THND, INC, or REF)

2 Status of automatic operation (STOP, HOLD, STRT, or ****)
***** : Reset
STOP : Automatic operation is in a stopped state.
HOLD : Automatic operation is in a halt state.
STRT : Automatic operation has been started.

3 Axis movement/dwell (MTN, DWL, or ***)

4 FIN wait state (FIN or ***)

5 Emergency stop (--EMG--) (displayed above in 3 and 4)

6 Alarm status (ALM, WNG, or ***)

7 Clock (hh:mm:ss)

8 Name of the path currently selected (only at 2–path control)

> **NOTE**
> The name of a path can be specified by the corresponding parameter with a string of up to seven characters. The characters may be numbers, letters, katakana characters, or symbols.

9 Status display such as program editing (INPUT, OUTPUT, SRCH, EDIT, LSK, or RSTR)
(8 and 9 are displayed in the same column. When a program is being edited, 9 is displayed.)

- **Key input display**

Data input via the address keys or the numerical keys are displayed at the left lower part of the screen.

- **Program number, sequence number display**

Program number, sequence number is displayed on the right upper part of the screen.

- **Alarm display**

Alarm number and its contents are displayed briefly.

- **Alarm message display**

Alarm message contents are displayed.

- **Present position display**

Relative position and position in the work coordinates are displayed in 3-times magnified characters.

- **Total position display**

Relative position, position in the work coordinates, position in the machine coordinate, and remaining move distance are displayed in one screen.

- **Command value display**

The following two displays are performed.

❑ Previously commanded modal value and command value to be executed (ACTIVE)

❑ Command value of the next block

- **Setting (parameter set by the operator) display**

Displays setting value.

- **Tool offset amount display**

Displays offset value. Relative position is also displayed at the same time.

- **Program display**

❑ Display of program for editing.
❑ Display of program currently under execution.
❑ Display of program list.
A list of program number and program name, of programs stored in the memory is displayed.
Used memory size and remaining memory size are also displayed.

- **Parameter display**
- **Self diagnosis result display**
- **Custom macro variables display**
- **External operator message, external alarm message display**

- **Actual speed and actual spindle speed**

❑ Actual feedrate per minute (mm/min or inch/min)
Movement along an arbitrary axis can also be excluded from the actual feedrate indications by parameter setting.
❑ Actual spindle speed (min^{-1})

- **Program check screen**

The following are displayed on one screen.
❑ Program number on execution
❑ Sequence number on execution
❑ Program text on execution
❑ Current position
❑ Modal G codes
❑ Modal M codes
❑ T code
❑ Actual feedrate and spindle speed
❑ Status

- **Operating monitor display**

The load values (torque values) of spindle motor and servo motor are displayed in bar chart.
The most recent sampling values are displayed in bar chart display. Set the rated load value of motor corresponding to each load meter to parameters. The load meter displays 100% when the load value is the rated load value.
The load meter can be displayed up to three servo motor axes and a parameter can be used to select any one of three axes.

- **Displaying the alarm history**

A maximum of 25 of the most recent alarms generated in CNC can be recorded. Each alarm record consists of the following items:
- ❑ Date and time
- ❑ Alarm number
- ❑ Alarm message

Any of the records can be deleted from the alarm history.
In addition, the operator message history can be displayed.

```
ALARM HISTORY                               O0100 N00001

 02. 02. 14  16:43:48
    2 010  3IMPROPER G-CODE
 02. 02. 13   8:22:21
     506   OVER TRAVEL    :+1
 02. 02. 12  20:15:43
     417   SERVO ALARM    :X AXIS DGTL PARAM






 MEM **** *** ***         09:36:48
[ ALARM ][  MSG  ][ HISTORY ][     ][ (OPRT) ]
```

- **Displaying external operator message history**

The history of external operator messages can be stored. The stored history can be displayed on the external operator message history screen.

26.2 LANGUAGE SELECTION

The Japanese, English, German, French, Italian, Spanish, Chinese, Korean, Portuguese, Hungarian, Polish, Swedish, and Dutch (M series only) are prepared as display languages. Select the language to be displayed by parameters.

26.3 CLOCK FUNCTION

Time is displayed in the hour/minute/second format on each display screen. Some screens allows display of the year, month, and day.
The custom macro system variable can be used to read the time. The time will be told through the window at PMC side.

26.4 RUN TIME & PARTS NUMBER DISPLAY

This function displays the integrated power-on time, the integrated cycle operation time, the integrated cutting time and timer on the cRT display screen. The integrated cycle operation time, the integrated cutting time and timer can be altered and preset, using the MDI.
In addition to the above, this function displays the count of the total number of parts machined, the number of parts required and the number of parts on the screen. Each time M02, M30 or a parameter set M code is executed, the count of the total in memory is incremented by 1.
If a program is prepared so as to execute M02, M30 or a parameter set M code each time one part machining is completed, the number of parts machined can be counted automatically.
If the count of the number of parts reaches the number of parts required, a signal is output to the PMC side.
It is possible to change and preset the number of parts required and the number of parts counted, using MDI.
The number of required parts and the number of counted parts can be read and written using custom macro variables. These values can also be read using the external data input function.

```
SETTING(TIMER)                        O0000 N00000

  PARTS TOTAL      =       0
  PARTS REQUIRED   =      25
  PARTS COUNT      =      10

  POWER ON         =    0H  0M
  OPERATING TIME   =    0H  0M  0S
  CUTTING TIME     =    0H  0M  0S
  FREE PURPOSE     =    0H  0M  0S
  CYCLE TIME       =    0H  0M  0S
             DATE  =  2002/04/10
             TIME  =   16:20:30
>_
 MDI **** *** ***           16:20:30
[ OFFSET ][ SETTING ][      ][      ][ (OPRT) ]
```

26.5 SOFTWARE OPERATOR'S PANEL

In this function, functions of switches on the machine operator's panel is done by operation on the MDI panel. Mode selection and jogging override, etc. can be operated by setting operation via the MDI panel with this function, thus allowing commitance of corresponding switches on the machine operator's panel.
This function is valid only when the screen is displayed with operator's panel. Move cursor with the cursor operation keys, and select various operations, viewing the screen.
The following operations can be done via the MDI panel:

A Model selection

B Manual pulse generator feed axis selection (available only with manual handle 1)
Move distance selection per pulse of manual pulse generator

C Rapid traverse override
Jogging speed override
Feedrate override

D Optional block skip (Block delete)
Single block
Machine lock
Dry run

E Memory protect

F Feed hold

G Jogging/incremental feed axis direction selection
Manual rapid traverse selection

H General-purpose switch: Eight general-purpose switches are provided and each of these
switches can be named by up to eight alphanumeric characters.

There is a parameter per groups A to G shown above, which decides validity of operation function by MDI panel.

```
OPERATOR'S PANEL                          O0000 N00000

 MODE : █MDI  AUTO  EDIT  STEP  JOG  ZRN

  STEP  MULT.  :  *1  █10  *100
  RAPID OVRD.  :  100%  50%   25%  █0
  JOG FEED     :   1.0%
           ██████████***************
  FEED OVRD.   :  140%
           ██████████████*****
ACTUAL POSITION (ABSOLUTE)
    X      0.000          Z      0.000

                               S     0 T0000
EDIT **** *** ***            09:36:48
[ MACRO ] [      ] [  OPR  ] [ TOOLLF ] [      ]
```

```
OPERATOR'S PANEL                           O0000 N00000

 BLOCK SKIP   :   OFF    ON
 SINGLE BLOCK :   OFF    ON
 MACHINE LOCK :   OFF    ON
 DRY RUN      :   OFF    ON
 PROTECT KEY  :   PROTECT  RELEASE
 FEED HOLD    :   OFF    ON


ACTUAL POSITION (ABSOLUTE)
    X     0.000        Z     0.000

                                S    0 T0000
EDIT **** *** ***          09:36:48
[ MACRO ][      ][ OPR ][ TOOLLF ][      ]
```

26.6 DIRECTORY DISPLAY OF FLOPPY CASSETTE

File names in the floppy cassette (FANUC CASSETTE F1) and program file (FANUC PROGRAM FILE Mate can be listed on the display (directory display). Each file name of up to 17 letters can be displayed in directory display.

Files in the floppy cassette are:

Part program, parameter/pitch error compensation data, tool compensation data, and etc.

When part program in part program memory is written into the floppy cassette, program number can be given to it as a file name. When NC parameter is written into the floppy cassette, "PARAMETER" is given them as a fixed name. When tool compensation data is written into the floppy cassette, "OFFSET" is given to it as a fixed name.

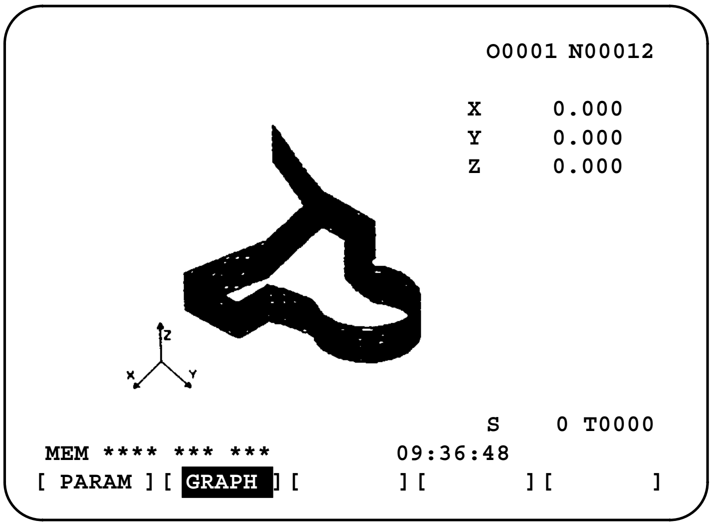
```
DIRECTORY(FLOPPY)                    O0000 N00000
  NO.   FILE NAME                    (METER) VOL
  0001  PARAMETER                       46.1
  0002  ALL.PROGRAM                     12.3
  0003  O0001                            1.9
  0004  O0002                            1.9
  0005  O0003                            1.9
  0006  O0004                            1.9
  0007  O0005                            1.9
  0008  O0010                            1.9
  0009  O0020                            1.9



EDIT **** *** ***         09:36:48
[ F SRH ][ READ  ][ PUNCH ][DELETE ][       ]
```

26.7 GRAPHIC DISPLAY FUNCTION

26.7.1 Graphic Display Function

This function allows display of tool path on the screen, making program check easier. The following functions are offered.

- Tool path of the machining program can be displayed. Machining process can be checked just by viewing the tool path drawing on the screen.
  Program check before machining can be done by displaying the programmed locus on the screen.
- For M series, display is possible with the XY plane, YZ plane, ZX plane, or isometric; for T series, with the XZ plane.
- Scaling of the screen is possible.



Graphic display of tool path (M series)

26.7.2 Dynamic Graphic Display

Created programs can be checked visually by displaying them using graphic data.

Dynamic graphic display function (for M series)

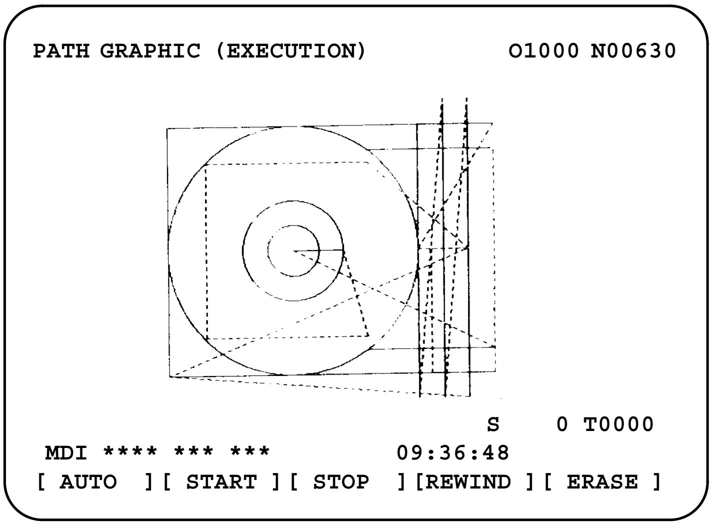
Graphic data can be displayed in the following two drawing modes:

- **Tool path drawing mode**

Tool paths are drawn with lines so programs can be checked closely.

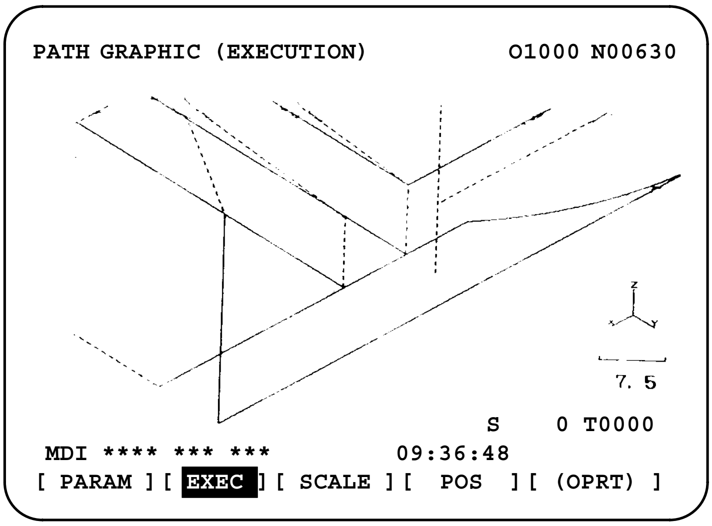
- Because tool paths are drawn at a high speed, programs can be checked quickly.

Two-dimensional drawing



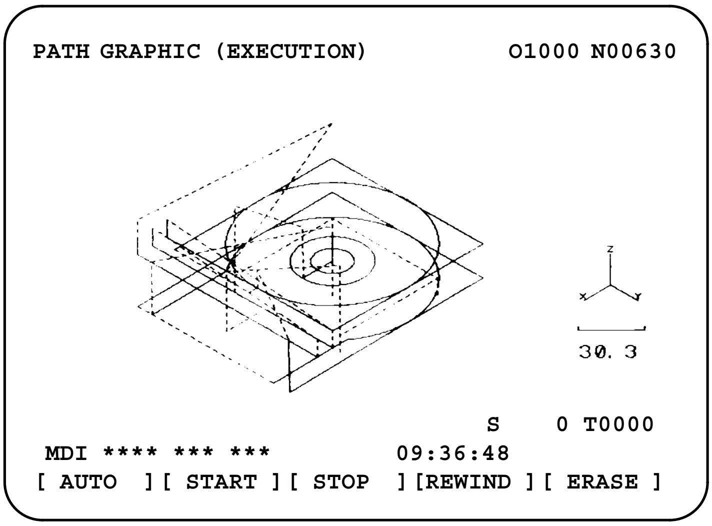
- With the automatic scaling function, figures can be drawn on the center of the screen at a desired magnification.
- On a drawing, any part of a figure can be magnified easily by specifying its center and scale.

Enlarged-view drawing



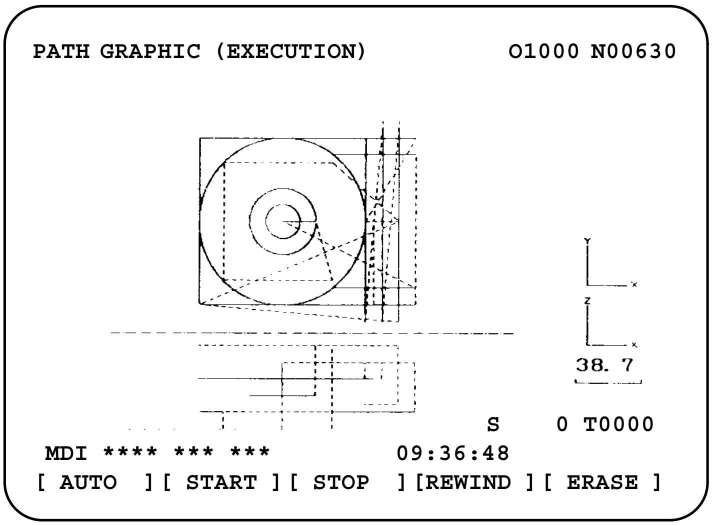
- In addition to two-dimensional drawings, isometric projection drawings and biplanar drawings can be created.

Isometric projection drawing



- Because the current position of a tool is marked on the drawn tool path, the progress of machining can be monitored accurately.

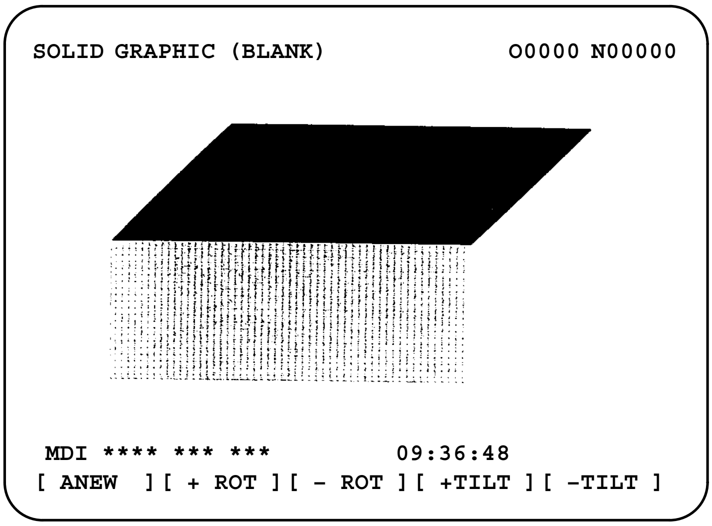
Biplanar drawing



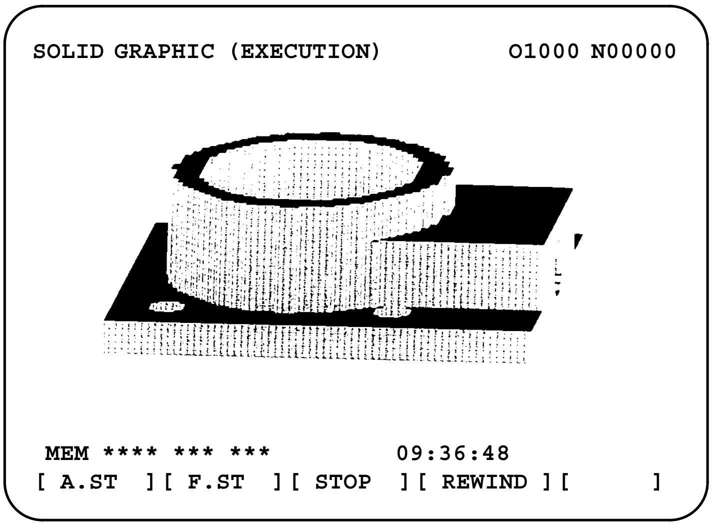
- **Machining profile drawing mode**

- The profile of a workpiece that changes as the tool moves can be simulated and drawn three-dimensionally, making it easier to check programs visually.

Blank figure

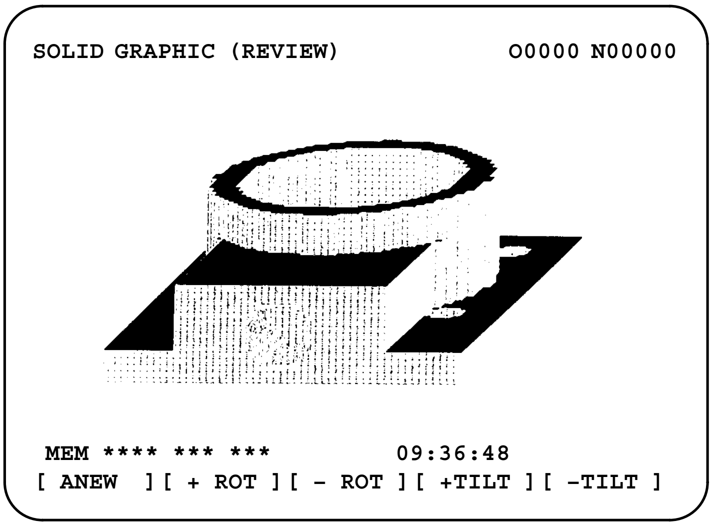


Final figure

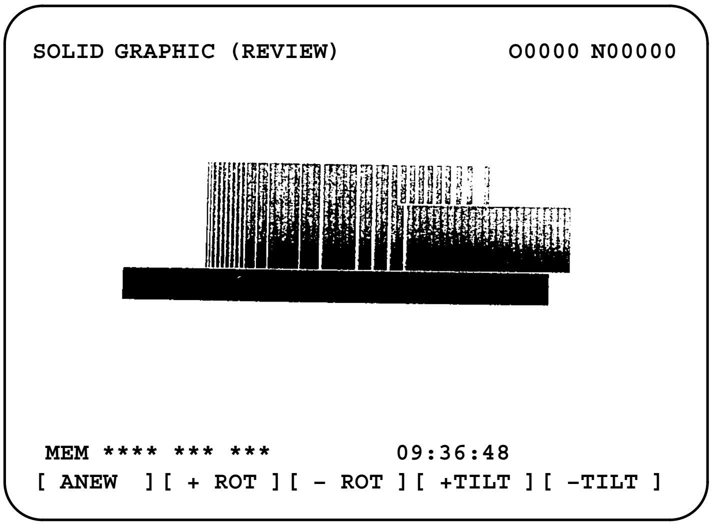


- The coordinate axes and projection angles can be changed at the operator’s option.

Modification of a coordinate axis (inclination)

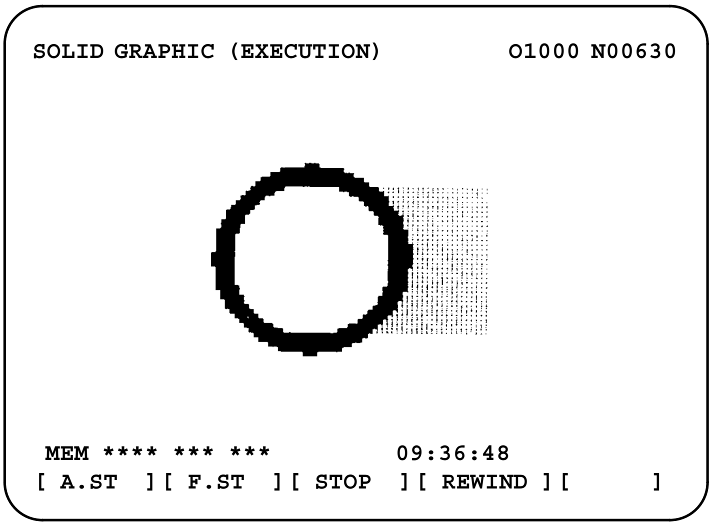


Modification of a coordinate axis (inclination)

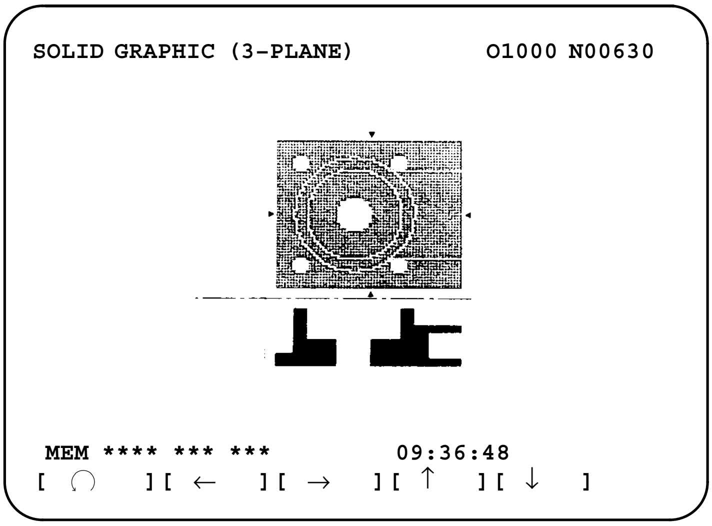


- In addition to three–dimensional drawings, two–dimensional drawings and tri–planar drawings can be created.

Two-dimensional drawing



Tri-planar drawing



Dynamic graphic display function (for T series)

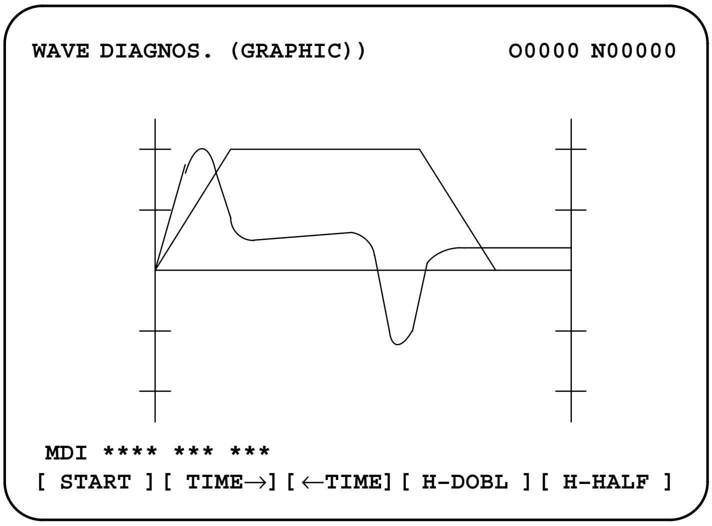
The following two display modes are available.
These functions are provided for conversational automatic programming function for lathe.

- **Tool path drawing mode**

Movement of the tool tip is drawn with fine lines.

26.8 SERVO WAVEFORM FUNCTION

The waveforms of servo data items (errors, torques, timing pulses, etc.) and signals between the CNC and the PMC can be displayed.



On this screen, the sampling period (6 to 32767 ms) and drawing start conditions can be specified.

26.9 SCREENS FOR SERVO DATA AND SPINDLE DATA

26.9.1 Servo Setting Screen

On the servo setting screen, parameters required for standard initialization of the servo motor are listed. The parameters can also be set.

```
SERVO SETTING                          O0000 N00000

                                X AXIS      Y AXIS
 INITIAL SET BIT               00000011    00000001
 MOTOR ID NO.                        12          12
 AMR                           00011111    00011111
 CMR                                  2           2
 FEEDGEAR N                           3           3
  (N/M)    M                         10          10
 DIRECTION SET                      111         111
 VEROCITY PULSE NO.                8000        8000
 POSITION PULSE NO.                8000        8000
 REF COUNTER                       8000        8000

 MDI **** *** *** ***       09:36:48
[ SV.SET ][ SV.TUN ][      ][      ][ (OPRT) ]
```

26.9.2 Servo Adjustment Screen

On the servo adjustment screen, parameters required for basic adjustment of the servo motor and statuses being monitored are listed for each axis.

```
SERVO SETTING                          O1000 N00000
 X AXIS
     (PARAMETR)                  (MONITOR)
FUNC.BIT      00110100   ALARM 1       00110100
LOOP GAIN         3000   ALRAM 2       00110100
TUNING ST.           1   ALARM 3       00000000
SET PERIOD          50   ALARM 4       00000000
INT.GAIN           251   LOOP GAIN         3000
PROP.GAIN        -2460   POS ERROR          100
FILTER            2450   CURRENT %           50



>_
 MEM STAT MTN *** ***       09:36:48
[ SV.SET ][ SV.TUN ][      ][      ][ (OPRT) ]
```

26.9.3 Spindle Setting Screen

On the spindle setting screen, parameters required for standard initialization of the serial spindle are listed. The parameters can also be set.

```
SPINDLE SETTING                          O0000 N00000

 GEAR SELECT    :1
 SPINDLE        :S11

 (PARAMETER)
  GEAR RATIO                          50
  MAX SPINDLE SPEED                 3000
  MAX MOTOR SPEED                   6000
  MAX C AXIS SPEED                   100


>_
 MDI **** *** ***          09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]
```

26.9.4 Spindle Adjustment Screen

On the spindle adjustment screen, parameters required for basic adjustment of the serial spindle and statuses being monitored are listed.

```
SPINDLE TUNING                           O1000 N00000

 OPERATION     : SYNCHRONIZATION CONTROL
 GEAR SELECT   : 1
 SPINDLE      : S11
     (PARAMETER)               (MONITOR)
 PROP.GAIN        -2460   MOTOR              100
 INT.GAIN           241   SPINDLE            150
 LOOP GAIN         3000   POS ERR S1         100
 MOTOR VOLT          30
 ZRN GAIN %         100   SYN.ERROR          128
 REF.SHIFT         2046

>_
 MEM STAT MTN *** ***      09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]
```

26.9.5 Spindle Monitor Screen

On the spindle monitor screen, various data items related to the spindle are listed.

```
SPINDLE MONITOR                              O1000 N00000
 ALARM          : AL-27(PC DISCON.)
 OPERATION      : SP.CONTOURING CONTROL
 FEED SPEED     :      100 DEG/MIN
 MOTOR SPEED    :      150 RPM
                       0    50   100  150  200(%)
 LOAD METER(%)

 CONTROL INPUT : MRDY *ESP ORCM

 CONTROL OUTPUT  : ORAR SST


>_
 MEM **** *** ***          09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]
```

26.10 SYSTEM CONFIGURATION DISPLAY FUNCTION

The configurations of software and hardware required for maintenance of the CNC are displayed.
The system configuration display function provides the following three screens:

- Slot information screen
- Software information screen
- Hardware (module) information screen

Slot information

```
SYSTEM CONFIG                     O1234 N56789

SLOT MODULE_ID SERIES VERSION
 00    3008 :40  B1H1    0001
 01    00CF :66  B435    0001
 03    019D :41  4068    0001
  ↑      ↑    ↑    ↑        ↑
  1      2    3    4        5





MEM **** *** ***          18:46:43
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][      ]
```

1 **Slot No.**

2 **Module ID**

3 **Software ID**

4 **Software series**

5 **Software edition**

Software information

```
SYSTEM CONFIG(SOFTWARE)           O1234 N56789

 SYSTEM     B0H1     0001
            BASIC+OPTION-A1         ←4
 SERVO      90B0     0001
 PMC(SYS)   406G     0001
            406G     0001
 PMC(LAD)   FS16     0001
 MACRO LIB  BZG1     0001
 BOOT       60M5     0001
 GRAPHIC-1  60V5     0001
 GRAPHIC-2  60V6     0001
 EMBED ETH  656A     0001
    ↑         ↑         ↑
    1         2         3
MEM **** *** ***          19:14:23
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][      ]
```

1 **Software type**

2 **Software series**

3 **Software edition**

4 **Contents of ROM (system ROM only)**

Hardware (module) information

The slot number, board name, modules mounted on the board are displayed for each slot.

```
SYSTEM CONFIG(MODULE)          O1234 N56789
                1
SLOT   00 MOTHER BOARD       ← 2

 AXIS CTRL CARD          :    08
 DISPLAY CTRL CARD       :    OE
 CPU CARD                :    11
 FROM DIMM               :    C1
 SRAM DIMM               :    05
 DRAM DIMM               :    A9
 PMC CPU                 :    01
         ↑                    ↑
         3                    4

MEM **** *** ***              19:33:34
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][       ]
```

1 **Slot number (The slot number corresponds to the number displayed on the slot information screen.)**

2 **Name of the PC board inserted in the slot**

3 **Hardware (modules) mounted on the PC board**

4 **Types of hardware (modules), mounted/not mounted**

26.11 HELP FUNCTION

When an alarm occurs, or when the operator is not certain what to do next, pressing the HELP key on the MDI panel displays detailed alarm information or instructions for operation.
One of the following three screens can be displayed:

- On the alarm detail screen, detailed information on the alarm currently activated is displayed. Using this information, the operator can identify the cause of the alarm and what action to take. Any alarm information can be displayed on this screen.
- On the operation instruction screen, when the operator is not sure of what to do next during CNC operation (i.e., program editing and data input/output) necessary instructions are displayed.
- Parameter numbers are listed on the parameter list screen. When the number of the parameter to be set or referenced is unknown, bring up this screen.

❑ An alarm detail screen for when an alarm (P/S 094) is activated.

```
HELP (ALARM  DETAIL)                O1234 N00001

 NUMBER: 094
 M'SAGE: P TYPE NOT ALLOWED (COORD CHG)
 FUNCTION: RESTART PROGRAM
 ALARM:
  WHEN COORDINATE SYSTEM SETTING IS
  CONDUCTED AFTER HOLDING
  AUTOMATIC OPERATION, P-TYPE (WHEN TOOL IS
  DAMAGED) PROGRAM
  RESTART CANNOT BE EXECUTED.


                                    S     0 T0000
 MEM **** *** *** ALM      09:36:48
[  ALAM  ][  OPR   ][  PARA  ][      ][ (OPRT) ]
```

❑ Parameter list screen

```
HELP (PARAMETER TABLE)              O1234 N00001
                                             1/4
 *SETTING                           (NO.0000 – )
 *READER/PUNCHER INTERFACE          (NO.0100 – )
 *AXIS CONTROL/SETTING UNIT         (NO.1000 – )
 *COORDINATE SYSTEM                 (NO.1200 – )
 *STROKE LIMIT                      (NO.1300 – )
 *FEED RATE                         (NO.1400 – )
 *ACCEL/DECELERATION CTRL           (NO.1600 – )
 *SERVO RELATED                     (NO.1800 – )
 *DI/DO                             (NO.3000 – )


                                    S     0 T0000
 MEM **** *** ***          09:36:48
[  ALAM  ][  OPR   ][  PARA  ][      ][ (OPRT) ]
```

❏ Operation instruction screen

```
HELP (OPERATION METHOD)              O1234 N00001
<<1. PROGRAM EDIT>>                          1/4
 *DELETE ALL PROGRAMS
  MODE   : EDIT
  SCREEN: PROGRAM
  OPR    : (O–9999) – <DELETE>

 *DELETE ONE PROGRAM
  MODE   : EDIT
  SCREEN: PROGRAM
  OPR    : (O+PROGRAM NUMBER) – <DELETE>


>_                                   S    0 T0000
 MEM **** *** ***           09:36:48
[  ALAM  ][  OPR  ][  PARA  ][      ][ (OPRT) ]
```

26.12 DATA PROTECTION KEY

A data protection key can be installed on the machine side for protection of various NC data. The following four input signals are offered, according to type of data to be protected.

- **KEY 1**
  Allows input of tool compensation amount and work zero point offset amount.
- **KEY 2**
  Allows setting data input and macro variable input.
- **KEY 3**
  Allows part program input and editing.
- **KEY 4**
  Allows PMC data (counter, data table) input.

26.13 DISPLAYING OPERATION HISTORY

This function displays a history of the key and signal operations, performed by the operator, upon the occurrence of a failure or alarm. The history can also be displayed for previously generated alarms.
The following history data is recorded :

- MDI key/soft key operation sequences
  Example : A to Z, <POS>, <PAGE↑>, [SF1]
- On/off status transitions of selected input and output signals
  Example : G0000.7↑, SBK↑
- NC alarm information
  Example : P/S0010
- Time (date, time) stamp
  Example : 02/04/10
  09:27:55

The history data can be output to an input/output device, connected via the reader/punch interface. Previously output history data can be input from an input/output device.

26.14 REMOTE DIAGNOSTIC

The remote diagnosis function allows you to use a commercially available personal computer as a service terminal and connect it to a CNC via a Ethernet for monitoring the CNC status and modifying CNC data from the personal computer.
The machine remote diagnosis package and Ethernet function are required.

26.15 DIRECTORY DISPLAY AND PUNCH FOR EACH GROUP

CNC programs stored in memory can be grouped according to their names, thus enabling the listing and output of CNC programs on a group–by–group basis.

To assign multiple CNC programs to a single group, assign names to those programs, beginning each name with the same character string. By searching through all the program names for a specified character string, the program numbers and names of all programs having names including that string are listed.

The CNC programs within a specified group can also be output.

❑ Group–unit program list screen displayed when a search is made for “GEAR–1000*”

```
 PROGRAM DIRECTORY(GROUP)       O0001 N00010

          PROGRAM(NUM.)        MEMORY'CHAR.)
     -USED:          60        3321
     -FREE:           2         429
 O0020  (GEAR-1000 MAIN)
 O0040  (GEAR-1000 SUB-1)
 O0200  (GEAR-1000 SUB-2)
 O2000  (GEAR-1000 SUB-3)




>_
 EDIT **** *** *** ***   16:53:25
[  PRGRM  ][  DIR  ][     ][     ][ (OPRT) ]
```

26.16 ERASE CRT SCREEN DISPLAY

Displaying the same characters in the same positions on the screen causes a LCD to degrade relatively quickly. To help prevent this, the screen can be cleared by pressing specific keys. It is also possible to specify the automatic clearing of the screen if no keys are pressed at specified with a parameter. (This cannot be performed for 0*i* with PC functions.)

26.17 PERIODIC MAINTENANCE SCREEN

The periodic maintenance screen shows the current statuses of those consumables that require periodic replacement (backup battery, LCD backlight, touch pad, etc.). An item whose service life has expired is indicated by the machine run time or the like.

```
PERIODICAL MAINTENANCE                 O0001 N12345
  (STATUS)
            ITEM NAME                      REMAIN
   * 01   BATTERY FOR CONTROLLER              0H
  @  02   BATTERY FOR PULSECODER           5000H
     03   LCD BACK LIGHT                  10000H
  @  04   COOLANT                           720H
     05
     06
     07
     08
     09
     10
>_
EDIT **** *** ***          19:27:05
[       ][MAINTE ][       ][       ][(OPRT) ]
```

26.18 MAINTENANCE INFORMATION SCREEN

The history of the maintenance carried out by FANUC service personnel and machine tool builder can be recorded via the screen. The screen has the following features:

- Alphabetical characters can be input from MDI. (Half–size kana can be input only when Japanese display is selected.)
- The recording screen can be scrolled, line by line.
- Edited maintenance information can be read and punched.
- Data can be stored into flash ROM.
- Full–size (shift JIS) codes can be displayed. (Input codes are read only.)

26.19 CONTRAST ADJUSTMENT SCREEN

Some operators may find the LCD difficult to read, depending on their eye level relative to the display. To make a monochrome LCD easier to read, the contrast can be adjusted.

```
SETTING(HANDY)

  PARAMETER WRITE  =1(0:DISABLE   1:ENABLE)
  TV CHECK         =0(0:OFF  1:ON)
  PUNCH CODE       =0(0:EIA  1:ISO)
  INPUT UNIT       =0(O:MM   1:INCH)
  I/O CHANNEL      =0(0–3:CHANNEL NO.)
  SEQUENCE NO.     =0(0:OFF  1:ON)
  TAPE FORMAT      =0(0:NO CNV  1:F15)
  SEQUENCE STOP    =    0(PROGRAM NO.)
  SEQUENCE STOP    =    0(SEQUENCE NO.)

  CONTRAST         (+=[ON:1]  -=[OFF:0])
>_
 MDI **** *** *** BAT 00:00:00
[NO.SRH ][ ON:1  ][ OFF:0 ][+INPUT ][ INPUT ]
```

26.20 ID INFORMATION SCREEN

Overview

When the α*i* servo or α*i* spindle is connected, if each of the units (motor or amplifier) actually connected it has ID information, the ID information can be read and displayed on the CNC screen.

26.20.1 Servo Information Screen

Displaying the servo ID screen

```
SERVO  INFORMATION                O0000 N00000

  X AXIS
 SERVO MOTOR SPEC  A06B-0268-B100
 SERVO MOTOR S/N   C00ZB1111
 PULSECODER SPEC.  A860-2000-T301
 PULSECODER S/N    00000001
 SERVO AMP SPEC.   A06B-6114-H211
 SERVO AMP S/N     V01311111
 PSM SPEC.         A06B-6087-H126#000001
 PSM S/N           V01311111



 MDI **** *** ***      19:12:26
(SYSTEM)(SV-INF)(SP-INF)(      )(      )
```

⁎ If no servo information is recorded, servo information is automatically stored in flash ROM. On the screen, if there is a difference between the servo information in flash ROM and the actual servo information, the corresponding items are preceded by *, as shown below. This allows you to determine whether the configuration of connected units is changed. At this time, the servo information is read from flash ROM.

```
SERVO  INFORMATION                O0000 N00000

  X AXIS
 SERVO MOTOR SPEC  A06B-0268-B100
 SERVO MOTOR S/N   C00ZB1111
 PULSECODER SPEC.  A860-2000-T301
 PULSECODER S/N    00000001
*SERVO AMP SPEC.   A06B-6114-H211
*SERVO AMP S/N     V01311111
 PSM SPEC.         A06B-6087-H126#000001
 PSM S/N           V01311111


>_
 MDI **** *** ***      19:12:26
(SYSTEM)(SV-INF)(SP-INF)(      )((OPRT))
```

Editing the servo ID screen

The servo information recorded or stored in flash ROM can also be edited. See Chapter 6 “Digital Servo” in the Maintenance Manual (B–64115EN).

26.20.2 α*i* Spindle Information Screen

Displaying the spindle ID screen

```
SPINDLE INFORMATION          O0000 N00000

  S1
 SP MOTOR SPEC       A06B-0852-B088#0007
 SP MOTOR S/N        C99XA1234


 SP AMP SPEC         A06B-6102-H106#H520CE
 SP AMP S/N          V0020090601
 PSM SPEC.           A06B-6087-H126#000001
 PSM S/N             V0020031702


>_
 MDI **** *** ***      19:12:05
(SYSTEM)(SV-INF)(SP-INF)(      )(     ))
```

✻ If no spindle information is recorded, spindle information is automatically stored in flash ROM. On the screen, if there is a difference between the spindle information in flash ROM and the actual spindle information, the corresponding items are preceded by *, as shown below. This allows you to determine whether the configuration of connected units is changed. At this time, the spindle information is read from flash ROM.

```
SPINDLE INFORMATION          O0000 N00000

  S1
 SP MOTOR SPEC       A06B-0852-B088#0007
 SP MOTOR S/N        C99XA1234


*SP AMP SPEC         A06B-6102-H106#H520CE
*SP AMP S/N          V0020090601
 PSM SPEC.           A06B-6087-H126#000001
 PSM S/N             V0020031702


>_
 MDI **** *** ***      19:12:05
(SYSTEM)(SV-INF)(SP-INF)(      )(     ))
```

Editing the spindle ID screen

The spindle information recorded or stored in flash ROM can also be edited.
See Chapter 7 “AC Spindle” in the Maintenance Manual (B–64115EN).

26.21 MACHINING CONDITION SELECTING FUNCTION

If the speed–oriented and precision–oriented parameters are set with the advanced preview control function or AI contour control function and then the precision level is set according to the machining conditions during machining, the parameters appropriate to the conditions can be automatically calculated to perform machining.

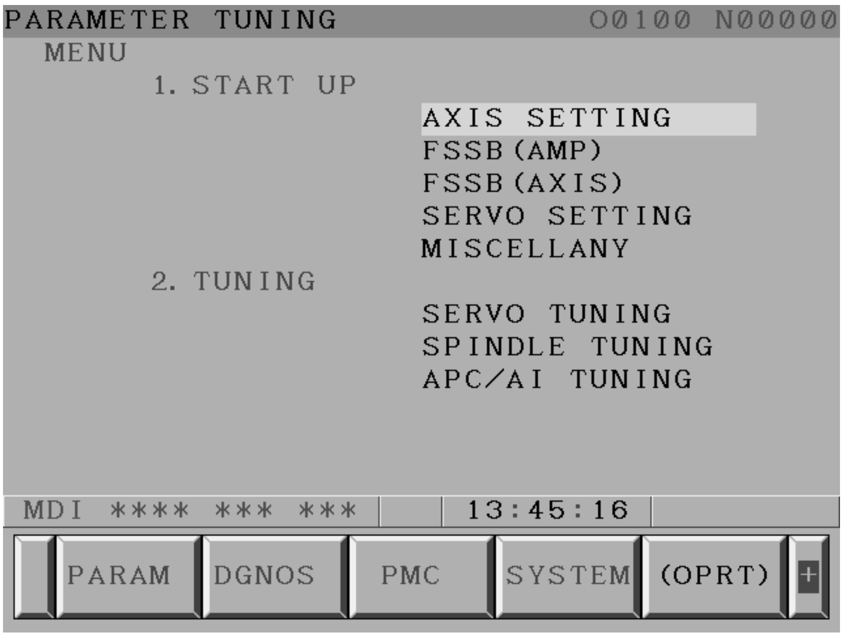
NOTE
This function cannot be used for inch output machines.

1) Set the speed–oriented (precision level 1) and precision–oriented (precision level 10) parameters on the machining parameter tuning screen.
2) A precision level between the speed–oriented (precision level 1) setting and the precision–oriented (precision level 10) setting entered on the machining parameter tuning screen can be selected on the precision level selection screen.
   Since the direct proportionality is established between the two levels, the optimal parameter setting can be selected by selecting a level between them.
   The precision level can be changed using the program as well as on the precision level selection screen.

26.22 PARAMETER SET SUPPORTING SCREEN

The PARAMETER SET SUPPORTING screen allows you to perform the following operations, so you can easily start up or adjust the machine.

- Displaying a minimum set of parameters together that must be set during startup of the machine
- Displaying the servo tuning, spindle tuning, machining parameter tuning screens



27 PART PROGRAM STORAGE AND EDITING

27.1 FOREGROUND EDITING

The following part program storage and editing is possible

- **Program tape registration to the memory**
  - Single program registration
  - Multi program tape registration
- **Program input via MDI**
- **Program deletion**
  - Single program deletion
  - All programs deletion
  - Multi programs deletion by specification the range
- **Program punching**
  - Single program punching
  - All programs punching
  - Multi programs punching by specification the range
- **Program editing**
  - ❑ Change
    - Word change
    - Change of 1-word to multi-words
  - ❑ Insertion
    - Word insertion
    - Multi words, and multi blocks insertion
  - ❑ Deletion
    - Word deletion
    - Deletion to EOB
    - Deletion to the specified word
- **Part program collation**
  Collation of program stored in the memory and program on the tape can be done.
- **Sequence number automatic insertion**
  The sequence number, where a certain increment value is added to the sequence number of the previous block can be automatically inserted at the head of each block in preparation of programs by the part program editing.
  The initial value of sequence number and a certain increment amount can be set.

27.2 BACKGROUND EDITING

Part program storage and editing can be done during machining. The same functions as foreground editing can be performed.
However, it is not possible to delete all programs at one time.

27.3 EXTANDED PART PROGRAM EDITING

The following editing is possible.

- **Conversion**
  - ❑ Address conversion
    An address in the program can be converted to another address. For example address X in the program can be converted to address Y.
  - ❑ Word conversion
    A word in the program can be converted to another word. For example, a programmed M03 can be converted to M04.
- **Program copy**
  A part or all of a program can be copied to make a new program.
- **Program move**
  A part or all of a program can be moved to make a new program.
- **Program merge**
  A new program can be created by merging two programs.
- **Editing in the key–in buffer**
  A word starting from the current cursor position or words up to an EOB can be copied and moved from the program to key–in buffer. In addition, characters input in the key–in buffer can be edited.

27.4 NUMBER OF REGISTERED PROGRAMS

Number of registered programs is 200.

27.5 PART PROGRAM STORAGE LENGTH

The part program storage length can be selected below :
640 m (for 0*i* Mate–TC and 0*i* Mate–MC)
320 m/640 m (for 0*i*–TC and 0*i*–MC).

27.6 PLAY BACK

Program can be prepared by storing machine position obtained by manual operation in the memory as program position. Data other than the coordinate value (M codes, G codes, feed rates, etc.) are registered in the memory by the same operation as part program storage and editing.

27.7 EXTERNAL I/O DEVICE CONTROL

Part program registration and punch can be commanded externally.

- **Program registration**
  A part program can be registered in memory through the connected input device for background editing using the external read start signal.
- **Program punch**
  A part program can be punched through the connected output device for background editing using the external punch start signal.

27.8 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

The following two screens can be displayed with graphic data for guidance in programming in the CNC format:

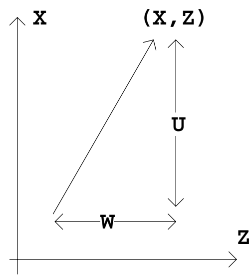
- G code list
- Standard format of a G-code block

Programs can be created by referring to guidelines and entering necessary data interactively.

```
                                        O0010 N00000
G00 :
G01 :
G02 :
G03 :
G04 :
G10 :
G20 :
G21 :
G25 :
G26 :
G27 :
G28 :

EDIT **** *** ***          11:42:31
[      ][      ][      ][      ][      ]
```

↓ When G01 is selected

```
PROGRAM                                O0010 N00000
G01 :

 G01  G    G    G
 X           U
 Z           W
 C
 F
 M
 S
 T
 ;

EDIT **** *** ***          11:56:41
[      ][     ][       ][      ][      ]
```



27.9 PASSWORD FUNCTION

The password function (parameter NE9) can be locked using parameter PASSWD and parameter KEYWD to protect program Nos. 9000 to 9999. In the locked state, parameter NE9 cannot be set to 0. In this state, program Nos. 9000 to 9999 cannot be modified unless the correct keyword is set.

A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

28 DIAGNOSIS FUNCTIONS

28.1 SELF DIAGNOSIS FUNCTIONS

The CNC checks the following itself.

- Abnormality of detection system
- Abnormality of position control unit
- Abnormality of servo system
- Overheat
- Abnormality of CPU
- Abnormality of ROM
- Abnormality of RAM
- Abnormality in data transfer between MDI
- Abnormality of part program storage memory
- Abnormality in tape reader read function
- Abnormality in data transfer between PMC

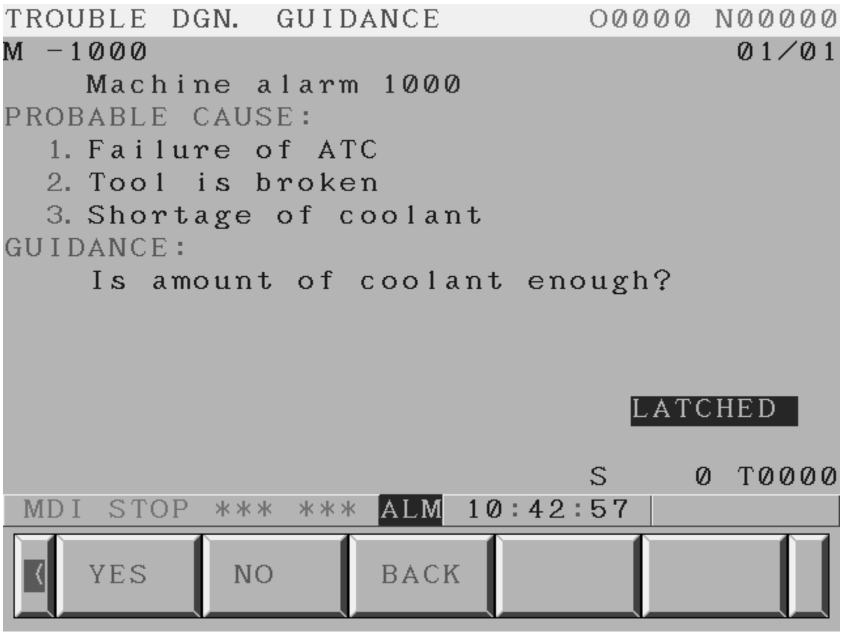
The CNC also checks other items.

Input/output signals from PMC to CNC, or vice versa, and inner status of the CNC can be displayed.

28.2 MACHINE ALARM DIAGNOSIS

Alarms specific to a machine tool can be added as alarms that can be diagnosed on the failure diagnosis guidance screen.
The following alarms can be added.

1. External alarm messages (alarm numbers 1000 to 19999)
2. Macro alarms (#3000) (alarm numbers 3000 to 3200)



29 DATA INPUT/OUTPUT

The NC has the following input/output data.
These data are input/output via various input/output devices as CRT/MDI, tape reader, etc.

- **Input data**
  The NC has the following input data.
  - ❑ Part program
  - ❑ Tool compensation amount and Work zero point offset value
  - ❑ Tool life management data
  - ❑ Setting data
  - ❑ Custom macro common variable
  - ❑ Pitch error compensation data
  - ❑ Parameters
- **Output data**
  The NC has the following output data.
  - ❑ Part program
  - ❑ Tool compensation amount and work zero point offset value
  - ❑ Setting data
  - ❑ Custom macro common variable
  - ❑ Pitch error compensation data
  - ❑ Parameters

29.1 READER/PUNCH INTERFACES

The following can be input/output via the reader/punch interface.

- Part program registration/output
- Tool offset amount, work zero point offset amount, input/output
- Tool life management data input
- Custom macro common variable input/output
- Pitch error compensation data input/output
- Parameter punch input/output

Usually, the screen is switched according to the type of data to be input from or output to an external device; for example, a parameter screen is used for parameter input/output, and a program screen is used for program input/output. However, a single ALL I/O screen can be used to input and output programs, parameters, offset data, and macro variables.

29.2 INPUT/OUTPUT DEVICES

The following Input/Output devices are prepared, which are connectable to the reader/puncher interface.

29.2.1 FANUC Floppy Cassette

When the Floppy Cassette is connected to the NC, machining programs stored in the NC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the NC.

29.2.2 FANUC Program File Mate

The built-in hard disk enables data to be stored and it can be connected to the reader/puncher interface to input data to CNC. This hard disk has a large storage capacity of approximately 50,000 m of paper tape data, so it can register maximum 1024 command programs.
It can be connected to the remote buffer to achieve high-speed transfer of maximum 86.4 kbps.
The hard disk is sealed to be continuously used under the factory environment.

29.2.3 FANUC Handy File

The FANUC Handy File is a compact multi functional input/ouput floppy disk unit for use with various types of FA equipment. Programs can be transferred or edited through operations performed directly on the Handy File or through remote operation from connected equipment.
Compared with media such as paper tape, a 3.5″ floppy disk is both compact and durable, and eliminates noise during input/output. Programs with a total capacity of up to 1.44 MB (equivalent to about 3600 m paper tape) can be saved on a single floppy disk.

29.3 EXTERNAL PROGRAM INPUT

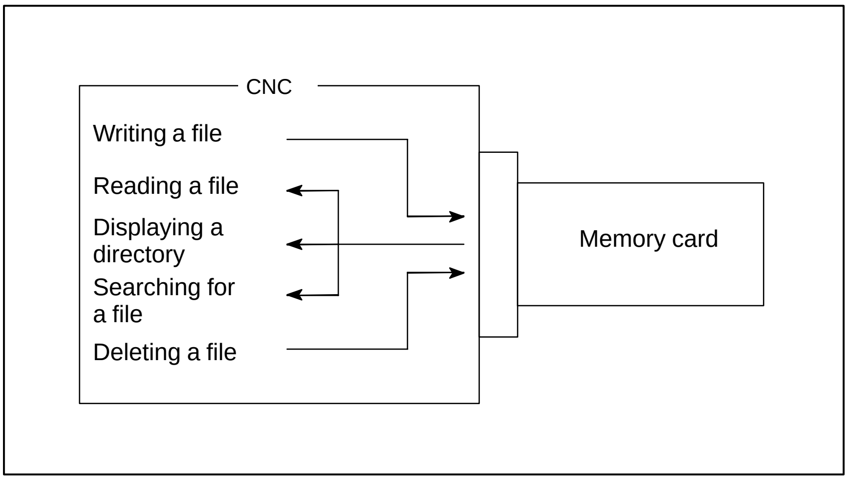
By using the external program input start signal, a program can be loaded from an input unit into CNC memory.
When an input unit such as the FANUC Handy File or FANUC Floppy Cassette is being used, a file can be searched for using the workpiece number search signals, after which the program can be loaded into CNC memory.

29.4 DATA INPUT/OUTPUT USING A MEMORY CARD

Files on a memory card can be referenced, and different types of data such as part programs, parameters, and offset data on a memory card can be input and output in text file format.

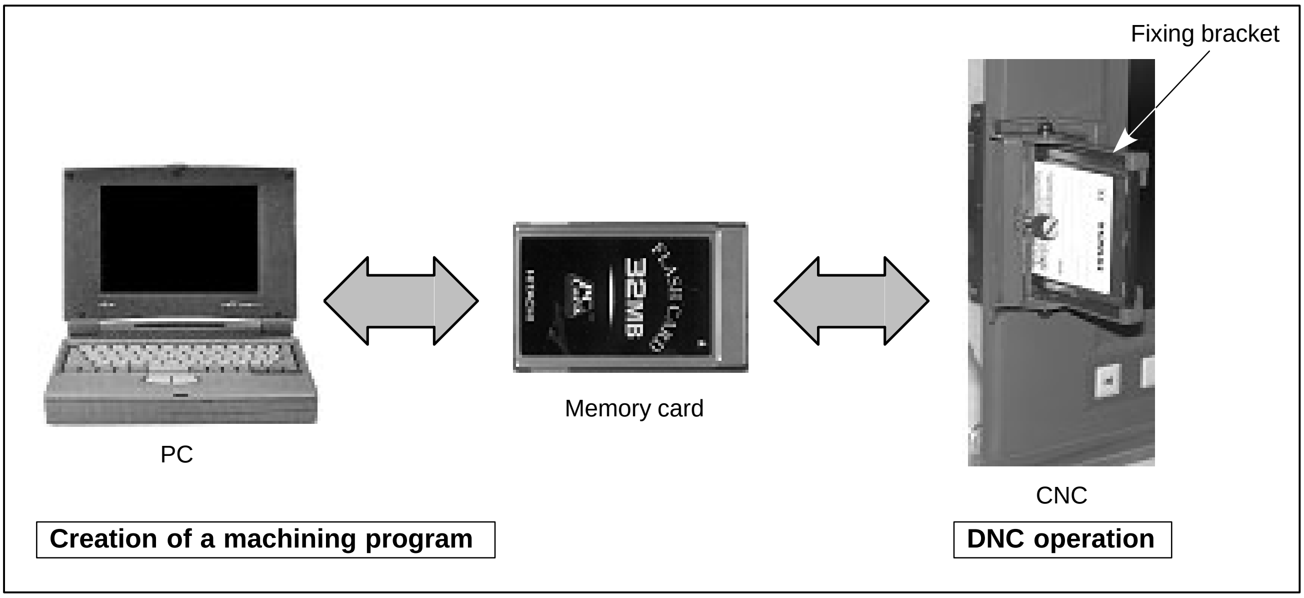
The major functions are listed below.

- Displaying a directory of stored files
  The files stored on a memory card can be displayed on the directory screen.
- Searching for a file
  A search is made for a file on a memory card and, if found, it is displayed on the directory screen.
- Reading a file
  Text–format files can be read from a memory card.
- Writing a file
  Data such as part programs can be stored to a memory card in text file format.
- Deleting a file
  A file can be selected and deleted from a memory card.



29.5 DNC OPERATION WITH MEMORY CARD

To perform DNC operation, load a machining program created on a PC onto a memory card and then inserting the memory card into the PCMCIA port on the front of the monitor.
During DNC operation, secure the memory card with the fixing bracket.



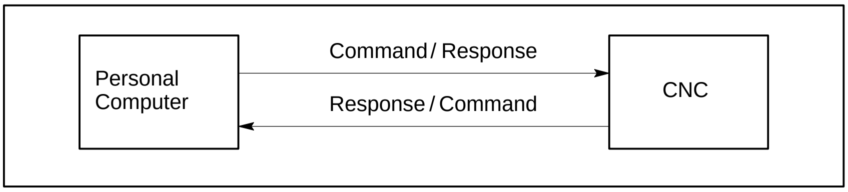
29.6 DNC2 CONTROL

The FANUC DNC2 is a communication protocol enabling data transmission between the FANUC CNC unit and a personal computer by connecting them via the RS–232C interface.

The FANUC DNC2 has the following features:

(1) This protocol is based on the communication protocol LSV2 used by some CNC manufacturers in Europe, so that software can easily be established even with a personal computer.
The RS–232C interface is used to connect a personal computer with the FANUC CNC. The RS–422 interface can also be used to improve the transmission rate.

(2) This protocol is used for one–to–one (point–to–point) communication between one FANUC CNC unit and one personal computer.
The protocol cannot provide multi–point communication between one personal computer and more than one CNC unit.



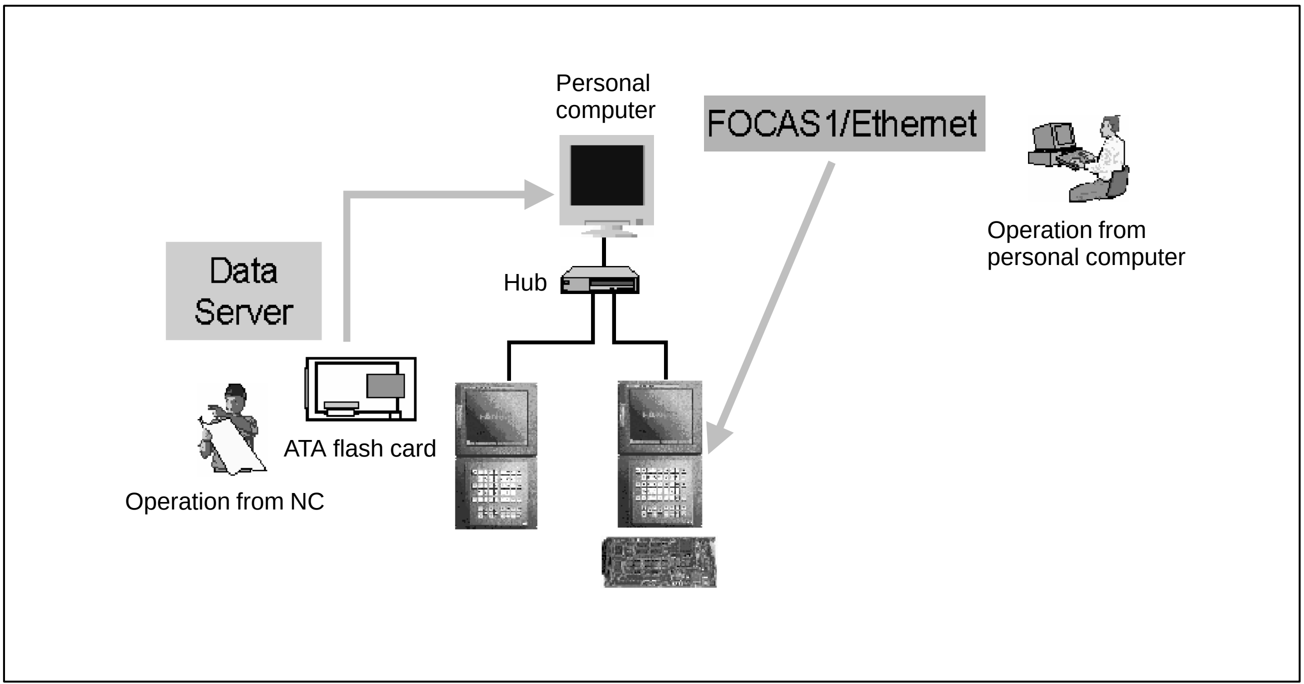
29.7 ETHERNET FUNCTION (OPTION BOARD)

When an option board (the Ethernet board or board with data server) is used, the following Ethernet functions are available:

When the Ethernet board is used
– FOCAS1/Ethernet function

When the board with data server board is used
– FOCAS1/Ethernet function
– Data server function

These functions can be used together at the same time.



29.7.1 FOCAS1/Ethernet Function

The FOCAS1/Ethernet function allows remote control and monitoring of CNCs from the personal computer.
For details, refer to “FAST Ethernet Board/FAST DATA SERVER Operator’s Manual (B–63644EN)” and “FANUC Open CNC FOCAS1/Ethernet CNC/PMC Data Window Library Description.”

NC data transfer

The following NC data can be transferred by operation on the personal computer:

- Data related to controlled axes and spindles
  - ❑ Absolute position, relative position, machine position
  - ❑ Remaining travel distance
  - ❑ Actual feedrate
- NC program
- Directory information in part program storage
- NC file data
  - ❑ Parameters
  - ❑ Tool offset values
  - ❑ Custom macro variables
  - ❑ Workpiece zero point offset values
  - ❑ Settings
  - ❑ P code macro variables
  - ❑ Pitch error compensation data
- Tool life management data
- History data
  - ❑ Operation history data
  - ❑ Alarm history data
- Data related to servo systems and spindles
- Data related to profile diagnosis
- Modal data
- Diagnostic data
- A/D conversion data
- Alarm information
- NC system identification information
- PMC data
  - ❑ Addresses G, F, Y, X, A, R, T, K, C, D

Remote control

The following operations can be performed by operation on the personal computer:

- Selecting NC programs
- Deleting NC programs
- External reset

Operation

The following operation can be performed by operation on the personal computer:

- DNC operation

29.7.2 Data Server Function

The data server function can perform NC data transfer and DNC operation by using FTP.
The data server function operates mainly as an FTP client. It also operates as an FTP server.
The data server function uses the ATA flash card included in (attached to) the data server board as an NC data storage area. For details, refer to "FAST Ethernet Board/FAST DATA SERVER Board Operator's Manual (B–63644EN)."

NC data transfer (between the personal computer and the ATA card on the data server board] <FTP client>

By operation on an NC, this function is operated as an FTP client and provides the following services:

- GET
- MGET
- PUT
- MPUT
- DIR
- DEL

NC data transfer [between the personal computer and the ATA card on the data server board] <FTP server>

By operation on the personal computer, the function is operated as an FTP server and provides the following services:

- GET
- MGET
- PUT
- MPUT
- DIR
- DEL

NC data transfer [between the hard disk on the data server board and part program storage]

The following NC data can be transferred by operation on an NC:

- NC programs
- NC file data
  - ❏ Parameters
  - ❏ Tool offset values
  - ❏ Custom macro variables
  - ❏ Workpiece origin offset values
  - ❏ Pitch error compensation data
  - ❏ M code group
- History data
  - ❏ Operation history data

Operation [between the hard disk on the data server board and part program storage]

The following operations can be performed by operation on an NC:

- DNC operation
- DNC operation by subprogram call (M198)

Operation [between the personal computer and part program storage]

The following operations can be performed by operation on an NC:

- DNC operation
- DNC operation by subprogram call (M198)

ATA flash card management

ATA flash card management can be performed by operation on an NC.

- ATA flash card formatting
- ATA flash card check
- Display of a list of files on the ATA flash card

29.8 POWER MATE CNC MANAGER

When the power mate CNC series is used as an additional axis (slave) of the CNC, the power motion manager allows the slave data to be displayed and set by the CNC.
The power mate CNC manager enables the following display and setting:

(1) Current position display (absolute/machine coordinates)

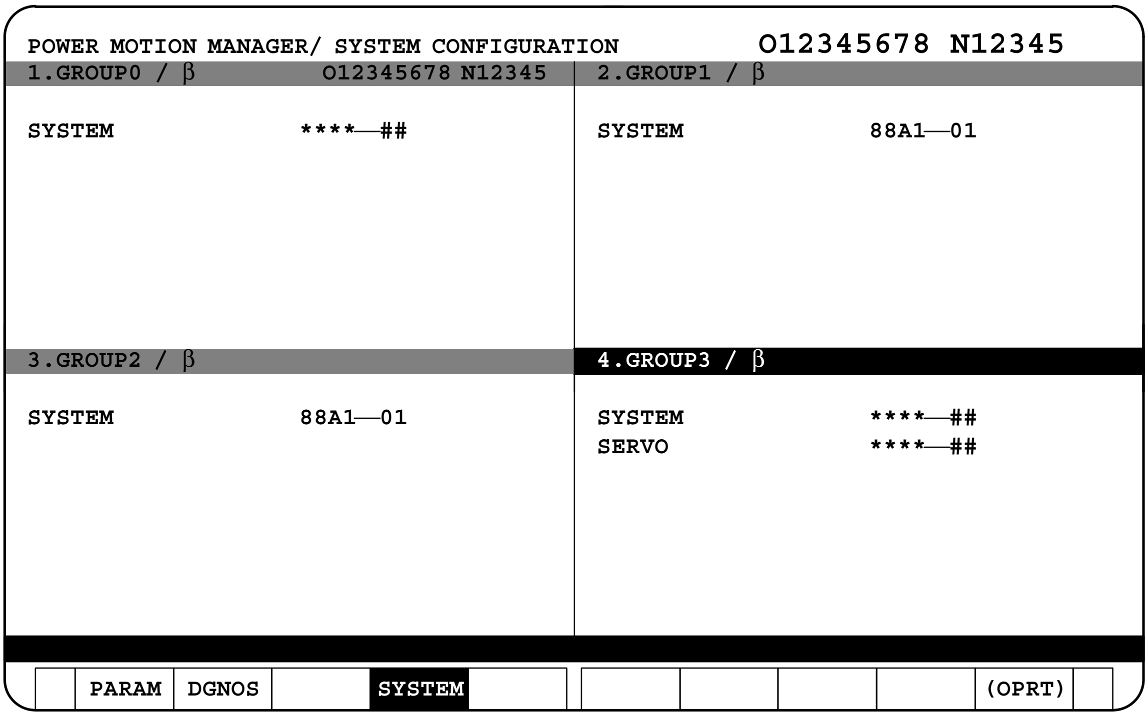
(2) Parameter display and setting

(3) Diagnosis display

(4) System configuration screen display

(5) Alarm display

The power mate CNC series that can be used as the slave is a β amplifier with I/O Link.



The figure above shows a sample four–slave display screen on a display unit with twelve soft keys. A unit with seven soft keys can also display the four–slave display screen.

29.9 FIELD NETWORKS

The field networks listed below are supported to transfer DI/DO signals assigned to PMC addresses to other CNCs or other vendors' devices that conform to the same communication standards.

PROFIBUS–DP

PROFIBUS–DP is a communication function defined by the PROFIBUS Association.
PROFIBUS–DP contains master and slave functions. The CNC can support both functions simultaneously.
For details, refer to "FANUC Profibus–DP Board Operator's Manual (B–62924EN)."

DeviceNet

DeviceNet is a communication function defined by Open DeviceNet Vendor Association, Inc. (ODVA). DeviceNet contains master and slave functions. The CNC can support both functions at the same time.
For details, refer to "FANUC DeviceNet Board Operator's Manual (B–63404EN)."

30 SAFETY FUNCTIONS

30.1 EMERGENCY STOP

With the emergency stop, all commands stops, and the machine stops immediately. Connect the "emergency stop" signal both to the control unit and to the servo unit side.
When emergency stop is commanded, servo excitation is also reset, and servo ready signal will also turn off. Move distance of the machine will still be reflected in the actual position and machine position will not be lost (Follow up function). After resetting the emergency stop, operation can thus be continued without need of another reference point return.

To design a safe machine tool, use an emergency stop signal for it properly.

The emergency stop signal is intended to bring a machine tool to an emergency stop. It is input to the CNC control unit, servo amplifier, and spindle amplifier. Generally, the B contact of a pushbutton switch is used to input the emergency stop signal.

Closing the contact used for the emergency stop signal (*ESP) releases the CNC unit from an emergency stop state, thus enabling it to control and operate the servo motor and spindle motor.

Opening the contact used for the emergency stop signal (*ESP) resets the CNC unit and brings it to an emergency stop. Opening the contact also decelerates the servo motor and spindle motor to a stop.

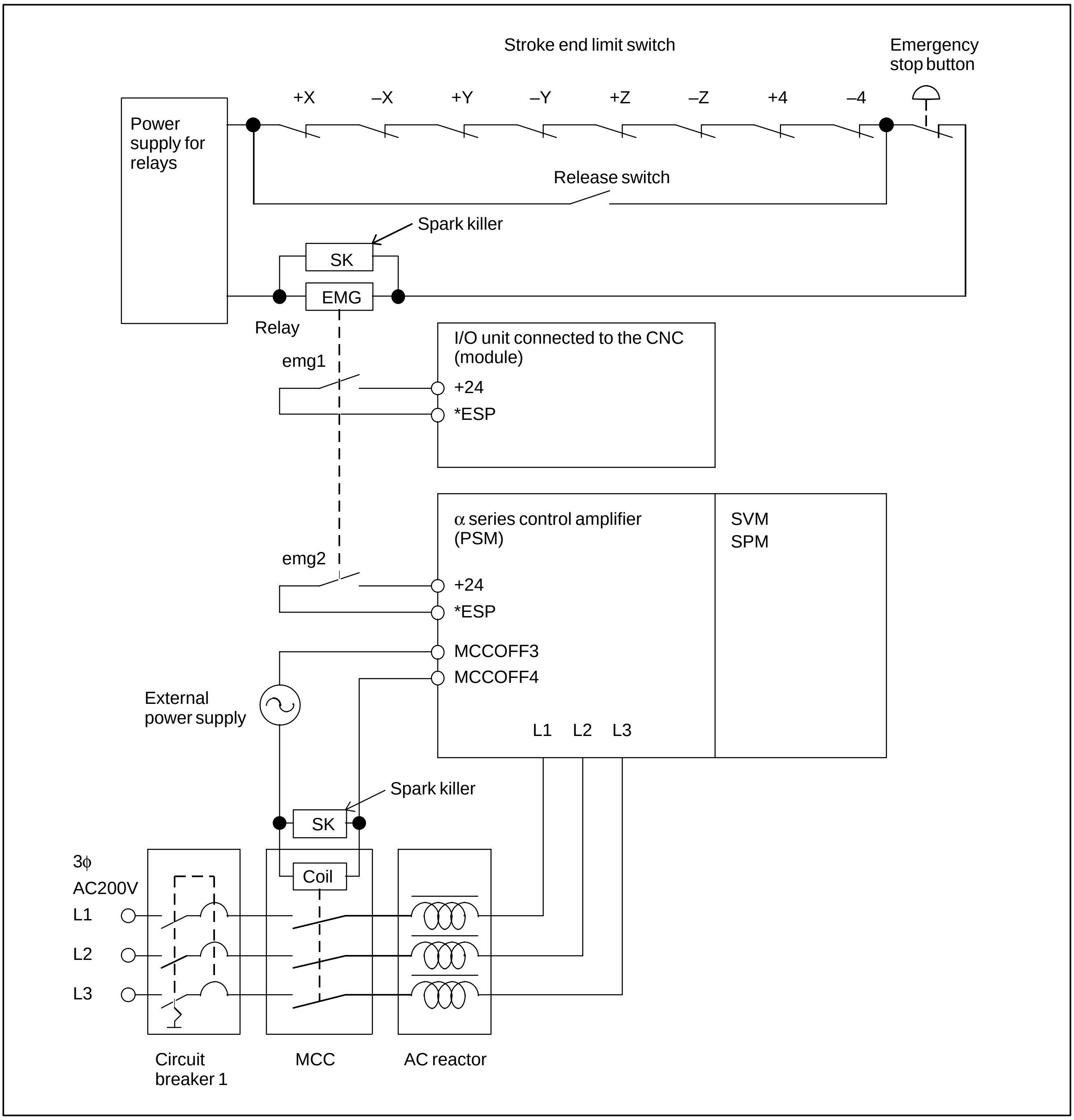
Switching off the electric power of the servo amplifier for a servo motor applies a dynamic brake to the servo motor. If the servo motor is used for a horizontal axis, however, a load on the horizontal axis may drop to cause the servo motor to rotate. To avoid this unintended motion, use a servo motor with a brake or use another appropriate provision.

Switching off the electric power of the servo amplifier for a spindle motor suddenly allows the spindle motor to keep rotating from force of habit, which can be dangerous.
Avoiding this danger requires a control function that detects when the emergency stop signal (*ESP) contact becomes open, and makes sure that the spindle motor decelerates to a stop, then switches off the electric power.

The FANUC servo amplifier αi series is designed with considerations on the behavior mentioned above. Just supply an emergency stop signal to the power supply module (hereafter called PSM) of the servo amplifier αi series. The PSM outputs an electric power MCC control signal. This signal can be used to switch on and off the electric power supplied to the power supply module.

Basically, this CNC control unit is designed to use a software limit function to detect overtravel, so an ordinary overtravel detection limit switch is unnecessary. However, a stroke end limit switch must be provided and connected to an emergency stop signal so that the emergency stop signal can cause the machine to stop if the machine goes over the software limit because of a servo feedback system failure.

The following diagram shows an example of connecting an emergency stop signal when the CNC control unit and αi series servo amplifier are used.



CAUTION

When connecting the CNC unit to a spindle motor and amplifier from a manufacturer other than FANUC, you should develop a sequence that, if the emergency stop signal contact becomes open when the spindle motor is running, decelerates the spindle motor to a stop safely, according to the respective manuals.

30.2 OVERTRAVEL FUNCTIONS

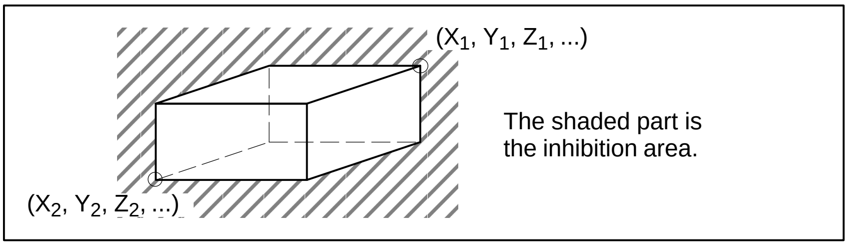
30.2.1 Overtravel

When the movable section has gone beyond the stroke end, a signal is output, the axis decelerates to a stop, and overtravel alarm is displayed. All directions on all axes has overtravel signals.

30.2.2 Stored Stroke Check 1

The movable section of the machine is parameter set in machine coordinates value. If the machine moves beyond the preset range, it decelerates to a stop and alarm is displayed. (This function is valid after manual reference point return at power on.)
This function can be used instead of hardware overtravel limit switch. When both is equipped with, both are valid.
Unlike overtravel detection, stored stroke check 1 checks whether the distance between the current position and that at which the tool will be stopped after deceleration exceeds the limit.

- For manual operation, parameter setting can be made to stop tool movement along an axis when the tool is on a boundary with the inhibition area and generate an alarm.
- For manual operation, parameter setting can be made to output just a stroke limit arrival signal without generating any alarm when the tool enters the inhibition area. (Movement along an axis is stopped.)



• Automatic alarm release

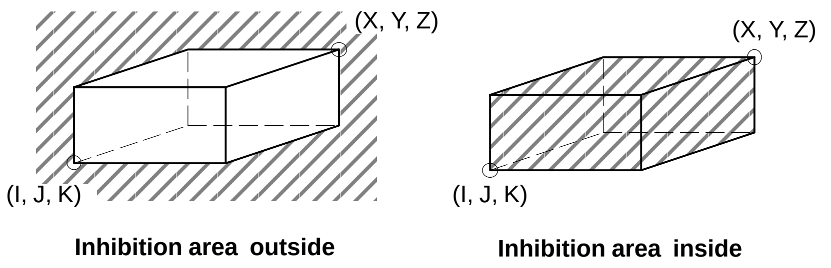
After an OT alarm is generated, moving the tool along the axis to the movable range can release the OT alarm without reset operation. Whether to enable automatic release is specified by parameter setting.

30.2.3 Stored Stroke Check 2 (G22, G23)

M series

An inhibition area can be specified inside or outside an area set by parameter or by program. Command distance from the machine coordinates zero point for limit positions. This function is valid after manual reference point return right after the power on. When specifying the limits with program, limits or axes X, Y, Z can be set.
The inhibition area can be changed according to the workpiece. The parameter decides whether the inhibition area is outside or inside the specified area.

- Parameter setting can be made to generate an alarm before the inhibition area is entered.



Format

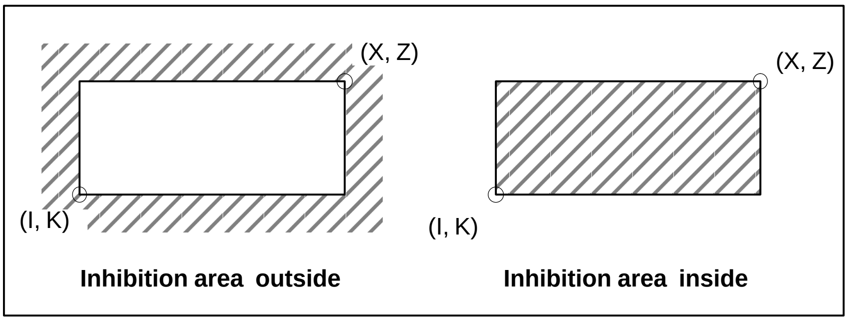
G22 X_ Y_ Z_ I_ J_ K_ ;

On/off of stored stroke check 2 is commanded by program as follows:
G22 : Stored stroke check function on
G23 : Stored stroke check function off

30.2.4 T series Stored Stroke Checks 2 and 3 (G22, G23)

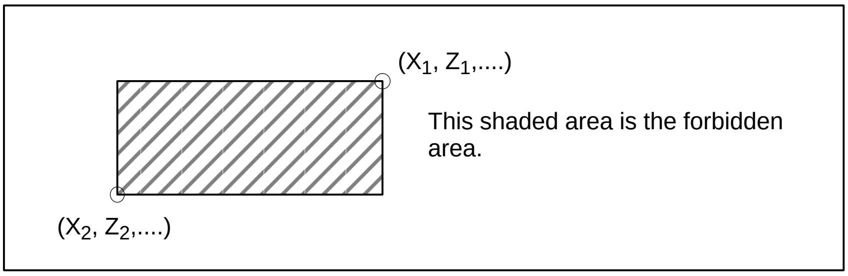
Stored stroke check 2 (G22, G23)

The designation of the forbidden area can be specified by parameters or program.
The forbidden area can be changed for each workpiece. Selection between inside or outside as the forbidden area is made by parameters.



Stored stroke check 3

Inside the area specified by parameters is the forbidden area.



Format

G22 X_ Z_ I_ K_ ;

On/off of stored stroke check 2 is commanded by program as follows:
G22 : Stored stroke check function on
G23 : Stored stroke check function off

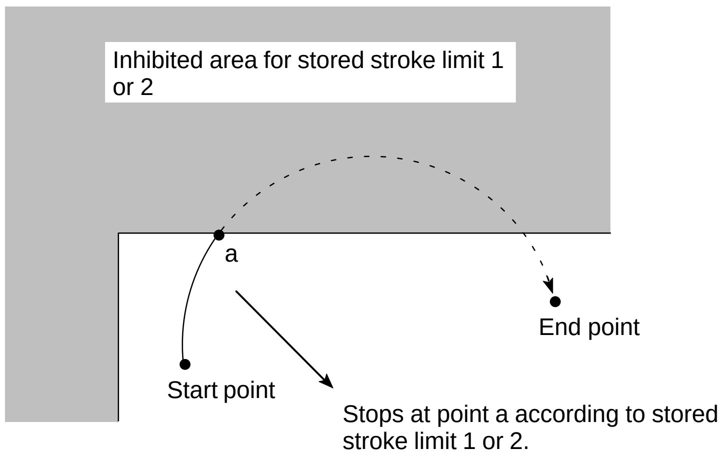
30.2.5 Stroke Limit Check Before Move

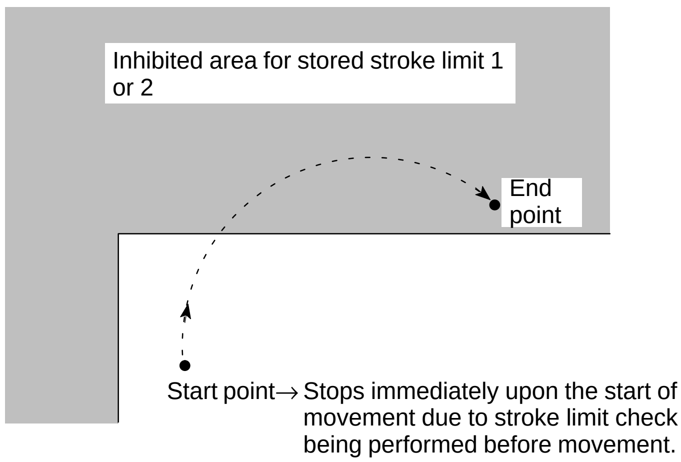
This function calculates the movement end point at the start of movement in a block, during automatic operation, based on the current machine position and the specified amount of travel, to check whether the end point falls within the inhibited area for stored stroke limit 1, 2, or 3. If the end point falls within an inhibited area, movement for that block is stopped immediately upon the start of movement and an alarm is issued.

NOTE

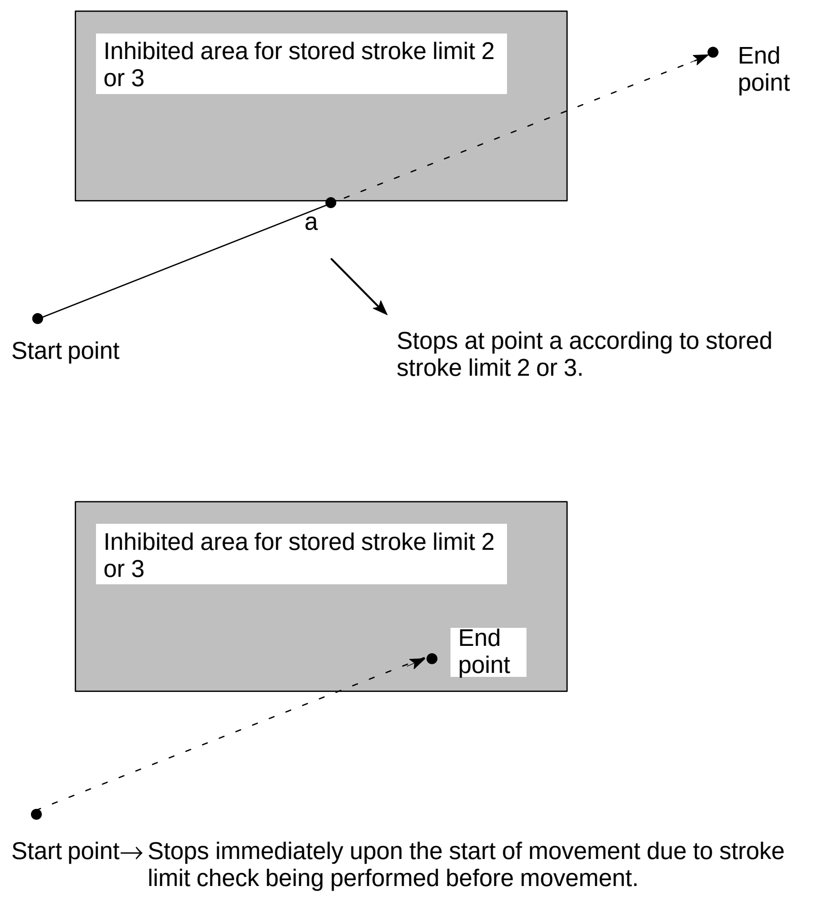
This function checks only whether the end point falls within an inhibited area. It does not check whether the tool passes through an inhibited area between the start and end points. However, an alarm is issued upon a tool's entering an inhibited area according to stored stroke limit 1, 2, or 3.

Example 1)





Example 2)



30.2.6 Stroke Limit Externally Setting

When a new tool is mounted, position the tip of the tool on the two corners of the limit area, and specify the machine coordinates of the corners in the parameters for stroke limit 1. The machine coordinates are stored in the CNC as the limit positions. Then input signals for setting the stroke limit. Stroke limit setting signals are provided for each axis and each direction. Checking of the stroke limit can also be selected by turning on or off the limit release signal common to all axes.

30.2.7 Chuck and Tail Stock Barrier

T series

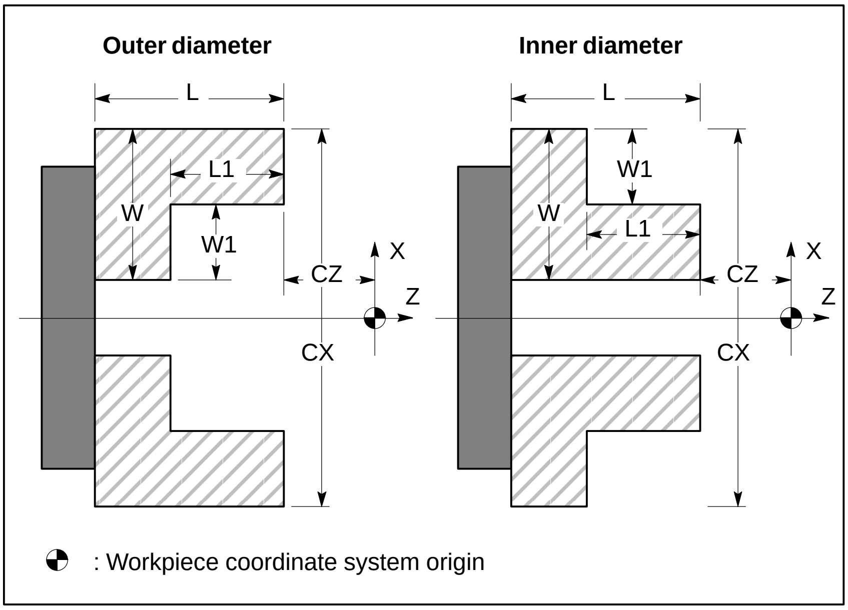
It is used for checking the interference between the chuck and tail stocks and preventing the damage of machines.
Set the area of entry prohibition from the exclusive setting screen according to the shapes of chuck and tail stocks.
When a tool enters the area of entry prohibition during cutting, the travelling of tool is stopped and an alarm message is displayed. The tool can be escaped from the prohibition area by moving in the opposite direction to that on entry. The Yes/No of this function is selected by the G22 (Stored stroke limit on). G23 (Stored stroke limit off), and signal of machine side.

G code	Tail stock barrier select signal	Chuck barrier	Tail stock barrier
G22	0	Yes	Yes
G22	1	Yes	No
G23	Irrelevant	No	No

The shape of chuck or tail stock is defined on the setting screen.

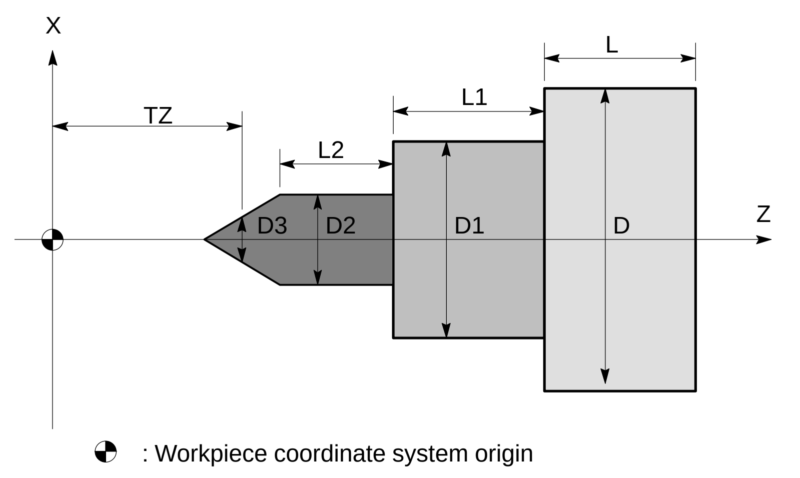
Explanations

- **Dimension definition of chuck**



Symbol	Description
L	Length of chucking claw
W	Size of chucking (radius input)
L1	Holding length of chucking claw
W1	Holding difference of chucking claw (radius input)
CX	Position of chuck (X axis)
CZ	Position of chuck (Z axis)

- **Dimension definition of tail stock**



Symbol	Description
L	Length of tail stock
D	Diameter of tail stock (Diameter input)
L1	Length of tail stock (1)
D1	Diameter of tail stock (1) (Diameter input)
L2	Length of tail stock (2)
D2	Diameter of tail stock (2) (Diameter input)
D3	Hole diameter of tail stock (Diameter input)
TZ	Position of tail stock (Z axis)

NOTE

This function cannot be used together with stored stroke check 2 or 3.

30.3 INTERLOCK

30.3.1 Interlock for Each Axis

Axis feed specified to each axis can be stopped separately. If interlock is specified to any of the moving axis during cutting feed, all axes of the machine movement will decelerate to a stop. When interlock signal is reset, the moving starts.

30.3.2 All Axes Interlock

Feed of all axes can be inhibited. When all axes interlock is commanded during move, it decelerates and stops. When all axes interlock signal is reset, the moving restarts.

30.3.3 Interlock for Each Axis Direction

Feeding of a specific axis in a specific direction can be inhibited independently of other axes. If the interlock signal is input to any of the axes during a cutting feed operation, all axes decelerate and come to a stop. When the interlock signal for each axis direction is released, the axes start moving again.
This function is usable for both the manual and automatic operations of the M series. For the T series, the DAU parameter (bit 4 of parameter No. 3003) can be used to specify whether to make the function usable only for the manual operation or for both the manual and automatic operations.

> **NOTE**
> To enable axis direction–specific interlocking for the automatic operation of the T series, set the DAU parameter (bit 4 of parameter No. 3003).

30.3.4 Start Lock

T series

Feeding of all axes can be inhibited only during automatic operation. When the start lock signal is input while the axes are moving, all axes decelerate and come to a stop. When the start lock signal is released, the axes start moving again.

30.3.5 Block Start Interlock

During automatic operation, the start of the next block can be disabled. A block that has already started is executed as is till its end.
When the block start interlock is released, the execution of the next block starts.

30.3.6 Cutting Block Start Interlock

During automatic operation, the start of a block containing a move command other than positioning can be disabled.
When the cutting block start interlock is released, the execution of the next block starts.
When spindle rotation is specified or when the spindle speed is changed, the cutting block start interlock can be applied until the spindle reaches a target speed so that the next cutting block can be executed at the target spindle speed.

30.4 EXTERNAL DECELERATION

Feed rate can be decelerated by an external deceleration signal from the machine side. A feed rate after deceleration can be set by parameter. External deceleration is prepared every axis and every direction.
When the tool is to be moved in the reverse direction, futile time may not be wasted since no external deceleration is applied.
By setting the corresponding parameter, whether to make this signal effective only for rapid traverse mode or for all feed modes can be specified for each axis and for each direction.
This function allows the maximum of valid strokes and keeps shock to the machine to a minimum, to stops at stroke end.

30.5 UNEXPECTED DISTURBANCE TORQUE DETECTION

When a cutting tool collides with the machine body or is damaged during cutting, the load torque applied to the motors is larger than during normal feeding or cutting. The abnormal load detection function calculates the load torque and transfers the value from the CNC to the PMC. If the load torque is larger than the value set in a parameter, the function stop the motor or reverses the motor rotation to retract the tool by the distance set in a parameter. In this way, damage to the machine is prevented. (The motor rotation reverse function is available only for the servo motor axes.)
Unexpected disturbance torque detection can be disabled only for a specific axis.

31 STATUS OUTPUT

31.1 NC READY SIGNAL

This signal is sent to the PMC when NC power is on and control becomes possible. Sending of this signal will be stopped when NC power is turned off.

31.2 SERVO READY SIGNAL

This signal is sent to the PMC when the servo system becomes operatable. Axes necessary to be braked must be braked when this signal is not sent.

31.3 REWINDING SIGNAL

This signal shows that tape reader or main program in memory is rewinding.

31.4 ALARM SIGNAL

This signal is transmitted when the NC comes under an alarm status.

31.5 DISTRIBUTION END SIGNAL

This signal is sent out when pulse distribution of the M, S, T, or B functions has ended, so that they can be used after move of the commanded block ends.

31.6 AUTOMATIC OPERATION SIGNAL

This signal is sent out when it is under automatic operation.

31.7 AUTOMATIC OPERATION START SIGNAL

This signal is sent out when automatic operation is being activated.

31.8 FEED HOLD SIGNAL

This signal is sent out when automatic operation is held by feed hold.

31.9 RESET SIGNAL

This signal is sent out to show that the NC has been reset.

31.10 IN–POSITION SIGNAL

This signal shows that an axis is under in–position status. This signal is output for all axes.

31.11 MOVE SIGNAL

This signal shows that an axis is moving. This signal is sent out for every axis.
This move signal can be combined with the interlock signal to automatically clamp and unclamp the machine, or control on/off of the lubricating oil.

31.12 AXIS MOVE DIRECTION SIGNAL

This signal is output to show move direction of each axis. This signal is output for each axis.

31.13 RAPID TRAVERSING SIGNAL

This signal shows that the move command is done under rapid traverse.

31.14 TAPPING SIGNAL

This signal is output to show that the machine is under tapping mode (G63 for M series) or tapping cycle (G74, G84 for M series), (G84, G88 for T series) is under operation.

31.15 THREAD CUTTING SIGNAL

This signal shows that the machine is under thread cutting mode (G33) or thread cutting cycle (T series).

31.16 CONSTANT SURFACE SPEED CONTROL SIGNAL

This signal shows that the machine is under constant surface speed control mode (G96).

31.17 INCH INPUT SIGNAL

This signal shows that input is done under inch input mode (G20).

31.18 DI STATUS OUTPUT SIGNAL

To inform the exterior of the states of software operator's panel, which are set via CRT/MDI, and machine operator's panel, following DI state output signals are sent.

- Mode-select check signal
- Single-block check signal
- Manual absolute on/off check signal
- Dry-run check signal
- Machine-lock check signal
- Auxiliary-function-lock check signal
- Optional block-skip check signal
- Mirror-image check signal

31.19 POSITION SWITCH FUNCTION

The position switch function outputs a signal to a specified controlled-axis when the machine coordinates of the controlled-axis are within the range specified by the corresponding parameter.
The parameter specifies an arbitrary controlled-axis and the operating range (machine coordinates) within which the position switch signal is output.
Up to sixteen position switch signals can be output.

32 EXTERNAL DATA INPUT

The external data input is as follows.

- External tool compensation
- External program number search
- External workpiece coordinate system shift
- External machine zero point shift
- External alarm message
- External operator message
- Substitution of the number of machined parts and number of required parts

32.1 EXTERNAL TOOL COMPENSATION

The tool compensation value for the offset number specified in the program can be externally modified.
The input signal designates whether the input tool offset amount is:

- Absolute or incremental
- Geometry offset or tool wear offset
- Cutter radius compensation amount or tool length compensation amount

It the machine is equipped with automatic measurement devices of tools and workpiece, error can be input to the NC with this function.
External tool compensation amount range is:
0 to ±7999
in least command increment.

32.2 EXTERNAL PROGRAM NUMBER SEARCH

A program number from 1 - 9999 can be given from outside to the NC to call the corresponding program from the NC memory.
In machines with automatic loading function of various workpiece, this function can be used to automatically select and execute program suitable to the workpiece.
With bit 3 (ESC) of parameter No. 6300, the external program number search function can also be cancelled by a CNC reset operation.

32.3 EXTERNAL WORKPIECE COORDINATE SYSTEM SHIFT

The work coordinate system can be shifted for the shift amount given from outside.
The shift amount specified by an input signal is set as an external offset value for workpiece zero points by which the workpiece coordinate system shifts. The shift amount is an absolute value, not an incremental value.
The shift amount range is :
0 to ±7999
in least command increment.

32.4 EXTERNAL MACHINE ZERO POINT SHIFT

The machine coordinate system is compensated by shift amount given from outside. This shift amount always take absolute value; never an increment value.
The shift amount range is:
0 to ±9999
in detection unit.
When shift amount is input, the actual machine move distance is the difference between the previous offset amount and current offset amount.
This function is used t compensate the machine coordinate system error caused by mechanical deformation.

32.5 EXTERNAL ALARM MESSAGE

By sending alarm number from outside, the NC is brought to an alarm status; an alarm message is sent to the NC, and the message is displayed on the screen of the NC. Reset of alarm status is also done with external data.
Up to 4 alarm numbers and messages can be sent at a single time. Alarms 0 to 999 can be sent. To distinguish these alarms from other alarms, the CNC displays them by adding 1000 to each alarm number. The messages of up to 32 characters can be sent together with an alarm.

32.6 EXTERNAL OPERATOR'S MESSAGE

Message to the operator is given from outside the NC, and the message is displayed.
The message is sent after a message number 0 to 999. Either a message consisting of up to 255 characters or up to four messages each consisting of up to 63 characters can be displayed at the same time by parameter setting.
The message numbers 0 to 99 are displayed along with the message.
To distinguish these alarms from other alarms, the CNC displays them by adding 2000 to each alarm number. When a message from 100 to 999 is displayed, the message number is not displayed; only its text is displayed.
An external data will clear the operator messages.

32.7 SUBSTITUTION OF THE NUMBER OF REQUIRED PARTS AND NUMBER OF MACHINED PARTS

The number of required parts and the number of machined parts can be preset externally. Values from 0 to 9999 can be preset.

33 KEY INPUT FROM PMC (EXTERNAL KEY INPUT)

When the PMC inputs the code signal corresponding to a key on the MDI panel to the CNC, the code signal can be input in the same way as with actual operation of the key on the MDI panel. For example, this function is usable in the following case:
After allowing to travel the tool at an arbitrary machining position by using the playback function (option), when to store its positions as the program command, X, Y, Z, <INSERT>, etc. must be input via key operations. However, these operations can be realized simply by depressing a switch on the operator’s panel at the machine side.
When the switch is pressed, the PMC inputs code signals corresponding to keys X, Y, Z, and <INSERT> to the CNC. This produces the same results as with actual key operations.

34 FUNCTIONS FOR PUNCH PRESS

34.1 PREPARATORY FUNCTIONS

The following G codes are provided for Series 0*i*–PC. The G codes are classified into two: A and B. One of the G code types can be selected using a parameter. In this manual, G code system A is assumed.

G code list (1/2)

System A	System B	Group	Meaning	
G00	G00	01	Positioning (Rapid traverse)	
G01	G01		Linear interpolation (Cutting feed)	
G02	G02		Circular interpolation/Helical interpolation (CW)	
G03	G03		Circular interpolation/Helical interpolation (CCW)	
G04	G04	00	Dwell	
G08	G08		Advanced preview control	
G09	G09		Exact stop	
G10	G10		Programmable data input	
G11	G11		Cancel programmable data input mode	
G17	G17	02	XpYp plane	where Xp: X–axis or an axis parallel to it Yp: Y–axis or an axis parallel to it Zp: Z–axis or an axis parallel to it
G18	G18		ZpXp plane	
G19	G19		YpZp plane	
G20	G20	06	Input in inch	
G21	G21		Input in mm	
G22	G22	04	Stored stroke limit function on	
G23	G23		Stored stroke limit function off	
G26	G26	00	Bolt hole circle	
G28	G50		Automatic reference point return	
G32	G32		Automatic safety zone setting	
G33	G33		Skip function	
G38	G38		Bending compensation X	
G39	G39		Bending compensation Y	
G40	G40	07	Cutter compensation cancel	
G41	G41		Cutter compensation left	
G42	G42		Cutter compensation right	
G40.1 (G150)	G40.1 (G150)	19	Normal direction control canceled	
G41.1 (G151)	G41.1 (G151)		Left–side normal direction control turned on	
G42.1 (G152)	G42.1 (G152)		Right–side normal direction control turned on	
G50	G34	11	Scaling off	
G51	G35		Scaling on	
G52	G93	00	Local coordinate system setting	
G53	G53		Machine coordinate system selection	

G code list (2/2)

System A	System B	Group	Meaning
G54	G54	14	Workpiece coordinates system 1 selection
G55	G55		Workpiece coordinates system 2 selection
G56	G56		Workpiece coordinates system 3 selection
G57	G57		Workpiece coordinates system 4 selection
G58	G58		Workpiece coordinates system 5 selection
G59	G59		Workpiece coordinates system 6 selection
G61	G61	15	Exact stop mode
G62	G62		Automatic corner override
G64	G64		Continuous cutting mode
G65	G95	00	Custom macro simple call
G66	G96	12	Custom macro modal call
G67	G97		Custom macro modal call cancel
G68	G68	00	Circular nibbling
G69	G69		Linear nibbling
G70	G70		Positioning & press off
G72	G72		Standard point command
G73	G75		Multi-piece machining command X
G74	G76		Multi-piece machining command Y
G75	G27		Automatic repositioning
G76	G28		Line at angle
G77	G29		Arc
G78	G36		Grid I
G79	G37		Grid II
G84	G84	16	Coordinate system rotating on
G85	G85		Coordinate system rotating off
G86	G86	00	Share proof
G87	G87		Square
G88	G78		Radius
G89	G79		Cut at angle
G90	G90	03	Absolute command
G91	G91		Incremental command
G92	G92	00	Coordinate system setting
G98	G98		Coordinate system setting (Multi-piece machining)

34.2 PRESS FUNCTIONS

34.2.1 One–cycle Press

In blocks which perform positioning (G00) along the X- or Y-axis in rapid traverse mode, a press start signal is sent to the press after positioning is completed, thus enabling punching. Note however that no press start signal is output in the following cases:

(1) When MDI mode has been selected

(2) When a M-code has been issued

(3) In a block between an M-code to clamp the workpiece and an M-code to unclamp the workpiece

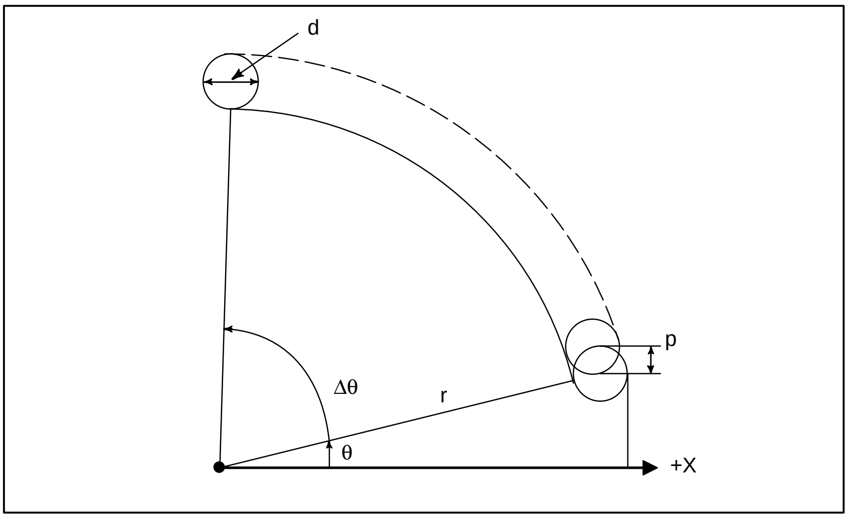
> **NOTE**
> Before positioning is completed, a press start signal may be output depending on parameter settings.

34.2.2 Continuous Press (Nibbling)

The term nibbling refers to repeated punching without bringing the press to a halt. A continuous-press-in-process signal is output in nibbling blocks.

Circular Nibbling (G68) And Linear Nibbling (G69)

(1) The following command specifies circular nibbling:
G68 I $\underline{r}$ J$\underline{\theta}$ K$\underline{\Delta\theta}$ P $\underline{d}$ Q $\underline{p}$;



r : Radius

θ : Angle between the line from the origin to the punch start point and the positive X-axis

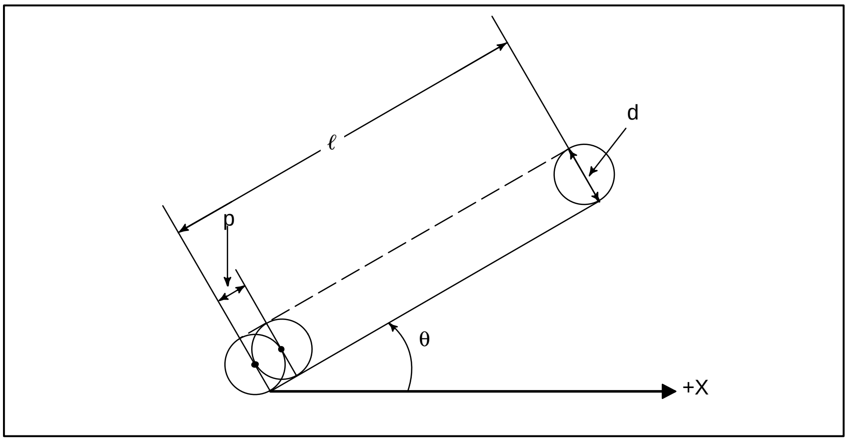
Δ θ : Angle between the line from the origin to the punch start point and the line from the origin to the punch end point

d : Tool diameter

p : Nibbling pitch

(2) The following command specifies linear nibbling:

G69 I ℓ Jθ P d Q p ;



ℓ : Length of the line

θ : Angle between the line and the positive X-axis

d : Tool diameter

p : Nibbling pitch

Nibbling Mode (M–code)

Nibbling can be performed in a block between an M-code for nibbling (M12) and an M-code for nibbling cancel (M13). These M-codes are specified by parameters.

Linear nibbling can be done by commanding G01 in the nibbling mode, while circular nibbling can be done by commanding G02 and G03 in the nibbling mode.

The tool diameter cannot be offset by G01, G02, G03 commands. Accordingly, these commands are used together with cutter compensation commands (G40, G41, G42) when nibbling is done by offsetting a continuous straight line or circular arc leftwards or rightwards by the tool diameter.

(1) Straight line

G01X —— Y —— Q —— ;

The end point is designated by address X, Y, while the pitch is designated by address Q.

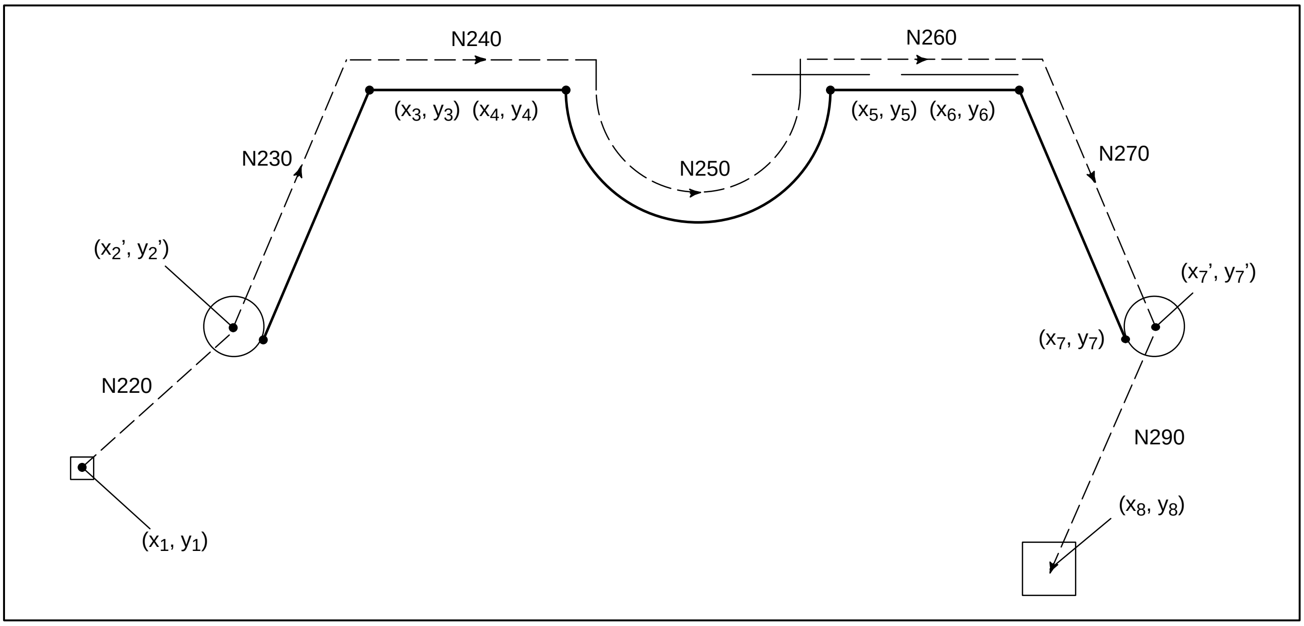
(2) Circular arc

$$\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\_\_ \ Y\_\_\_ \quad \begin{Bmatrix} I\_\_\_J\_\_\_ \\ R\_\_\_ \end{Bmatrix} Q\_\_\ ;$$

The end point is designated by address X, Y, the radius of circular arc is designated by address I, J, or R and also the pitch is designated by address Q.

Q: Pitch

N200G00G90X x_1 Y y_1 ;
N210M12;
N220G41X x_2 Y y_2 T02D02 ;
N230G01X x_3 Y y_3 Q ______ ;
N240X x_4 Y y_4 ;
N250G03X x_5 Y y_5 I ______ J ______ ;
N260G01X x_6 Y y_6 ;
N270X x_7 Y y_7 ;
N280M13;
N290G40G00X x_8 Y y_8 T03 ;



The G40, G41, and G42 codes function as follows.

G code	Function
G40	Cutter compensation cancel
G41	Leftward offset for moving direction of tool
G42	Rightward offset for moving direction of tool

Assume that diameter (radius) of the tool selected by T02 is preset to offset NO.02.

34.2.3 Manual Press

Pressing a button on the press machine can output a press-in-progress signal. It is possible to specify whether punching is to be performed on one cycle only or to be continued while the button is pressed.

34.2.4 Positioning and Press off (G70)

G70 specifies rapid traverse for positioning. A press start signal is not output after positioning is completed.

Format

G70IP -- ;

34.2.5 Setting for Press Start Signal

The following parameters can be used to adjust the press start signal output timing so that punching is smooth.

(1) Punching specification for forming mode

(2) Setting up an upper limit to the positioning offset for press start signal output

(3) Timer setting (early PF) for outputting a press start signal before positioning is completed

(4) Timer setting for executing the next block before press is completed

34.2.6 Press Start Lock

It is possible to inhibit the press start signal from being output.

34.2.7 Press Start Wait

It is possible to defer the output of the press start signal until the press is ready to respond.

34.2.8 Changeable Nibbling Mode 2 Steps

Up to two nibbling pitches can be specified according to an external signal or an M-code in a parameter. This function can be used to select a maximum traveling pitch for a machine in which the speeds of a flywheel can be switched or according to the thickness of raw material.

34.2.9 External Motion Function

In a block in which press start signal output is specified (except for nibbling), an external operation signal, rather than the press start signal, can be output as required by an external signal. This function can be used for tapping.

34.3 SWITCHING RAPID TRAVERSE FEEDRATES, TIME CONSTANTS, AND/OR SERVO LOOP GAINS ACCORDING TO THE POSITIONING DISTANCE

For rapid traverse, the rapid traverse feedrates, time constants, and servo loop gains can be switched according to the positioning distance set up by a parameter for each axis. Use of this function can improve positioning accuracy.

X– and Y–axes : Rapid traverse feedrates, time constants, and/or servo loop gains are switched according to the positioning distance (up to 7 steps).
Servo loop gains for rapid traverse and cutting feed are switched

T–axis : Rapid traverse time constant and servo loop gains are switched according to the positioning angle (up to 3 steps).

C–axis : Rapid traverse feedrates and time constants are switched according to the positioning angle (up to 3 steps).

34.4 POSITIONING TIME CONSTANT CONTROL

For automatic rapid traverse, positioning along the X– and Y–axes can be performed exactly in the specified time regardless of the positioning distance. Use of this function for nibbling can smooth punching regardless of what the nibbling pitch is, because positioning can be done exactly in the constant time. (2 steps)

34.5 PATTERN FUNCTION

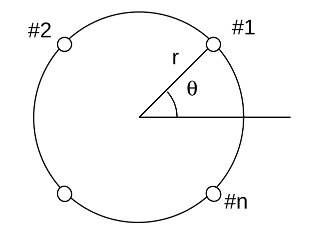
It is possible to punch out at two or more positions according to a certain form by the command of one block. Whenever it is positioned at each position in the rapid traverse rate, the press start signal is outputted.

CAUTION
M code and pattern function can not be specified in the same block.

34.5.1 Bolt Hole Circle (G26)

Format

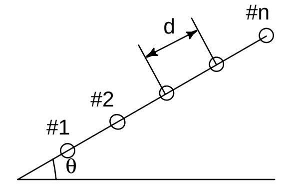
G26I r J θ K n ;



34.5.2 Line at Angle (G76)

Format

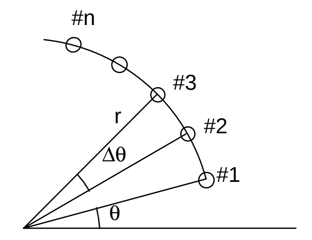
G76I d J θ K n ;



34.5.3 Arc (G77)

Format

G77I r J θ P Δθ K n ;

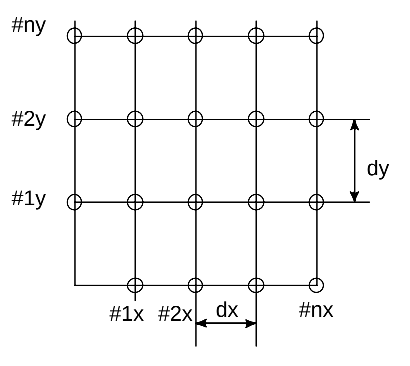


34.5.4 Grid (G78, G79)

Format

G78I dx P nx J dy K ny ;
G79I dx P nx J dy K ny ;

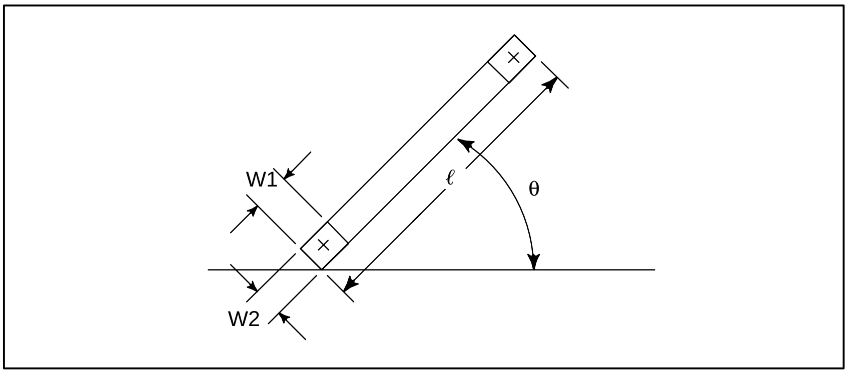
G78 specifies punching from X–axis direction.
G79 specifies punching from X–axis direction.



34.5.5
Share Proof (G86)

Format

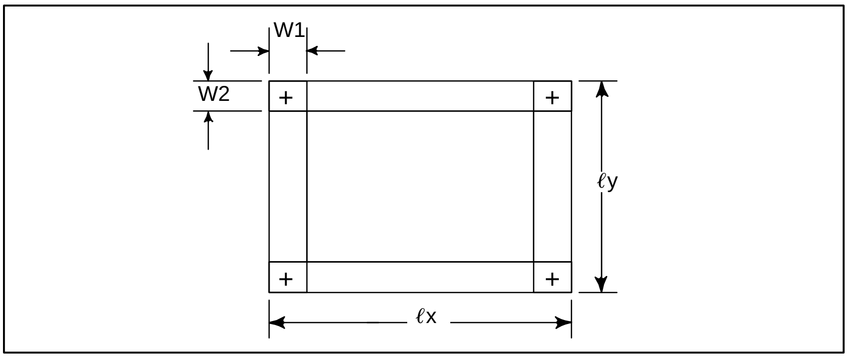
G86I $\underline{\ell}$ J $\underline{\theta}$ P $\underline{W1}$ Q $\underline{W2}$;



34.5.6
Square (G87)

Format

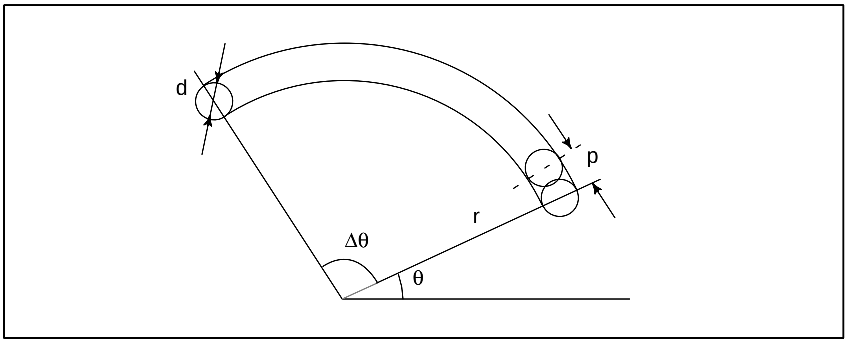
G87I $\underline{\ell x}$ J $\underline{\ell y}$ P $\underline{W1}$ Q $\underline{W2}$;



34.5.7 Radius (G88)

Format

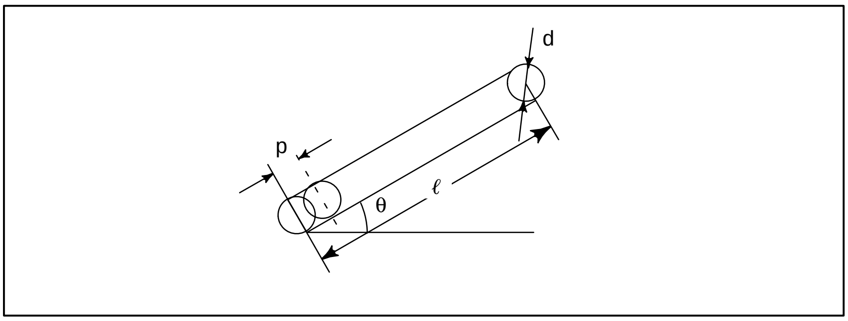
G88I $\underline{\ell x}$ J$\underline{\theta}$ K$\underline{\Delta\theta}$ P$\underline{d}$ Q$\underline{p}$;



34.5.8 Cut at Angle (G89)

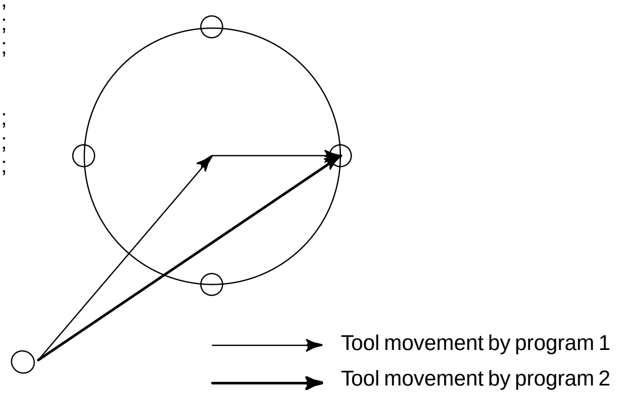
Format

G89I $\underline{\ell}$ J$\underline{\theta}$ P$\underline{d}$ Q$\underline{p}$;



34.6 PATTERN BASE POINT COMMAND (G72)

```
Program 1
G90X100Y100  ;
G70X200Y200  ;
G26I50JOK4   ;

Program 2
G90X100Y100  ;
G72X200Y200  ;
G26I50J0K4   ;
```



G72X– Y– ;

An under–mentioned coordinates values can be instructed by an above–mentioned command.

(1) Center point of Bolt hole circle (G26), Arc (G77), Circular nibblint (G68), Radius (G88)

(2) Starting point of the line at angle (G76), grid (G78, G79), linear nibbling (G69), cut at angle (G89), share proof (G86), and square (G87)

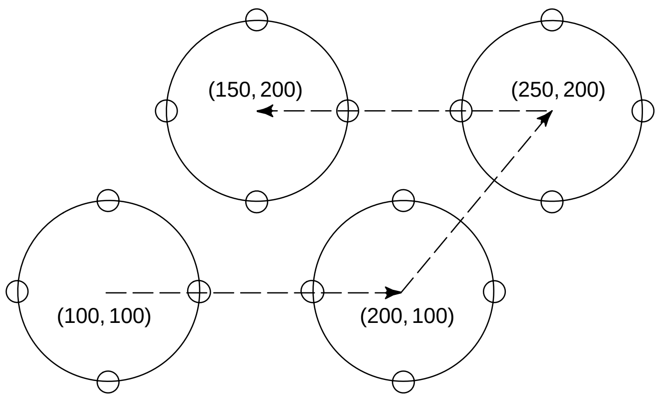
G72 only designates the pattern base point; it does not move a tool.

CAUTION

M code, T code, and S code can not be specified in the same block as G72.

34.7 MEMORY AND CALL BY A/B MACRO

```
        •
        •
        •
G72X100.Y100. ;
A2G26I50.J0K4 ;← Designated for storage after execution
G72X200.Y100. ;
B2 ;            ← Recall of pattern A2
G72X250.Y200. ;
B2 ;
G72X150.Y200. ;
B2 ;
        •
        •
        •
```

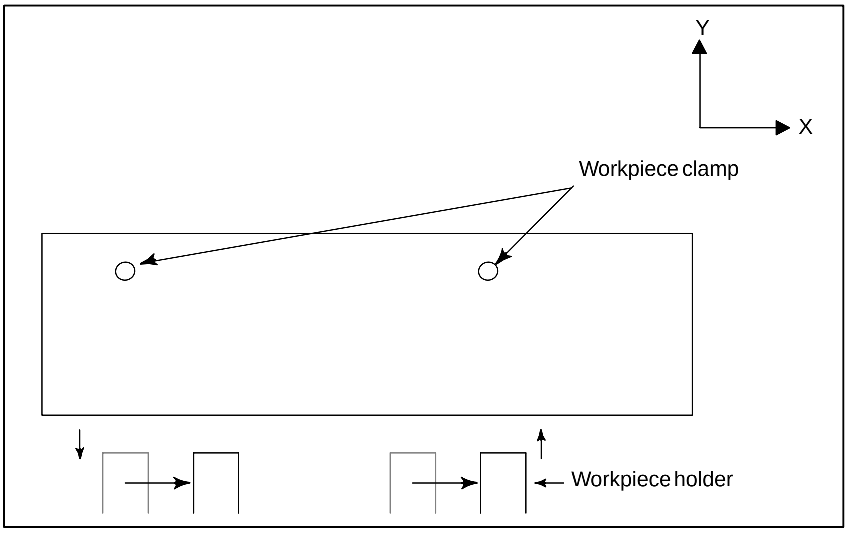


When it is desired to repeatedly use a pattern with the same figure among the pattern functions, it is possible to store the pattern with a number and access it whenever necessary.

(1) Storage A1 – A5
A pattern function is executed and stored when address A (1 to 5) is specified before a G code of pattern function.
A2G26I50.J04K; G26I50.J04K is stored in storage A2 of the patternfunciton.

(2) Call B1 – B5
The pattern function memorized in address A can be called by address B (1 to 5) command.
B2; – G26I50.J0K4 is called and executed.
Blocks (up to 192 characters per block) can be stored and called except for the pattern function.

34.8 AUTOMATIC REPOSITIONING (G75)



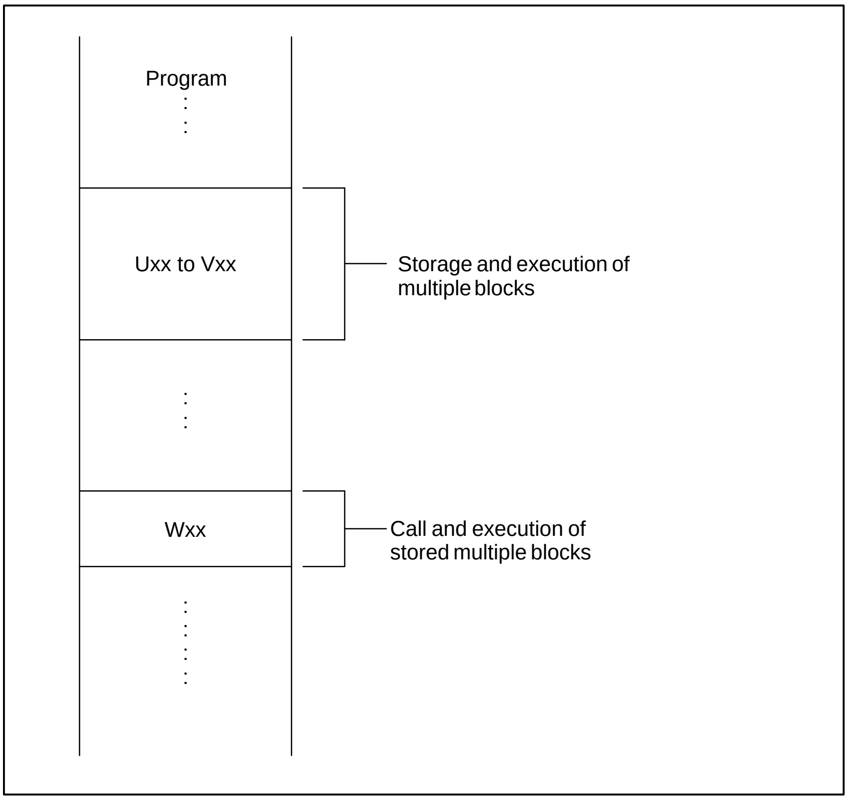
G75X– ;

A series of the following commands can be given, using G75X_ ; command.

(1) M10 ;	:	Workpiece clamp
(2) G70G91YyR ;	:	Escape of Y axis
(3) G70G91X–X ;	:	Shift of X axis
(4) G70G91Y–yR ;	:	Return of Y aixs
(5) M11 ;	:	Workpiece unclamp

yR, M10, and M11 can be set by parameters. The shift amount of the X and Y axis in G75 is not added to the absolute coordinate value.

34.9 U/V/W MACRO FUNCTION



The macro function can register a plurality of blocks as one macro and call them whenever necessary.

34.9.1 Storage of Macro (U, V Command)

When 2–digit figures 01 to 89 following address word U are added prior to the instructions of plural blocks to be stored and the same figure is used for the last figure following address word W, plural blocks instructed for that period are stored in memory.
The 2–digit figure following address U or V is called the macro number. When macro number is 01 – 59, the blocks between U and V are memorized accompanying the execution of the command.
On the contrary, when the Macro No. is 60 to 89, the command is not executed and only the storage operation is performed. It is possible to normally give instructions to all blocks inserted between U and V except instructions for Macro storage by other Macro Nos.

34.9.2 Macro Call (W Command)

It is possible to call a plurality of blocks stored previously as Macro by giving instructions with a 2–digit numeral following address W.

34.9.3 Multiple Call of Macro

Up to triple accessing is possible: another macro is called from a given macro, which was called from still another macro.

34.9.4 Storage Capacity of Macro

The number of characters which can be stored for each macro having a macro number between 01 to 89 can be specified with a parameter (up to 2700 characters).

34.9.5 Storage and Call of Plural Macro (Macro No.90 to 99)

Plural macros can be stored as one macro by using the macro number 90–99 and can call it.
The number of macros which can be stored on each of the macro number 90–99 is up to 15 or less.

34.10 MULTI–PIECE MACHINING

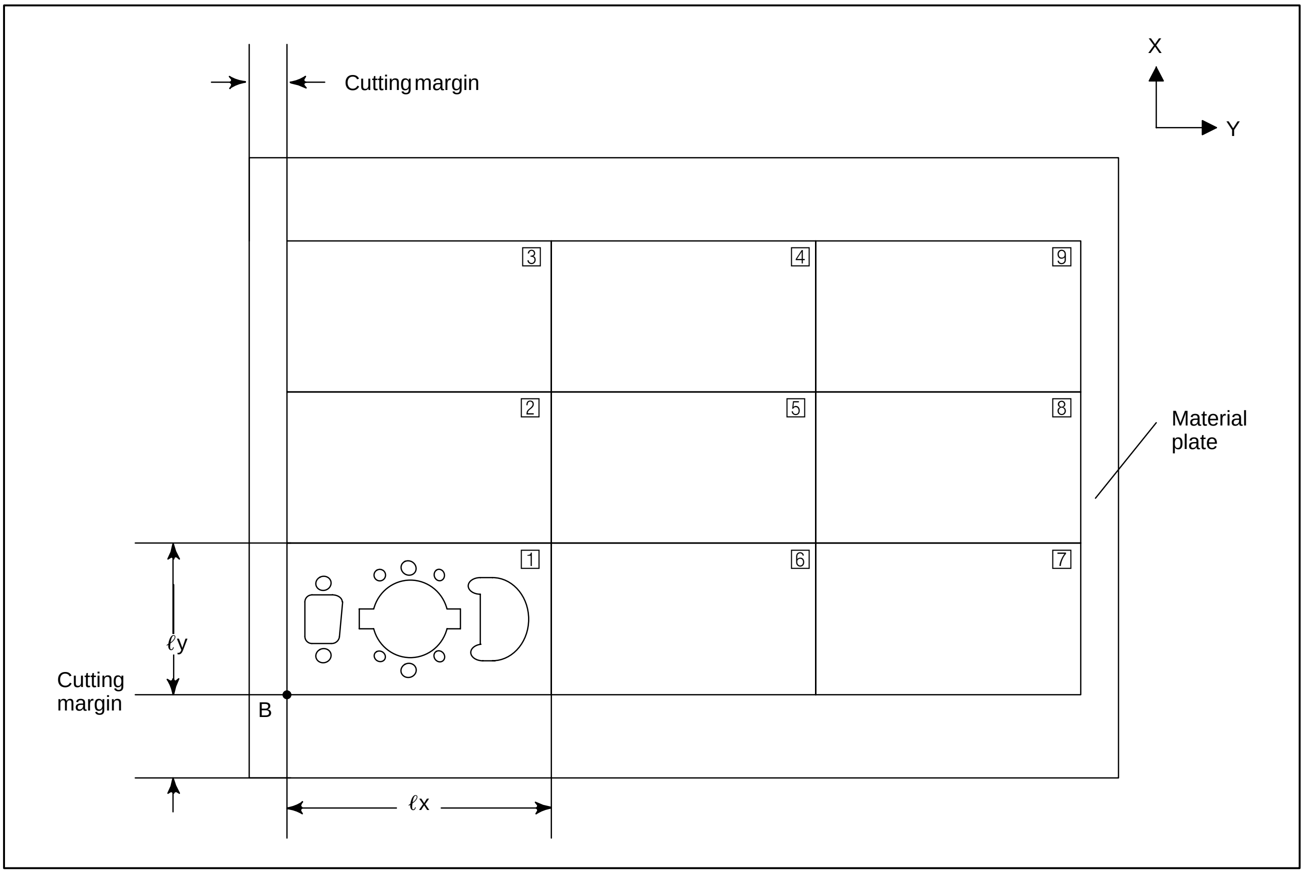


Fig. 34.10

It is a function for the multi–piece machining by which several identical product boards can be produced from one blank board with an easy command.
A simple setting method allows so–called trial punching by the command of the multi–piece machining function, thus facilitating a machining command tape check prior to the whole machining.

34.10.1 Base Point Command of Multi–piece Machining

Specify a command prior to the punching command for a single plate.

Format

G98X xb Y yb I lx J ly P nx K ny ;

xb : X axis coordinates values in the multi–piece machining base point (B of Fig. 13.10)

yb : Y axis coordinates values in the multi–piece machining base point (B of Fig. 13.10)

lx : X axis direction length (Give by the positive number) of one product

ly : Y axis direction length (Give by the positive number) of one product

nx : The number of products in the X axis direction (Caution)

ny : The number of products in the Y aixs direction (Caution)

CAUTION

Product board 1 (in Fig. 34.10) is not included in the number.

After a G98 command, specify the punching command for a single plate at the lower left (1 in Fig. 34.10) which performs the multi–piece machining function operation at grid intervals, using a macro function (storing a machining command between U and V commands).

- **Command method**

```
    :
G92X_Y_ ;
G98X_Y_I_J_P_K_ ;
```

Punching command for a single plate at the lower left

34.10.2 Multi–piecemachining Command

When machining commands stored by a macro function are to be called to perform the multi–piece machining function, use the following commands.

Format

G73W wn Q q ;
G74W wn Q q ;

G73 :When punching from X axis direction.
G74 :When punching from Y axis direction.
wn :Macro number
q :Machining start area specification
q=1 Start from the lower left of the sheet (1 in the fig.).
q=2 Start from the lower right of the sheet (7 in the fig.).
q=3 Start from the upper left of the sheet (3 in the fig.).
q=4 Start from the upper right of the sheet (9 in the fig.).

- **Command method**

:
G92X_Y_ ;
G98X_Y_I_J_P_K_ ;

Punching command for a single plate at the lower left
Stored in Ux–Vx

G73WxQ1 ; Execution of multi–piece machining command
M30 ;

34.10.3 Setting of Processing Method of Multi–piece Machining

The processing mehtod can be selected by the parameter setting from the external signal MDI when the processing command tape of the multi–piece machining is used for machining.

Setting 0 : Command tape for the multi–piece machining is not used.
1 : Trial punching of the multi–piece machining
2 : Remainder punching after the trial punching of the multi–piece machining
3 : All blank punching of multi–piece machining

34.10.4 Command to Restart Punching Multiple Products

By specifying a restart point by a P–code in a block containing G73/G74 (commands for punching multiple products), it is possible to restart punching multiple products at the specified restart point.

• Command format

G73 (G74) W w Q q P p ;

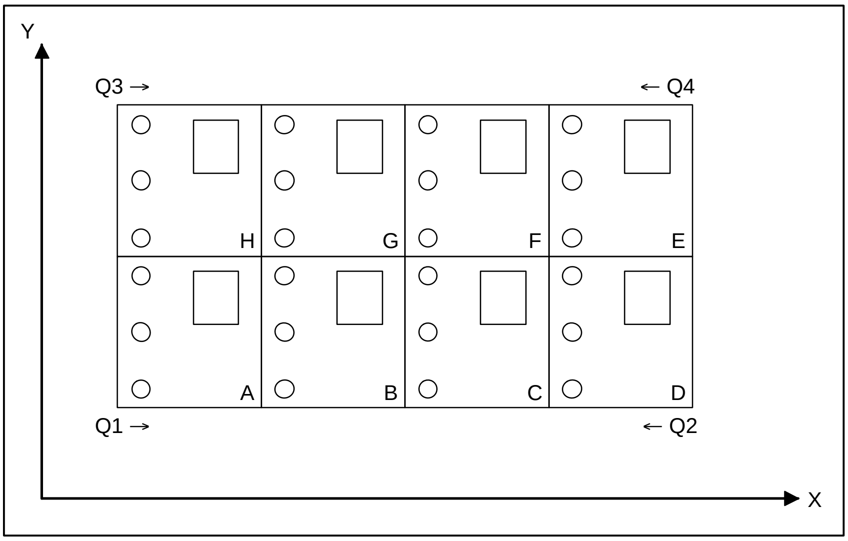
w : Macro number

q : Machining start area specification

p : Restart point

The P–code specifies a restart point with a machining sequence specified by G73/G74 or a Q–code.

Command range : 1≦p≦total number of workpieces



Examples

G98X_Y_I_J_P3K1 ;
U90 ;
:
V90 ;
G73W90Q_P5 ;

The above program performs machining as follows:

(1) Partial machining
- (a) Q1 command : Sequence of E→F→G→H
- (b) Q2 command : Sequence of H→G→F→E
- (c) Q3 command : Sequence of D→C→B
- (d) Q4 command : Sequence of B→C→D

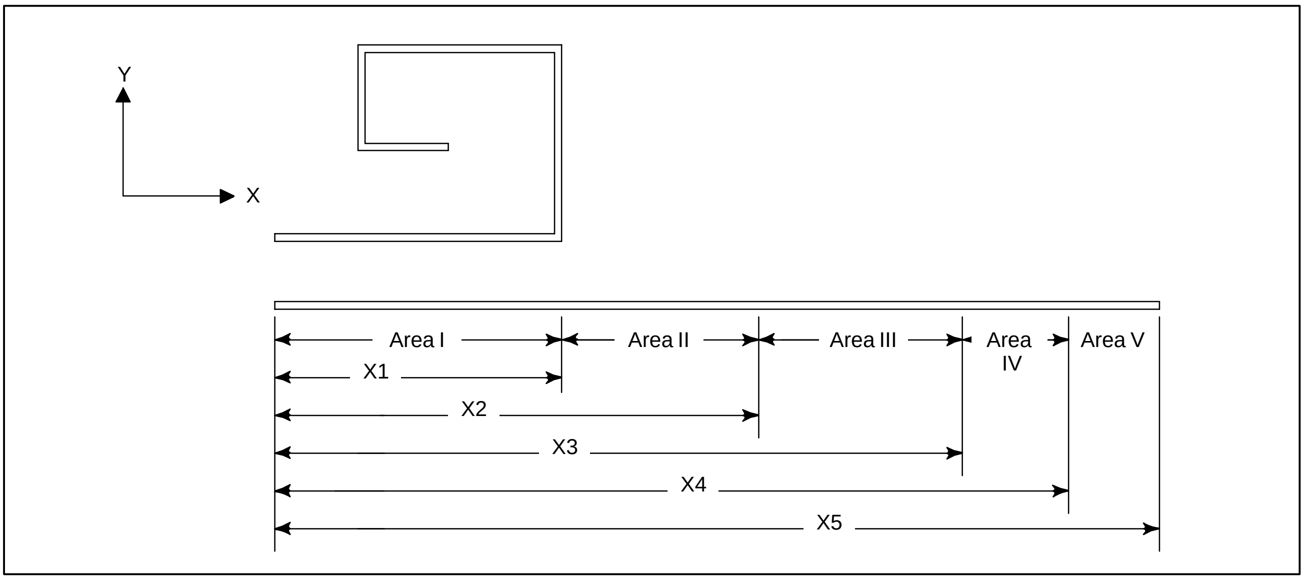
(2) Total machining
- (a) Q1 command : Sequence of E→F→G→H
- (b) Q2 command : Sequence of H→G→F→E
- (c) Q3 command : Sequence of D→C→B→A
- (d) Q4 command : Sequence of A→B→C→D

34.11 BENDING COMPENSATION (G38, G39)

The hole position gap accompanied bending is compensated and the drilling is performed.

• Program format

- Bending compensation for X axis direction
  G38I $\underline{X1}$ J $\underline{X2}$ K $\underline{X3}$ P $\underline{X4}$ Q $\underline{X5}$ R $\underline{\alpha}$;
- Bending compensation for Y axis direction
  G39I $\underline{Y1}$ J $\underline{Y2}$ K $\underline{Y3}$ P $\underline{Y4}$ Q $\underline{Y5}$ R $\underline{\beta}$;



α, β	:	These are called the benging coefficient. The compensation amount attendant on a single bending is specified.
X1 to X5	:	The distance of X and Y axes direction from the standard point is specified.
(Y1 to Y5)	:	Bending can be performed up to 4 times.

In a block after a bending command is sent, it is judged to which area automatically designated the command value of the move command belongs, thus allowing compensation in accordance with the bending frequency.

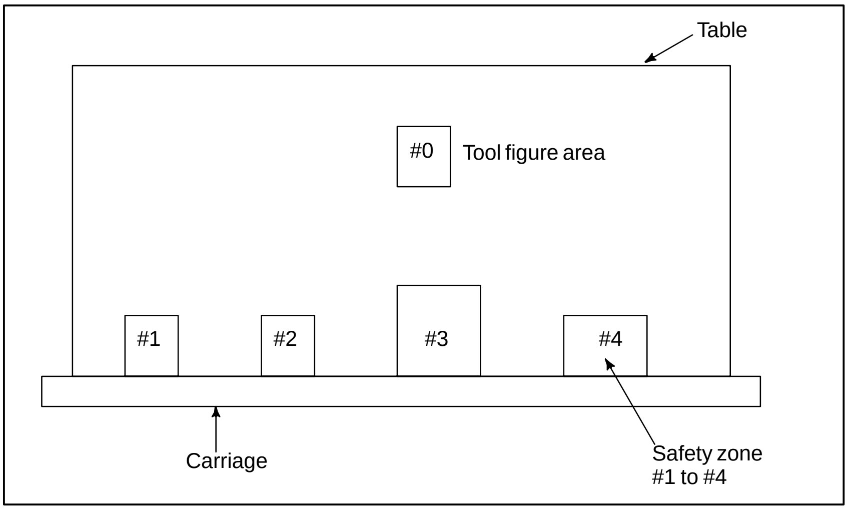
When bending compensation cancel is required.

(1) Specify the cancel of bending compensation.
G38R0 : ... (Cancel of X axis direction)
G39R0 : ... (Cancel of Y axis direction)

(2) Specify M02, M30.
It is possible to cancel even in the reset state according to reset, clear, and emergency stop.

34.12 SAFETY ZONE CHECK

This is the safety function to set the safety zone for protecting the workpiece holder that holds the workpiece set on the carriage, and disable punching in that area or forbid the tool to approach thereinto.



This function permits to set tool figure area (#0) and up to four safety zones (#1 - #4), as shown above.

34.12.1 Punch Forbidden Area and Approach Forbidden Area

The safety zone is settable in two types, punch forbidden area and approach forbidden area, that are selected by the parameter.

(1) Punch forbidden area
When the tool figure area goes into the safety zone and the punching is commanded, an alarm is given to disable punching. In the case of rapid traverse rate when the end point of positioning is in the forbidden area, an alarm is given without moving the axis. (Previous check)
In the case of move command without punching, the tool figure area can go into the punch forbidden area, but manual punching is impossible after going into this area.

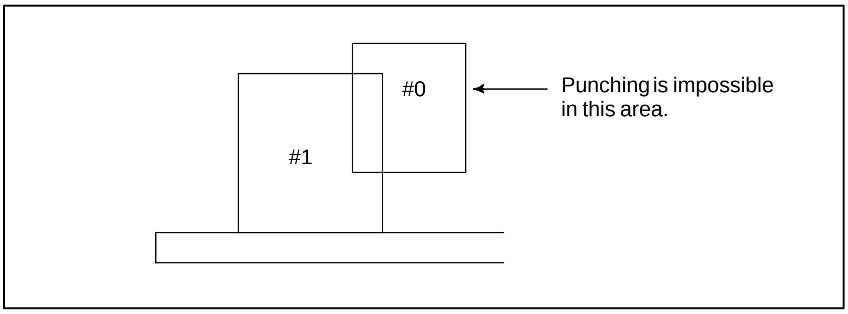


Fig. 34.12.1 (a)

(2) Approach forbidden area

The tool figure area can not go into the safety zone. When the tool figure area approaches into the safety zone by the move command, the axis is immediately stopped and an alarm is given. This is valid in either manual or automatic operation mode.

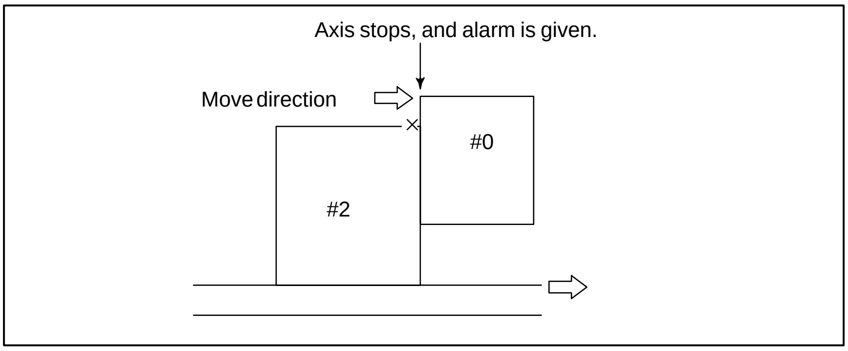


Fig. 34.12.1 (b)

34.12.2 Setting the Safety Zone

Set the machine coordinate value when the workpiece holder is positioned at the tool center (punching position) by the parameters.

34.12.3 Automatic Setting of the Safety Zone

The detector on the machine automatically detects the positions of the workpiece holders mounted on the carriage. Values representing the detected positions are then set in the safety zone parameters.

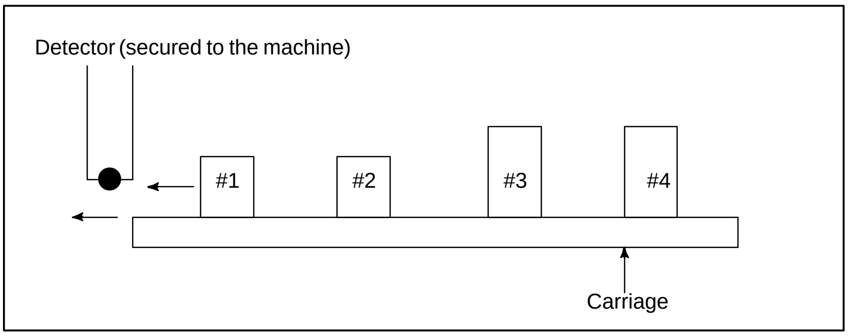


Fig. 34.12.3

The signal for detecting the position of the workpiece holders (#1 to #4) turns on and off as workpiece holders #1 to #4 pass the detector as shown in Fig. 34.12.3. The safety zone is determined from the signal. Auto detection can be instructed through the use of G32 or an external signal.

34.13 TOOL FUNCTIONS

34.13.1 T Code Output

Selection of tools can be done by commanding tool numbers with an 8–digit numeral after address T. The 8–bit numeral is output to PMC in a 32–bit binary code. This code is kept till the next T code is commanded. Maximum input digits are set by parameters.

CAUTION

1 The T–code must be issued in positioning mode (G00).
2 A program must contain this T–code in a block in which press is specified for the first time or any preceding block. If a T–code is not issued, a press start signal will not be output, and therefore the program will not advance to the next block.

34.13.2 T–axis Control

The CNC can control the turret axis (T–axis) for tool indexing as specified by a parameter. This function selects a tool by issuing a T–code and positioning the turret at a tool position specified on the tool registration menu, where up to 136 tools can be registered. The T–code for T–axis control consists of one to four digits.
Use of this function can reduce the tool replacement time largely, because it supersedes tool replacement on the machine side by using T–axis positioning.

34.13.3 Ignoring T–code

T–codes can be ignored. An external signal is used to determine whether to ignore the T–code. Once it is specified to ignore T–codes, the program operates exactly as if it contained no T–code. This function can be used, for example, to check the program by marking on a workpiece with a marking tool.

34.13.4 Tool Offset Compensation

Tool position compensation along the X– and Y–axes can be effected for tools specified by a code consisting of character T and one to four digits that follow it. The amount of compensation is specified in the least command increment defined on the tool registration menu during installation. (Up to 136 tools, values: Positive or negative number consisting of eight digits)

CAUTION

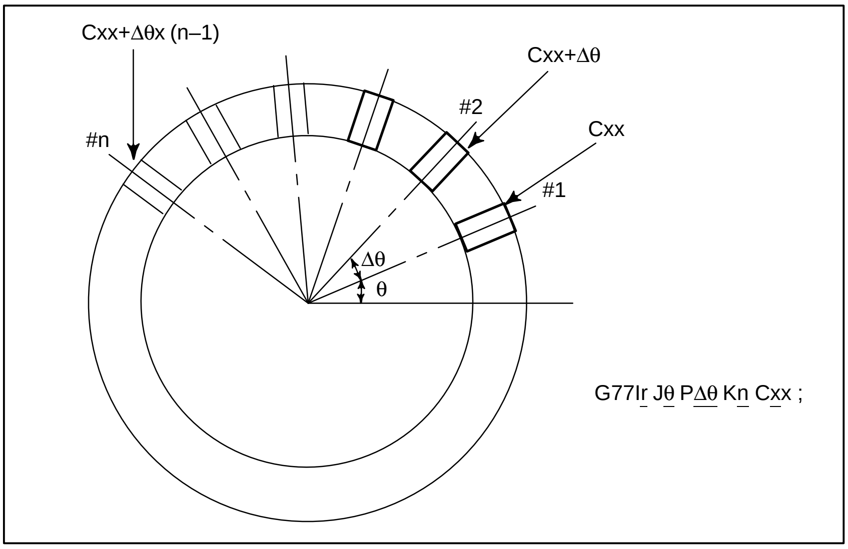
If an inch/metric switch–over option is specified, and the measurement unit used in the machine is different from that used in the input system, a maximum possible error is the sum of half the least input increment and half the least command increment. This error will not be accumulated.

34.13.5 Tool Life Management

Set the maximum number of punches for each tool, on the tool registration screen. When the number of executed punches for the selected tool exceeds the maximum number of punches for that tool, the tool life reached signal PTLCH is output. This signal is used to issue an alarm on the PMC or request the operator to change the tool.

The number of executed punches and maximum number of punches can be set and displayed by using the [TOOL] soft key on the <OFFSET/SETTING> screen, and the [TOOL LIFE] screen.

34.14 C AXIS CONTROL



The angle position of the die (Tool) can be altered by the command from tape, memory, and MDI.
In the past, a plurality of dies were required when the location was different, even when the die shape was the same, but one die is sufficient by optionally changing the die angle position, and 4 corners of a work can be chamfered with a single die, thus reducing the die change time. In the bolt hole circle, arc pattern function, and circular nibbling, the C–axis is controlled automatically so that one side of a die always faces the center of the circle at each hole position.

34.14.1 Relation with Absolute/Incremental Command (G90/G91)

C–axis command is always regarded as an absolute command regardless of incremental command (G91) and absolute command (G90).

34.14.2 Positioning to Smaller Angle Rotation Direction

C axis can be automatically positioned to the commanded position in the smaller rotation angle from the present position.

34.14.3 Relation between T Command and C Axis Command

This function allows a T command to be executed after positioning the C–axis to the reference point automatically when the T command is specified on the tape, memory or MDI if the T–axis can move only when the C–axis is at the reference point (the fixed position on the machine at which C–axis can be reached by the manual reference point return).

34.14.4 Controlling the Punch and Die

Synchronous C–axis control can be performed for the punch and die by effecting control using either the same servo motor or separate servo motors.

34.14.5 C–axis Position Compensation

This function can be used for die–indexable tools. It simplifies mechanical adjustment like reference position adjustment performed during tool mounting by effecting automatic C–axis compensation for tool indexing. The amount of compensation and the tool number for the die–indexable tool are specified by parameters. (Up to 20 tools with tool numbers of 1 to 9999)

34.14.6 C–axis Backlash Compensation for Individual Indexes

C–axis backlash compensation can be effected for die–indexable tools for individual indexes. Changing the backlash compensation value by index enables high–precision machining. The amount of compensation and the tool number for the die–indexable tool are specified by parameters. (Up to 20 tools with tool numbers of 1 to 9999)

34.15 S CODE OUTPUT

The S code is specified with a 5-digit numeric value following address S. The 5-digit numeric value is output to the PMC as a 32-bit binary code. The code is maintained until another S is specified. The maximum number of input digits for S can be specified using a parameter.

34.16 ADVANCED PREVIEW CONTROL (G08)

This function is designed for high–speed precise machining in auxiliary function. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase as the feedrate becomes higher can be suppressed.
The tool can then follow specified values accurately and errors in the machining profile can be reduced.
This function becomes effective when advanced preview control mode is entered.

Format

G08 P_ ;

P1 : Turn on advanced preview control mode.
P0 : Turn off advanced preview control mode.

in advanced preview control mode, the following functions are available:

- Linear acceleration/deceleration before interpolation
- Automatic corner deceleration function

35 INTERFACE WITH THE POWER MATE CNC

35.1 FANUC SERVO MOTOR (β SERIES I/O LINK OPTION MANUAL HANDLE INTERFACE (PERIPHERAL EQUIPMENT CONTROL)

This function allows the use of a manual pulse generator on the host side to perform manual handle feed for the (β servo unit. The host posts pulses from the manual pulse generator to the (β servo unit via the I/O Link. A magnification can also be applied to the pulse output signal of the manual pulse generator by parameter setting. This function can be used only with the peripheral equipment control interface.

III. AUTOMATIC PROGRAMMING FUNCTION

1 GENERAL

Manual Guide 0*i* was developed to aid in generating a machining program for the Series 0*i*–B/0*i*–C. Manual Guide 0*i* is provided for lathes and machining centers (or milling machines).

A machining program, in which alphabetical addresses and numerical values are paired, includes a series of machining commands specified by the operator. This enables a CNC program to be created as a series of commands that perform machining operations. Multiple machining commands can be combined to perform complicated machine work.
Manual Guide 0*i* can also edit an exiting machining program. Online help that comes with the product can be used during development of a program or used as a reference for programming.

Manual Guide 0*i* has the following functions.

1) Process data input
   Helps the operator to input process data such as the cutting speed or spindle rotation commands.
2) G–code assistance
   Uses text or a figure to describe G–codes that can be normally used by the CNC and assist in inputting the codes.
3) M–code assistance
   Describes M–codes that can be used by the machine and assists in inputting the codes. Machine tool builders can change an indicated M code by using the macro executor.
4) Cycle machining
   Manual Guide 0*i* for machining centers (or milling machines) allows drilling, facing of standard figures, pocketing, and grooving to be input and executed as cycle machining.
   Manual Guide 0*i* for lathes allows turning drilling, turning rough/finish machining of an arbitrary figure, turning grooving, and turning threading to be input and executed as cycle machining.
5) Contour programming
   Allows input of an arbitrary figure consisting of lines and circles. This contour programming function includes advanced figure calculations such as 10–block pending calculation or 11–pattern auxiliary calculation.

2 PROCESS ASSISTANCE

The [PROCESS CONTROL INFORMATION] screen allows you to input information on the process requirements of a machining program.

```
PROCESS CONTROL INFORMATION

        -- FEED ---     F=
        -- SPINDLE -- DIR=
                        S=
        -- COOLANT -- CLT=
        -- TOOLING -- NUM=
                        H=
                        D=
        -- TL COMPG --LEN=
                      DIA=


INPUT DESIRED FEED RATE
0 ~ 30000
NUM=
```

3 G–CODE ASSISTANCE

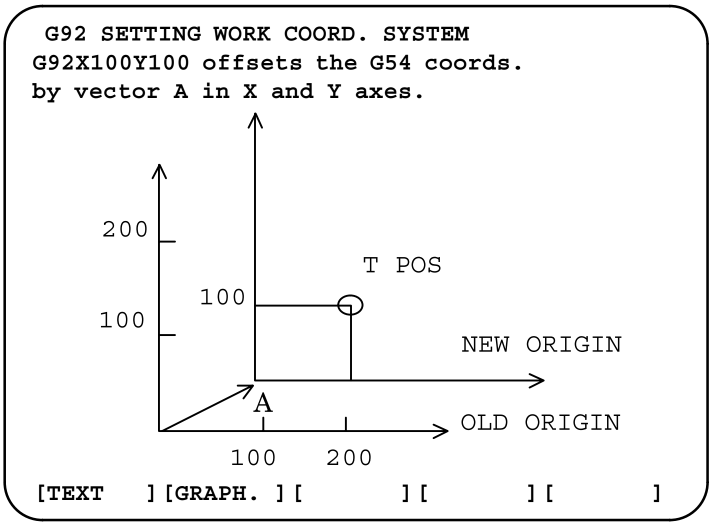
To access the “G–Code Assistance” press the “G CODE” soft–key on the display unit. This will display the G–Code help menu. The G–code help menu lists all of the G–codes supported by your control system. The menu is divided into several pages. The total number of help pages and the current page appear at the top of the menu.

```
G CODE HELP   1 OF 7
  G00  RAPID MOVE
  G01  LINEAR MOVE
  G02  CW CIRCLE
  G03  CCW CIRCLE
  G04  DWELL
  G08  LOOK AHEAD CONTROL
  G09  EXACT STOP
  G17  XY PLANE SELECT
  G18  XZ PLANE SELECT


  PAGE KEYS FOR MORE
NUM=
  INPUT 17 FOR G17 HELP
[       ][       ][       ][       ][       ]
```

The following description assumes G–codes used to set the work coordinate system.
By typing in “92” and pressing the “INPUT” key on the MDI key panel, the help information for G92 is displayed.

```
 G92 SETTING WORK COORD. SYSTEM
By specifying G92 command, a workpiece
coordinate system (selected by G54 to
G59) is shifted to set a new workpiece
coordinate system origin. Then, the
workpiece zero point offset values.
This means that all the workpiece
coordinate systems are shifted by the
same amount.






[TEXT   ][GRAPH. ][       ][       ][       ]
```

The above screen uses text for explanation, but a figure for explanation can also be displayed by pressing [GRAPH.]. However, not all G–codes are shown in this figure.


G92 SETTING WORK COORD. SYSTEM
G92X100Y100 offsets the G54 coords.
by vector A in X and Y axes.
200
T POS
100
100
NEW ORIGIN
A
OLD ORIGIN
100
200
[TEXT][GRAPH.][ ][ ][ ]

4 M–CODE ASSISTANCE

M–codes are used by the CNC to ask the executing of the machine auxiliary process. The M–code assistance function can be used to input M–codes. To select M–code assistance, press the [M CODE] soft key of the display unit. A list of M–codes appears.

```
M CODE HELP   1 OF 3
  M00  PROGRAM STOP
  M01  OPTIONAL STOP
  M02  PROGRAM END
  M03  SPINDLW CW
  M04  SPINDLE CCW
  M05  SPINDLE STOP
  M06  TOOL CHANGE
  M07  FLOOD COOLANT
  M08  MIST COOLANT


  PAGE KEYS FOR MORE
NUM=
  INPUT 30 FOR M30 HELP
[       ][       ][       ][       ][       ]
```

The following is the screen example of optional stop M code.

```
 M01 OPTIONAL STOP
M01 will cause the part program to stop
execution only if the OPTIONAL STOP
function is active. This is usually a
push button on the operator panel.
If the optional stop becomes active, the
operator will be required to press the
cycle start button to continue.







[TEXT   ][GRAPH. ][       ][       ][       ]
```

5 CANNED CYCLE MACHINING

5.1 CYCLE MACHINING FOR MACHINING CENTERS (OR MILLING MACHINES)

The cycle machining function of Manual Guide 0*i* allows you to input cycle machining blocks.
Manual Guide 0*i* for machining centers (or milling machines) can use the following cycle machining functions.

Hole Machining			
	Machining type block	G1000	Center drilling
		G1001	Drilling
		G1002	Tapping
		G1003	Reaming
		G1004	Boring
		G1005	Fine boring
		G1006	Back boring
	Figure block	G1210	Hole pattern – Points
		G1211	Hole pattern – Line (EQ)
		G1212	Hole pattern – Line (NE)
		G1213	Hole pattern – Grid
		G1214	Hole pattern – Square
		G1215	Hole pattern – Circle
		G1216	Hole pattern – Arc (EQ)
		G1217	Hole pattern – Arc (NE)
Face machining			
	Machining type block	G1020	Facing – Rough
		G1021	Facing – Finish
	Figure block	G1220	Fixed figure – Square
		G1221	Fixed figure – Circle
Pocket machining			
	Machining type block	G1040	Pocketing – Rough
		G1041	Pocketing – Bottom finish
		G1042	Pocketing – Side finish
		G1043	Pocketing – Chamfer
		G1044	Pocket – Center drilling
		G1045	Pocket – Drilling
	Figure block	G1220	Fixed figure – Square
		G1221	Fixed figure – Circle
		G1222	Fixed figure – Track
Groove machining			
	Machining type block	G1050	Grooving – Rough
		G1051	Grooving – Bottom finish
		G1052	Grooving – Side finish
		G1053	Grooving – Chamfer
		G1054	Center drilling
		G1055	Drilling
	Figure block	G1223	Fixed figure – Line

5.2 CYCLE MACHINING FOR LATHES

Manual Guide 0*i* for lathes can use the following cycle machining functions.

Lathe Drilling			
	Machining type block	G1100	Center drilling
		G1101	Drilling
		G1102	Tapping
		G1103	Reaming
		G1104	Boring
Stock Removal in Turning			
	Machining type block	G1120	Outer rough
		G1121	Inner rough
		G1122	End face rough
		G1123	Outer finish
		G1124	Inner finish
		G1125	End face finish
	Figure block	G1400	Start point
		G1401	Line
		G1402	Arc CW
		G1403	Arc CCW
		G1406	End of figures
Grooving in Turning			
	Machining type block	G1130	Outer rough
		G1132	End face rough
		G1133	Outer finish
		G1135	End face finish
	Figure block	G1460	Normal form
		G1461	Trapezoidal form
Threading			
	Machining type block	G1140	Outer thread
		G1141	Inner thread
	Figure block	G1450	Start point and end points

6 OPERATIONS OF CONTOUR PROGRAMMING

The contour programming function of Manual Guide 0*i* allows you to input a maximum of 40 types of arbitrary figures consisting of lines and circles. This “contour programming” involves high–performance contour calculation such as 10 blocks pending and auxiliary calculation with 11 patterns.

In “contour programming,” an operator can enter contour figures consisting of lines and circles and easily create a tool motion NC program of G01/G02/G03 that traces the contour figure.

Only tool motion commands are supported; other commands such as auxiliary functions must be entered manually in some other way.

While entering a contour figure, auxiliary calculation (other than cross point calculation) can be used to obtain the end point coordinate values of a figure.

6.1 AUXILIARY CALCULATION FUNCTION

(1) Data items in which auxiliary calculation can be used

(a) Start point
- Coordinate (X, Y) of start point

(b) Line
- Coordinate (X, Y) of end point
- Angle of a line (A)

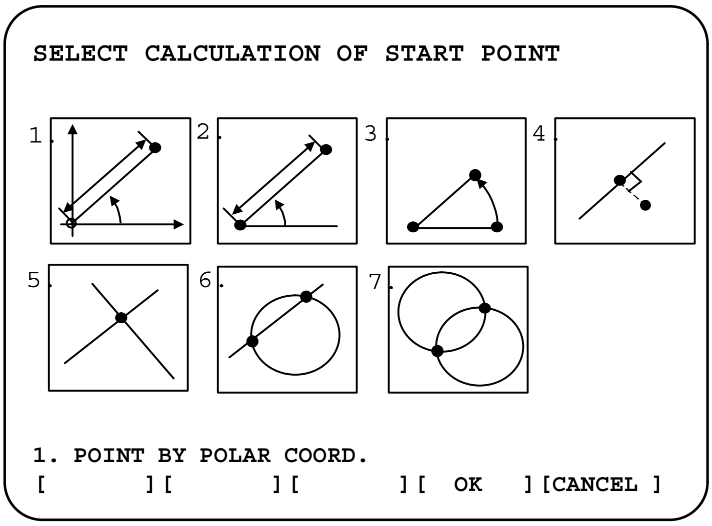
(c) Arc
- Coordinate (X, Y) of end point
- Coordinate (I, J) of center
- Specifying an arc

(2) Calculation type available in auxiliary calculation

(a) Calculation of coordinate
- A point specified by polar coordinate
- A point specified by a point, angle and distance
- A point specified by rotating a point
- Neighbor point of a line
- Cross point between 2 lines
- Cross point between line and arc
- Cross point between 2 arcs

(b) Calculation of angle
- Angle of the line passes 2 points
- Angle of a line rectangular to the line passes 2 points

(c) Specify an arc (Center and radius)
- An arc passes 1 point and its center coordinate has been determined
- An arc passes 2 points and its radius has been determined
- An arc passes 3 points



(Example of the auxiliary calculation menu screen)

APPENDIX

A RANGE OF COMMAND VALUE

A.1 T SERIES

Linear axis

• In case of metric input, feed screw is metric

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 mm	0.0001 mm
Least command increment	X : 0.0005 mm (diameter) Z : 0.001 mm (radius)	X : 0.00005 mm (diameter) Z : 0.0001 mm (radius)
Max. programmable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	240000 mm/min	100000 mm/min
Feedrate range ***1**	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev	Feed per minute : 1 to 100000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Backlash compensation	0 to ±0.255 mm	0 to ±0.255 mm
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec

• In case of inch input, feed screw is metric

	Increment system	
	IS–B	**IS–C**
Least input increment	0.0001 inch	0.00001 inch
Least command increment	X : 0.00005 inch (diameter) Z : 0.0001 inch (radius)	X : 0.000005 inch (diameter) Z : 0.00001 inch (radius)
Max. programmable dimension	±9999.9999 inch	±393.70078 inch
Max. rapid traverse ***1**	240000 mm/min	100000 mm/min
Feedrate range ***1**	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev	Feed per minute : 0.01 to 4000 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.9999 inch
Backlash compensation	0 to ±0.255 mm	0 to ±0.255 mm
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of inch input, feed screw is inch**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.0001 inch	0.00001 inch
Least command increment	X : 0.00005 inch (diameter) Z : 0.0001 inch (radius)	X : 0.000005 inch (diameter) Z : 0.00001 inch (radius)
Max. programmable dimension	±9999.9999 inch	±999.99999 inch
Max. rapid traverse ***1**	9600 inch/min	4000 inch/min
Feedrate range ***1**	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev	Feed per minute : 0.01 to 4000 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.9999 inch
Backlash compensation	0 to ±0.0255 inch	0 to ±0.0255 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of metric input, feed screw is inch**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 mm	0.0001 mm
Least command increment	X : 0.00005 inch (diameter) Z : 0.0001 inch (radius)	X : 0.000005 inch (diameter) Z : 0.00001 inch (radius)
Max. programmable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	9600 inch/min	960 inch/min
Feedrate range ***1**	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev	Feed per minute : 1 to 100000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Backlash compensation	0 to ±0.0255 inch	0 to ±0.0255 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

Rotation axis

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 deg	0.0001 deg
Least command increment	0.001 deg	0.0001 deg
Max. programmable dimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse ***1**	240000 deg/min	100000 deg/min
Feedrate range ***1**	1 to 240000 deg/min	1 to 100000 deg/min
Incremental feed	0.001, 0.01, 0.1, 1deg/step	0.0001, 0.001, 0.01, 0.1 deg/step
Backlash compensation	0 to ±0.255 deg	0 to ±0.255 deg

NOTE

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity.
As a whole system, limitations depending on servo system must also be considered.

A.2 M SERIES

Linear axis

• In case of metric input, feed screw is metric

	Increment system		
	IS–A	**IS–B**	**IS–C**
Least input increment	0.01 mm	0.001 mm	0.0001 mm
Least command increment	0.01 mm	0.001 mm	0.0001 mm
Max. programmable dimension	±999999.99 mm	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	240000 mm/min	240000 mm/min	100000 mm/min
Feedrate range ***1**	1 to 240000 mm/min	1 to 240000 mm/min	1 to 100000 mm/min
Incremental feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.99 mm	0 to ±999.999 mm	0 to ±999.9999 mm
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec	0 to 99999.999 sec

• In case of inch input, feed screw is metric

	Increment system		
	IS–A	**IS–B**	**IS–C**
Least input increment	0.001 inch	0.0001 inch	0.00001 inch
Least command increment	0.01 inch	0.0001 inch	0.00001 inch
Max. programmable dimension	±99999.999 inch	±9999.9999 inch	±393.70078 inch
Max. rapid traverse ***1**	240000 mm/min	240000 mm/min	100000 mm/min
Feedrate range ***1**	0.01 to 9600 inch/min	0.01 to 9600 inch/min	0.01 to 4000 inch/min
Incremental feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.999 inch	0 to ±99.9999 inch	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of inch input, feed screw is inch**

	Increment system		
	IS–A	**IS–B**	**IS–C**
Least input increment	0.001 inch	0.0001 inch	0.00001 inch
Least command increment	0.001 inch	0.0001 inch	0.00001 inch
Max. programmable dimension	±99999.999 inch	±9999.9999 inch	±9999.9999 inch
Max. rapid traverse ***1**	9600 inch/min	0.01 to 9600 inch/min	4000 inch/min
Feedrate range ***1**	0.01 to 9600 inch/min	0.01 to 9600 inch/min	0.01 to 4000 inch/min
Incremental feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.999 inch	0 to ±99.9999 inch	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of metric input, feed screw is inch**

	Increment system		
	IS–A	**IS–B**	**IS–C**
Least input increment	0.01 mm	0.001 mm	0.0001 mm
Least command increment	0.001 inch	0.0001 inch	0.00001 inch
Max. programmable dimension	±999999.99 mm	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	9600 inch/min	9600 inch/min	4000 inch/min
Feedrate range ***1**	1 to 240000 mm/min	1 to 240000 mm/min	1 to 100000 mm/min
Incremental feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.99 mm	0 to ±999.999 mm	0 to ±999.9999 mm
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec	0 to 9999.9999 sec

Rotation axis

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 deg	0.0001 deg
Least command increment	±0.001 deg	±0.0001 deg
Max. programmabledimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse ***1**	240000 deg/min	100000 deg/min
Feedrate range ***1**	1 to 240000 deg/min	1 to 100000 deg/min
Incremental feed	0.001, 0.01, 0.1, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step

NOTE

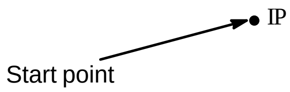
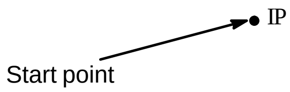
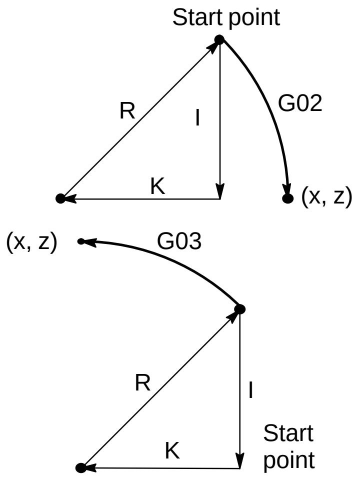
*1 The feedrate range shown above are limitations depending on CNC interpolation capacity.
As a whole system, limitations depending on servo system must also be considered.

B FUNCTIONS AND TAPE FORMAT LIST

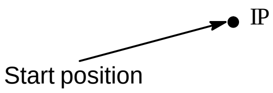
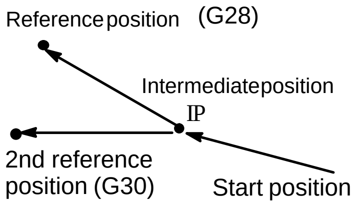
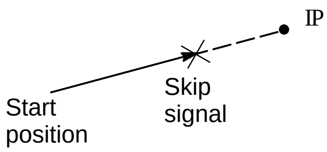
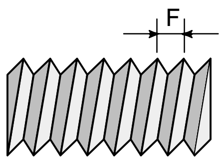
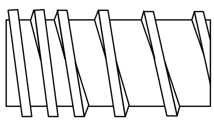
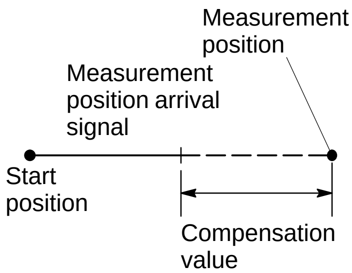
B.1 T SERIES

Some functions cannot be added as options depending on the model.
In the tables below, IP _:presents a combination of arbitrary axis addresses using X and Z.
x = 1st basic axis (X usually)
z = 2nd basic axis (Z usually)

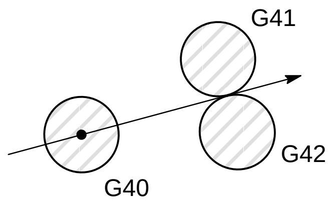
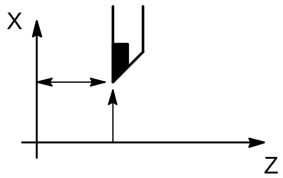
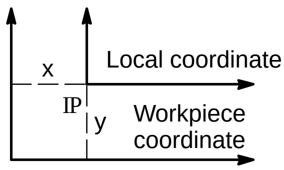
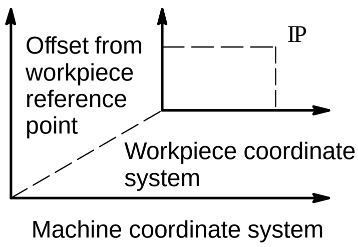
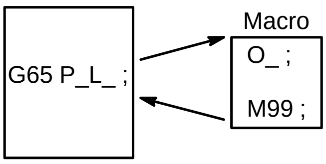
(1/5)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		G17 {G02 / G03} X_ Y_ {R_ / I_ J_} F_ ; G18 {G02 / G03} X_ Z_ {R_ / I_ K_} F_ ; G19 {G02 / G03} Y_ Z_ {R_ / J_ K_} F_ ;
Dwell (G04)		G04 {X_ / P_} ;
Cylindrical interpolation (G07.1) (G107)		G07.1 IP_R_; Cylindrical interpolation mode G07.1 IP0 ; Cylindrical interpolation mode cancel R: Radius of cylinder
Polar coordinate interpolation (G12.1, G13.1) (G112, G113)		G12.1 ; Polar coordinate interpolation mode G13.1 ; Polar coordinate interpolation mode cancel
Change of offset value by program(G10)		Tool geometry offset value G10 P_ X(U)_ Y(V)_ Z(W)_ R(C)_ Q_ ; P=1000+Geometry offset number Tool wear offset value G10 P_ X(U)_ Y(V)_ Z(W)_ R(C)_ Q_ ; P=Wear offset number

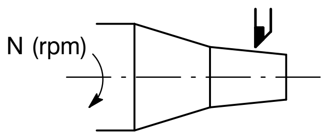
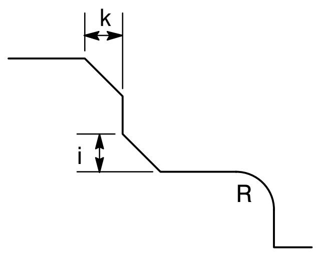
(2/5)

Functions	Illustration	Tape format
Plane selection (G17, G18, G19)		G17 ; G18 ; G19 ;
Inch/metric conversion (G20, G21)		Inch input : G20 Metric input : G21
Stored stroke check 2, 3 (G22, G23)		G22X_ Z_ I_K_ ; G23 ;
Spindle speed fluctuation detection (G25, G26)		G25 ; G26 P_ Q_ R_ ;
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd reference position re-turn (G30)		G28 IP_ ; G30 IP_ ;
Skip function (G31)		G31 IP_ F_;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_;
Variable–lead threading		G34 IP_ F_K_;
Automatic tool compensation (G36, G37)		G36 X xa ; G37 Z za ;

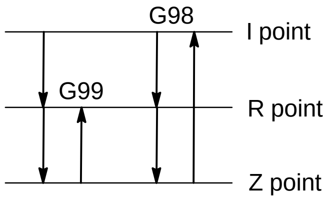
(3/5)

Functions	Illustration	Tape format
Tool nose radius compensation (G40, G41, G42)		$\left\{\begin{array}{l}\text{G41}\\\text{G42}\end{array}\right\}$ IP_ ; G40 : Cancel
Coordinate system setting Spindle speed setting (G50)		G50 IP_ ; Coordinate system setting G50 S_ ; Maximum spindle speed setting
Polygon turning (G50.2, G51.2) (G250, G251)		G51.2 (G251) P_Q_ ; P,Q : Rotation ratio of spindle and rotary axis G50.2 (G250) ; Cancel
Workpiece coordinate system preset (G50.3)		G50.3 IP 0 ;
Local coordinate system setting (G52)		G52 IP _ ;
Machine coordinate system selection (G53)		G53 IP _ ;
Workpiece coordinate system selection (G54 to G59)		$\left\{\begin{array}{c}\text{G54}\\:\\\text{G59}\end{array}\right\}$ IP _ ;
Custom macro (G65, G66, G67)		One–shot call G65 P_L <argument> ; P : Program number L : Repetition count Modal call G66 P_L <argument> ; G67 ; Cancel

(4/5)

Functions	Illustration	Tape format
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.13. FUNCTIONS TO SIMPLIFY PROGRAMMING	N_ G70 P_ Q_ ; G71 U_ R_ ; G71 P_ Q_ U_ W_ F_ S_ T_ ; G72 W_ R_ ; G72 P_ Q_ U_ W_ F_ S_ T_ ; G73 U_ W_ R_ ; G73 P_ Q_ U_ W_ F_ S_ T_ ; G74 R_ ; G74 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75 R_ ; G75 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76 P_ Q_ R_ ; G76 X(u)_ Z(w)_ P_ Q_ R_ F_ ; { G90 / G92 / G94 } X_ Z_ R_ F_ ;
Canned cycle for drilling (G80 to G89)	See Chapter 13, “Functions to Simplify Programming” in Part II.	G80 ; Cancel G83 X(U)_C(H)_Z(W)_R_Q_P_F_M_K_; G84 X(U)_C(H)_Z(W)_R_P_F_M_K_; G85 X(U)_C(H)_Z(W)_R_P_F_M_K_; G87 Z(W)_C(H)_X(U)_R_Q_P_F_M_K_; G88 Z(W)_C(H)_X(U)_R_P_F_M_K_; G89 Z(W)_C(H)_X(U)_R_P_F_M_K_;
Constant surface speed control (G96/G97)	m/min or feet/min 	G96 S_ ; G97 ; Cancel
Feed per minute (G98) Feed per revolution (G99)	mm/min inch/min mm/rev inch/rev	G98 … F_ ; G99 … F_ ;
Chamfering, Corner R		X_ ; { C(K) ±k / R_ } P_ ; Z_ ; { C(I) ±i / R_ } P_ ;

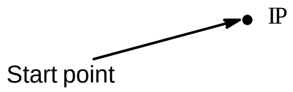
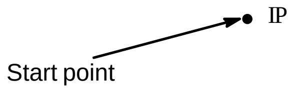
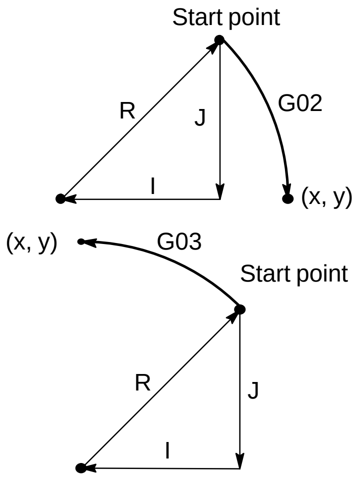
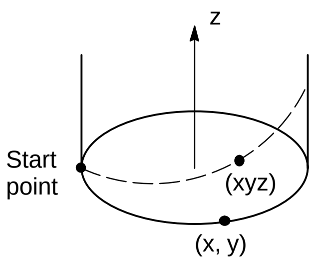
(5/5)

Functions	Illustration	Tape format
Absolute/incremental programming (G90/G91) (With G code system B or C)		G90_ ; Absolute programming G91_ ; Incremental programming G90_ G91_ ; Absolute and incremental programming
Return to initial point/R point (G98, G99) (With G code system B or C)	 	G98_ ; G99_ ;

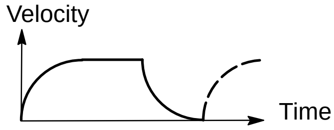
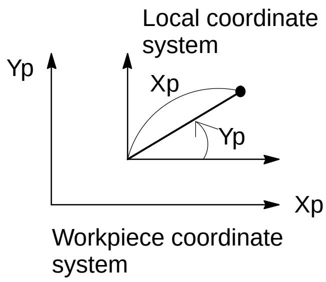
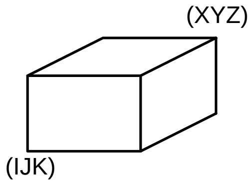
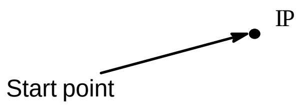
B.2 M SERIES

Some functions cannot be added as options depending on the model.
In the tables below, IP _:presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_Y_Z_A_).
x = 1st basic axis (X usually)
y = 2nd basic axis (Y usually)
z = 3rd basic axis (Z usually)

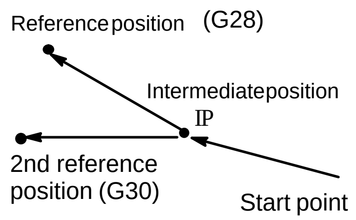
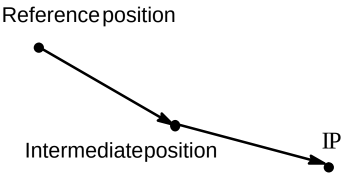
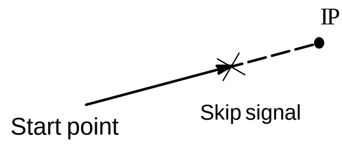
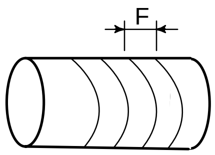
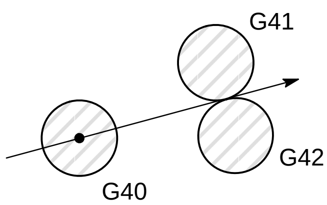
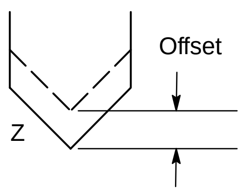
(1/6)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		G17 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Y_ $\begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix}$ F_ ; G18 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Z_ $\begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix}$ F_ ; G19 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ Y_ Z_ $\begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix}$ F_ ;
Helical interpolation (G02, G03)	 When G03 is specified for the XY plane	G17 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Y_ $\begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix}$ α_F_ ; G18 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ X_ Z_ $\begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix}$ α_F_ ; G19 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ Y_ Z_ $\begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix}$ α_F_ ; α: Any address other than that of a circular interpolation axis
Dwell (G04)		G04 $\begin{Bmatrix} X\_ \\ P\_ \end{Bmatrix}$;
Cylindrical interpolation (G07.1)		G07.1 IP_R_ ; Cylindrical interpolation mode R : Radius of cylinder G07.1 IP 0 ; Cylindrical interpolation mode cancel

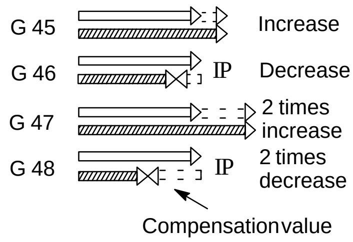
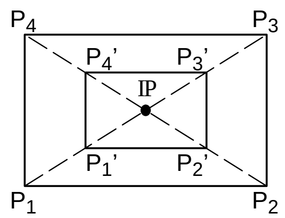
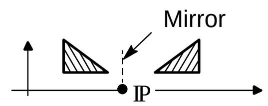
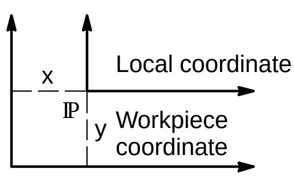
(2/6)

Functions	Illustration	Tape format
Advanced preview control (G08)		G08 P1 ; Advanced preview control mode on G08 P0 ; Advanced preview control mode off
Exact stop (G09)		G09 $\left\{\begin{matrix} G01 \\ G02 \\ G03 \end{matrix}\right\}$ IP_;
Change of offset value by program (G10)		· Tool offset memory A G10 L11 P_ R_ ; · Tool offset memory B G10 L10 P_ R_ ; (Geometry offset value) G10 L11 P_ R_ ; (Wear offset value) · Tool offset memory C G10 L10 P_ R_ ; (Geometry offset value/H) G10 L11 P_ R_ ; (Wear offset value/H) G10 L12 P_ R_ ; (Geometry offset value/D) G10 L13 P_ R_ ; (Wear offset value/D)
Polar coordinate input (G15, G16)		G17 G16 Xp_Yp ··· ; G18 G16 Zp_Xp ··· ; G19 G16 Yp_Zp ··· ; G15 ; Cancel
Plane selection (G17, G18, G19)		G17 ; G18 ; G19 ;
Inch/metric conversion (G20, G21)		G20 ; Inch input G21 ; Metric input
Stored stroke check (G22, G23)		G22 X_Y_Z_I_J_K_ ; G23 ; Cancel
Reference position return check (G27)		G27 IP_ ;

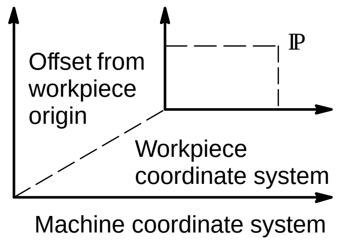
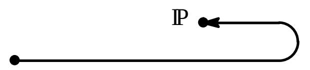
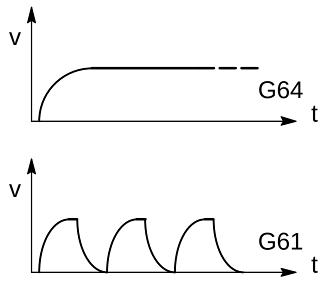
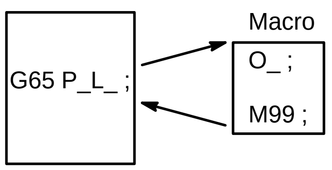
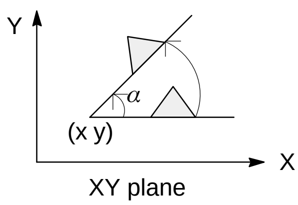
(3/6)

Functions	Illustration	Tape format
Reference position return (G28) 2nd, reference position return (G30)		G28 IP_ ; G30 IP_ ;
Return from reference position to start point (G29)		G29 IP_ ;
Skip function (G31)		G31 IP_ F_;
Threading (G33)		G33 IP _ F ; F : Lead
Cutter compensation C (G40 to G42)		{G17 / G18 / G19} {G41 / G42} D_ ; D : Tool offset number G40 : Cancel
Normal–direction control (G40.1, G41.1, G42.1) (G150, G151, G152)		G41.1 (G151) Normal–direction control left G42.1 (G152) Normal–direction control right G40.1 (G150) Normal–direction control cancel
Tool length compensation A (G43, G44, G49)		{G43 / G44} Z_ H_ ; {G43 / G44} H_ ; H : Tool offset number G49 : Cancel

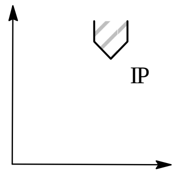
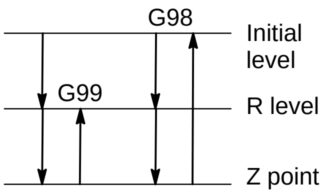
(4/6)

Functions	Illustration	Tape format
Tool length compensation B (G43, G44, G49)		$\begin{Bmatrix} \text{G17} \\ \text{G18} \\ \text{G19} \end{Bmatrix} \begin{Bmatrix} \text{G43} \\ \text{G44} \end{Bmatrix} \begin{Bmatrix} \text{Z\_} \\ \text{Y\_} \\ \text{X\_} \end{Bmatrix} \text{H\_ ;}$ $\begin{Bmatrix} \text{G17} \\ \text{G18} \\ \text{G19} \end{Bmatrix} \begin{Bmatrix} \text{G43} \\ \text{G44} \end{Bmatrix} \text{H\_ ;}$ H : Tool offset number G49 : Cancel
Tool length compensation C (G43, G44, G49)		$\begin{Bmatrix} \text{G43} \\ \text{G44} \end{Bmatrix} \alpha\text{\_H\_ ;}$ α : Any address of a single axis H : Tool offset number G49 : Cancel
Tool offset (G45 to G48)		$\begin{Bmatrix} \text{G45} \\ \text{G46} \\ \text{G47} \\ \text{G48} \end{Bmatrix} \text{IP \_ D\_ ;}$ D : Tool offset number
Scaling (G50, G51)		$\text{G51 X\_Y\_Z\_} \begin{Bmatrix} \text{P\_} \\ \text{I\_J\_K\_} \end{Bmatrix} ;$ P, I, J, K : Scaling magnification X, Y, Z : Coordinates of center of scaling G50 ; Cancel
Programmable mirror image (G50.1, G51.1)		G51.1 IP_ ; G50.1 ; ······ Cancel
Local coordinate system setting (G52)		G52 IP_ ;
Machine coordinate system selection (G53)		G53 IP_ ;

(5/6)

Functions	Illustration	Tape format
Workpiece coordinate system selection (G54 to G59) Additional workpiece coordinate system selection (G54.1)		G54 : G59 IP _ ; G54.1 P _ IP_ ;
Unidirectional positioning (G60)		G60 IP_ ;
Cutting mode Exact stop mode Tapping mode Automatic corner override		G64_ ; Cutting mode G61_ ; Exact stop mode G63_ ; Tapping mode G62_ ; Automatic corner override
Custom macro (G65, G66, G67)		One–shot call G65 P_L <argument> ; P : Program number L : Repetition count Continuous–state call G66 P_L <argument> ; G67 ; Cancel
Coordinate system rotation (G68, G69)		G68 { G17 X_Y_ / G18 Z_X_ / G19 Y_Z_ } R α ; G69 ; Cancel
Canned cycles (G73, G74, G80 – G89)	Refer to II.14. FUNCTIONS TO SIMPLIFY PROGRAMMING	G80 ; Cancel G73 G74 G76 G81 : G89 X_ Y_ Z_ P_ Q_ R_ F_ K_ ;
Absolute/incremental programming (G90/G91)		G90_ ; Absolute command G91_ ; Incremental command G90_ G91_ ; Combined use

(6/6)

Functions	Illustration	Tape format
Change of workpiece coordinate system (G92)		G92 IP_ ;
Workpiece coordinate system preset (G92.1)		G92.1 IP 0 ;
Feed per minute/rotation (G94, G95)	mm/min inch/min mm/rev inch/rev	G98 F_ ; G99 F_ ;
Constant surface speed control (G96, G97)		G96 S_ ; G97 S_ ;
Initial point return / R point return (G98, G99)		G98_ ; G99_ ;

C LIST OF TAPE CODE

ISO code										EIA code										Remarks	Custom macro B	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Not used	Used
0			◯	◯		○				0			◯			○				Number 0		
1	◯		◯	◯		◯			◯	1						◯			◯	Number 1		
2	◯		◯	◯		○		◯		2						○		◯		Number 2		
3			◯	◯		○		◯	◯	3				◯		○		◯	◯	Number 3		
4	◯		◯	◯		○	◯			4						○	◯			Number 4		
5			◯	◯		○	◯		◯	5				◯		○	◯		◯	Number 5		
6			◯	◯		○	◯	◯		6				◯		○	◯	◯		Number 6		
7	◯		◯	◯		○	◯	◯	◯	7						○	◯	◯	◯	Number 7		
8	◯		◯	◯	◯	○				8					◯	○				Number 8		
9			◯	◯	◯	○			◯	9				◯	◯	○			◯	Number 9		
A		◯				○			◯	a		◯	◯			○			◯	Address A		
B		◯				○		◯		b		◯	◯			○		◯		Address B		
C	◯	◯				○		◯	◯	c		◯	◯	◯		○		◯	◯	Address C		
D		◯				○	◯			d		◯	◯			○	◯			Address D		
E	◯	◯				○	◯		◯	e		◯	◯	◯		○	◯		◯	Address E		
F	◯	◯				○	◯	◯		f		◯	◯	◯		○	◯	◯		Address F		
G		◯				○	◯	◯	◯	g		◯	◯			○	◯	◯	◯	Address G		
H		◯			◯	○				h		◯	◯		◯	○				Address H		
I	◯	◯			◯	○			◯	i		◯	◯	◯	◯	○			◯	Address I		
J	◯	◯			◯	○		◯		j		◯		◯		○			◯	Address J		
K		◯			◯	○		◯	◯	k		◯		◯		○		◯		Address K		
L	◯	◯			◯	○	◯			l		◯				○		◯	◯	Address L		
M		◯			◯	○	◯		◯	m		◯		◯		○	◯			Address M		
N		◯			◯	○	◯	◯		n		◯				○	◯		◯	Address N		
O	◯	◯			◯	○	◯	◯	◯	o		◯				○	◯	◯		Address O		
P		◯		◯		○				p		◯		◯		○	◯	◯	◯	Address P		
Q	◯	◯		◯		○			◯	q		◯		◯	◯	○				Address Q		
R	◯	◯		◯		○		◯		r		◯			◯	○			◯	Address R		
S		◯		◯		○		◯	◯	s			◯	◯		○		◯		Address S		
T	◯	◯		◯		○	◯			t			◯			○		◯	◯	Address T		
U		◯		◯		○	◯		◯	u			◯	◯		○	◯			Address U		
V		◯		◯		○	◯	◯		v			◯			○	◯		◯	Address V		
W	◯	◯		◯		○	◯	◯	◯	w			◯			○	◯	◯		Address W		
X	◯	◯		◯	◯	○				x			◯	◯		○	◯	◯	◯	Address X		
Y		◯		◯	◯	○			◯	y			◯	◯	◯	○				Address Y		

ISO code										EIA code										Remarks	Custom macro B	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Not used	Used
Z		○		○	○	◦		○		z			○		○	◦			○	Address Z		
DEL	○	○	○	○	○	◦	○	○	○	Del		○	○	○	○	◦	○	○	○	Delete (deleting a mispunch)	×	×
NUL						◦				Blank						◦				No. punch. With EIA code, this code cannot be used in a significant information section.	×	×
BS	○				○	◦				BS			○		○	◦		○		Backspace	×	×
HT					○	◦			○	Tab			○	○	○	◦	○	○		Tabulator	×	×
LF or NL					○	◦		○		CR or EOB	○					◦				End of block		
CR	○				○	◦	○		○	___						◦				Carriage return	×	×
SP	○		○			◦				SP				○		◦				Space	□	□
%	○		○			◦	○		○	ER					○	◦		○	○	Absolute rewind stop		
(			○		○	◦				(2–4–5)				○	○	◦		○		Control out (start of comment)		
)	○		○		○	◦			○	(2–4–7)		○			○	◦		○		Control in (end of comment)		
+			○		○	◦		○	○	+		○	○	○		◦				Plus sign	Δ	
–			○		○	◦	○		○	–		○				◦				Minus sign		
:			○	○	○	◦		○		___						◦				Colon (address O)		
/	○		○		○	◦	○	○	○	/			○	○		◦			○	Optional block skip		
.			○		○	◦	○	○		.		○	○		○	◦		○	○	Period (decimal point)		
#	○		○			◦		○	○	Parameter (No. 6012)						◦				Sharp		
$			○			◦	○			___						◦				Dollar sign	×	×
&	○		○			◦	○	○		&					○	◦	○	○		Ampersand	Δ	○
’			○			◦	○	○	○	___						◦				Apostrophe	Δ	Δ
*	○		○		○	◦		○		Parameter (No. 6010)						◦				Asterisk	Δ	
,	○		○		○	◦	○			,			○	○	○	◦		○	○	Comma		
;	○		○	○	○	◦		○	○	___						◦				Semicolon	×	×
<			○	○	○	◦	○									◦				Left angle bracket	Δ	Δ

ISO code										EIA code										Remarks	Custom macro B	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Not used	Used
=	○		○	○	○	∘	○		○	Parameter (No. 6011)						∘				Equal sign	△	
>	○		○	○	○	∘	○	○		——						∘				Right angle bracket	△	△
?			○	○	○	∘	○	○	○	——						∘				Question mark	△	○
@	○	○				∘				——						∘				Commercial at mark	△	○
”			○					○		——						∘				Quotation mark	△	△
[	○	○		○	○	∘		○	○	Parameter (No. 6013)						∘				Left square bracket	△	
]	○	○		○	○	∘	○		○	Parameter (No. 6014)						∘				Right square bracket	△	

NOTE

1 The symbols in the Remarks column have the following meanings:
   Blank: Registered in memory as significant information. Any invalid use of these codes in information other than a comment will cause an alarm.
   × : Not registered in memory (ignored)
   △ : Registered in memory but ignored during the execution of a program
   ○ : Registered in memory. The use of these codes in information other than a comment will cause an alarm.
   □ : Not registered in memory when used in information other than a comment. Registered in memory when used in a comment.

2 Any code other than those listed in the table is always ignored, provided its parity is valid.

3 Any code having an invalid parity will cause a TH alarm. Within a comment, however, such a code is ignored and will not cause a TH alarm.

4 With EIA code, the code with all eight holes punched has special meaning. It is ignored and does not cause any parity alarm.

D EXTERNAL DIMENSIONS OF EACH UNIT

<table>
<tr><th colspan="3">Name</th><th>Specification</th><th>Fig., No.</th></tr>
<tr><td colspan="3">CNC control unit (7.2″/8.4″ LCD, MDI horizontal type)</td><td></td><td>Fig. U1</td></tr>
<tr><td colspan="3">CNC control unit (7.2″/8.4″ LCD, MDI vertical type)</td><td></td><td>Fig. U2</td></tr>
<tr><td colspan="3">I/O unit for 0i</td><td>A02B–0309–C001</td><td>Fig. U5</td></tr>
<tr><td colspan="3">HSSB interface board type 2 (1CH) on the personal computer side (ISA)</td><td>A20B–8001–0583</td><td rowspan="2">Fig. U16(a)</td></tr>
<tr><td colspan="3">HSSB interface board type 2 (2CH) on the personal computer side (ISA)</td><td>A20B–8001–0582</td></tr>
<tr><td colspan="3">HSSB interface board type 2 (1CH) on the personal computer side (PCI)</td><td>A20B–8001–0961</td><td rowspan="2">Fig. U16(b)</td></tr>
<tr><td colspan="3">HSSB interface board type 2 (2CH) on the personal computer side (PCI)</td><td>A20B–8001–0960</td></tr>
<tr><td colspan="2">α position coder</td><td>10000min^{-1}</td><td>A860–0309–T302</td><td>Fig. U17</td></tr>
<tr><td colspan="3">Manual pulse generator</td><td>A860–0203–T001</td><td>Fig. U18</td></tr>
<tr><td colspan="3" rowspan="6">Pendant type manual pulse generator</td><td>A860–0203–T004</td><td rowspan="6">Fig. U19</td></tr>
<tr><td>A860–0203–T005</td></tr>
<tr><td>A860–0203–T007</td></tr>
<tr><td>A860–0203–T010</td></tr>
<tr><td>A860–0203–T012</td></tr>
<tr><td>A860–0203–T013</td></tr>
<tr><td colspan="3">Separate detector interface unit</td><td>A02B–0236–C205, C204</td><td>Fig. U20</td></tr>
<tr><td colspan="3">Battery case for separate detector interface unit (ABS)</td><td>A06B–6050–K060</td><td>Fig. U21</td></tr>
<tr><td colspan="3">CNC battery unit for external installation</td><td>A02B–0236–C281</td><td>Fig. U22</td></tr>
<tr><td rowspan="3">Punch panel</td><td rowspan="3">Narrow width type</td><td>Cable length : 1m</td><td>A02B–0120–C191</td><td rowspan="3">Fig. U24</td></tr>
<tr><td>Cable length : 2m</td><td>A02B–0120–C192</td></tr>
<tr><td>Cable length : 5m</td><td>A02B–0120–C193</td></tr>
<tr><td rowspan="3">Machine operator’s panel</td><td colspan="2">Main panel B</td><td>A02B–0236–C231</td><td>Fig. U25</td></tr>
<tr><td colspan="2">Sub panel A</td><td>A02B–0236–C232</td><td>Fig. U26</td></tr>
<tr><td colspan="2">Sub panel B1</td><td>A02B–0236–C235</td><td>Fig. U27</td></tr>
</table>

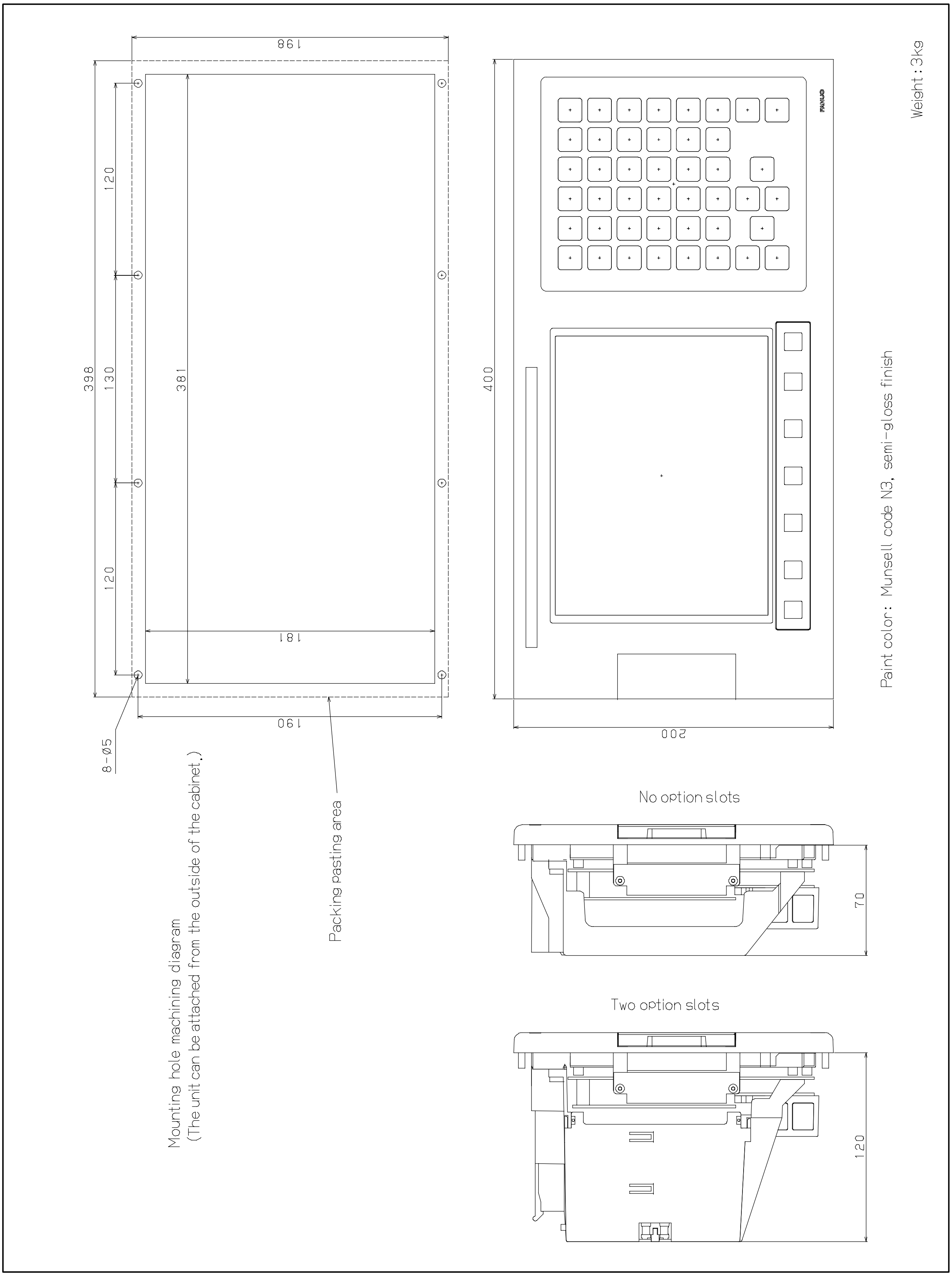


Fig.U1 CNC control unit (7.2″/8.4″ LCD, MDI horizontal type)

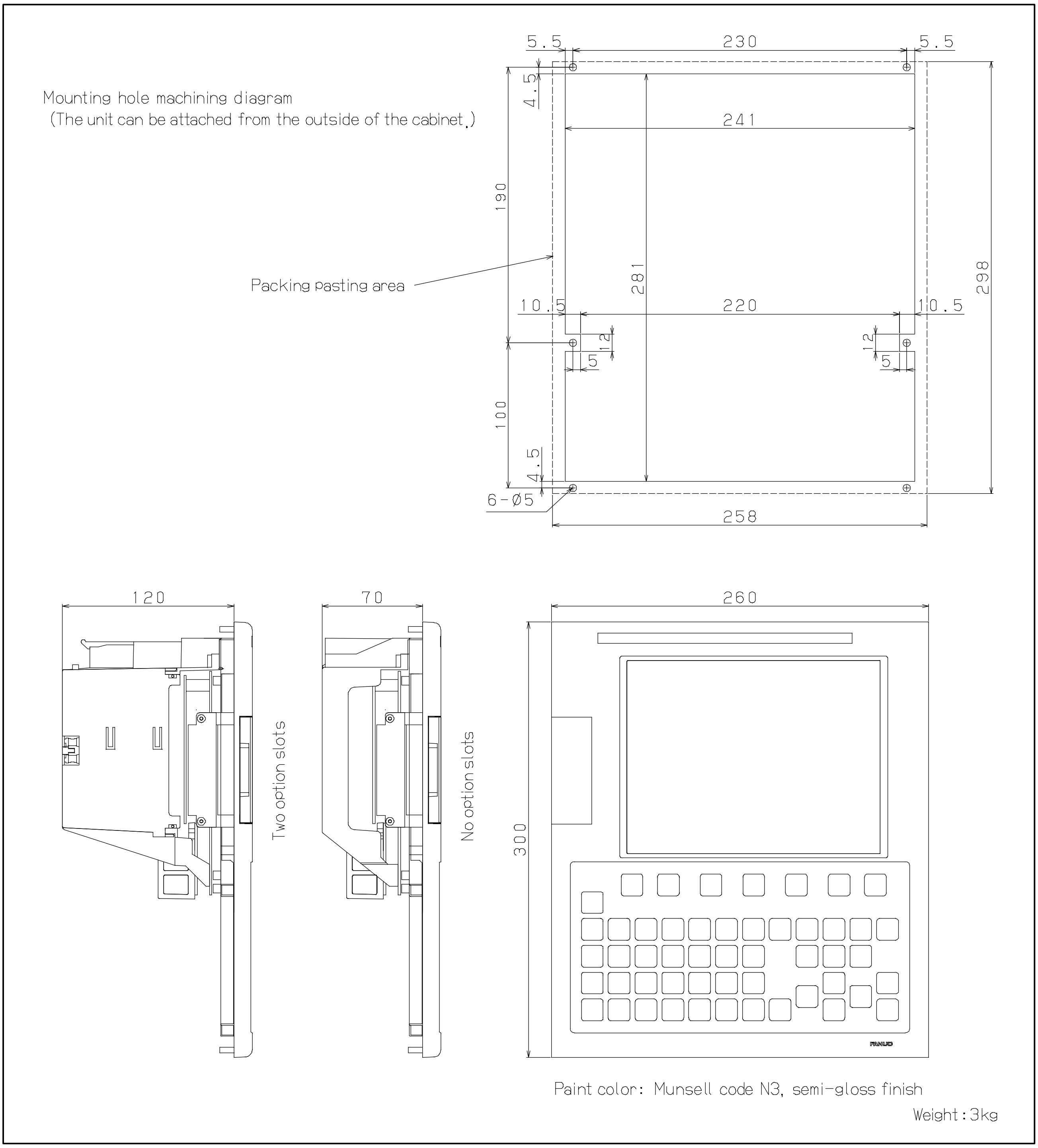


Fig.U2 CNC control unit (7.2″/8.4″ LCD, MDI vertical type)

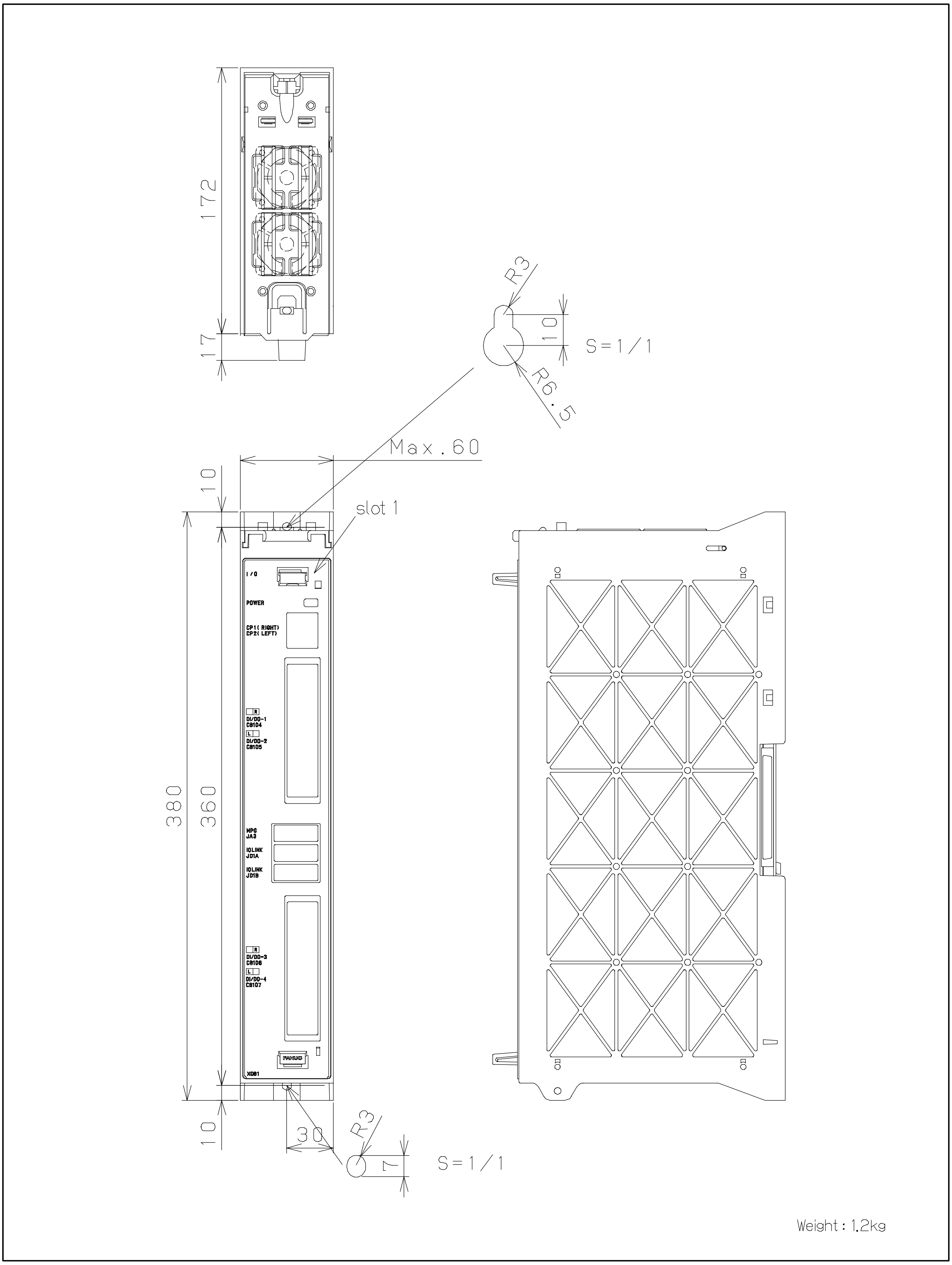


Fig.U5 I/O unit for 0*i*
Specification No. : A02B–0309–C001

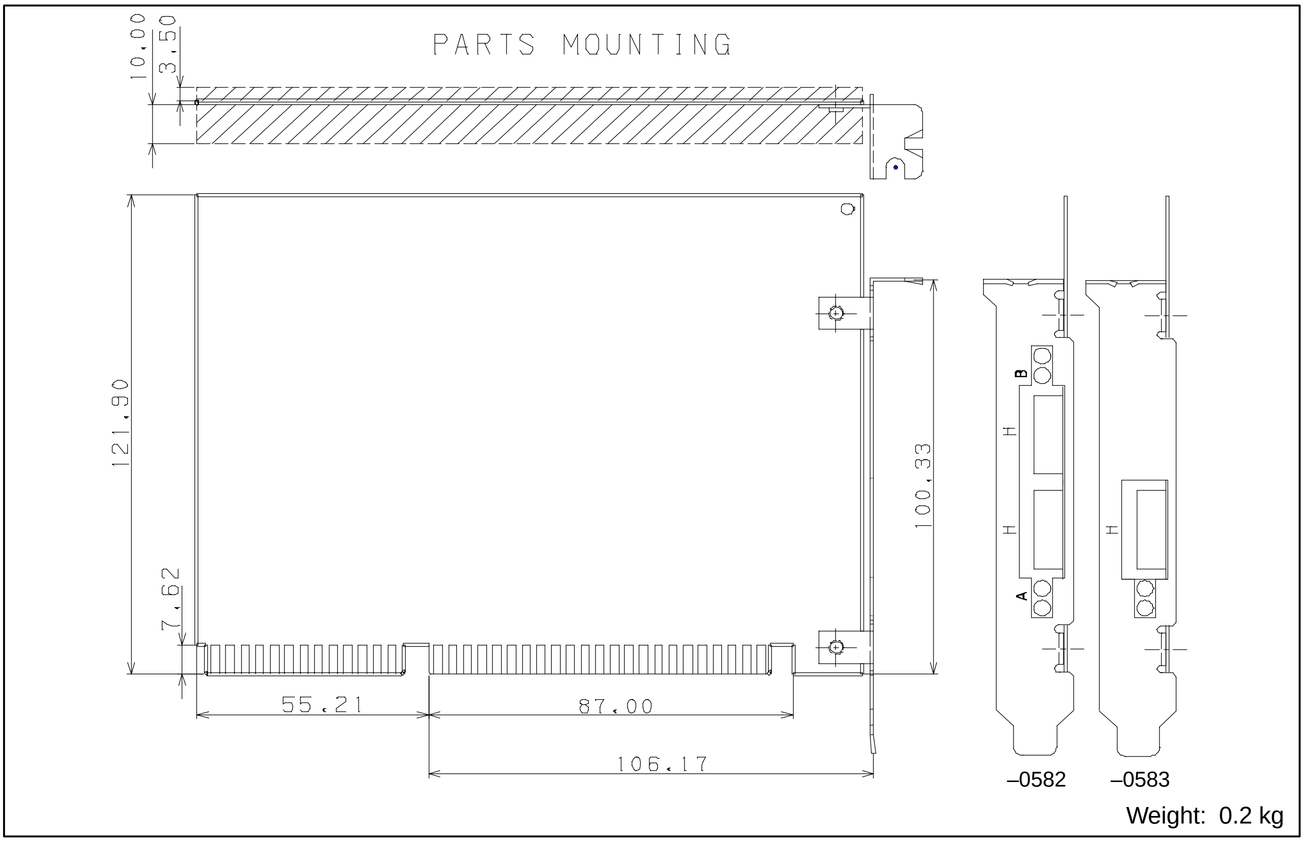


Fig.U16 (a) High–speed serial bus interface board type 2 (PC) (ISA bus version)
Specification No. : A20B–8001–0583 (1 CH)
A20B–8001–0582 (2 CH)

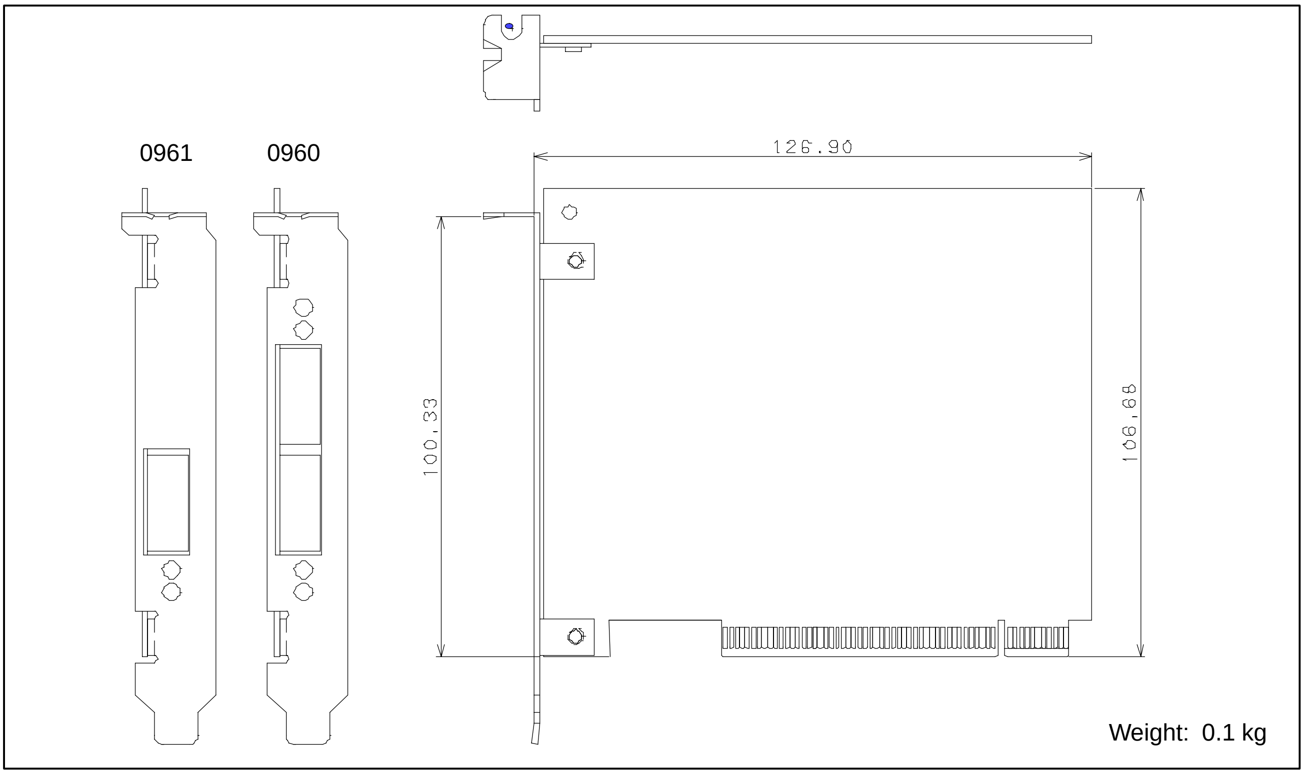


Fig.U16 (b) Interface Board for Personal Computer (PCI bus version)
Specification No. : A20B–8001–0960 (2 CH)
A20B–8001–0961 (1 CH)

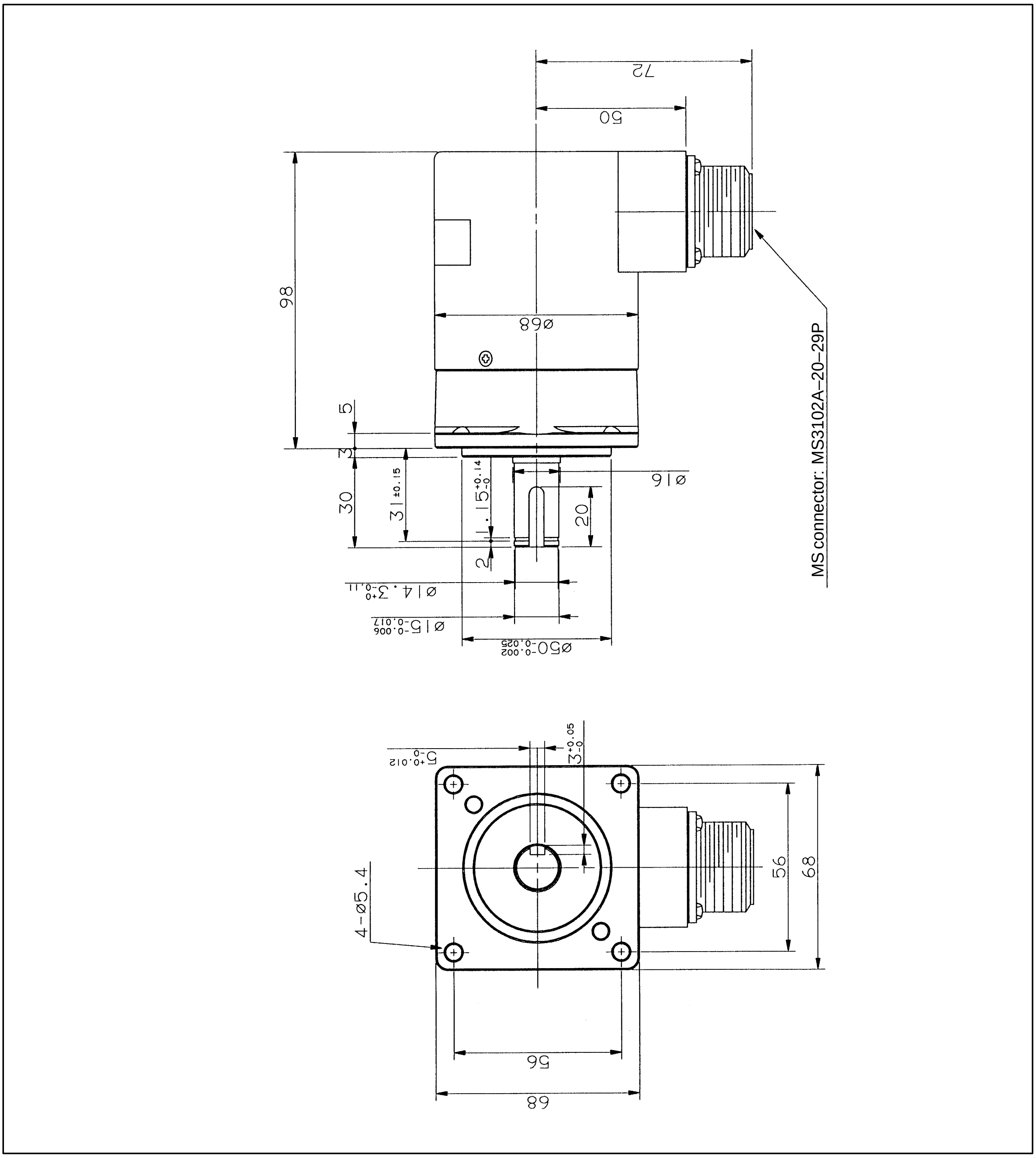


Fig.U17 α position coder
Specification No.: A860–0309–T302 (10000 min^{-1} maximum)

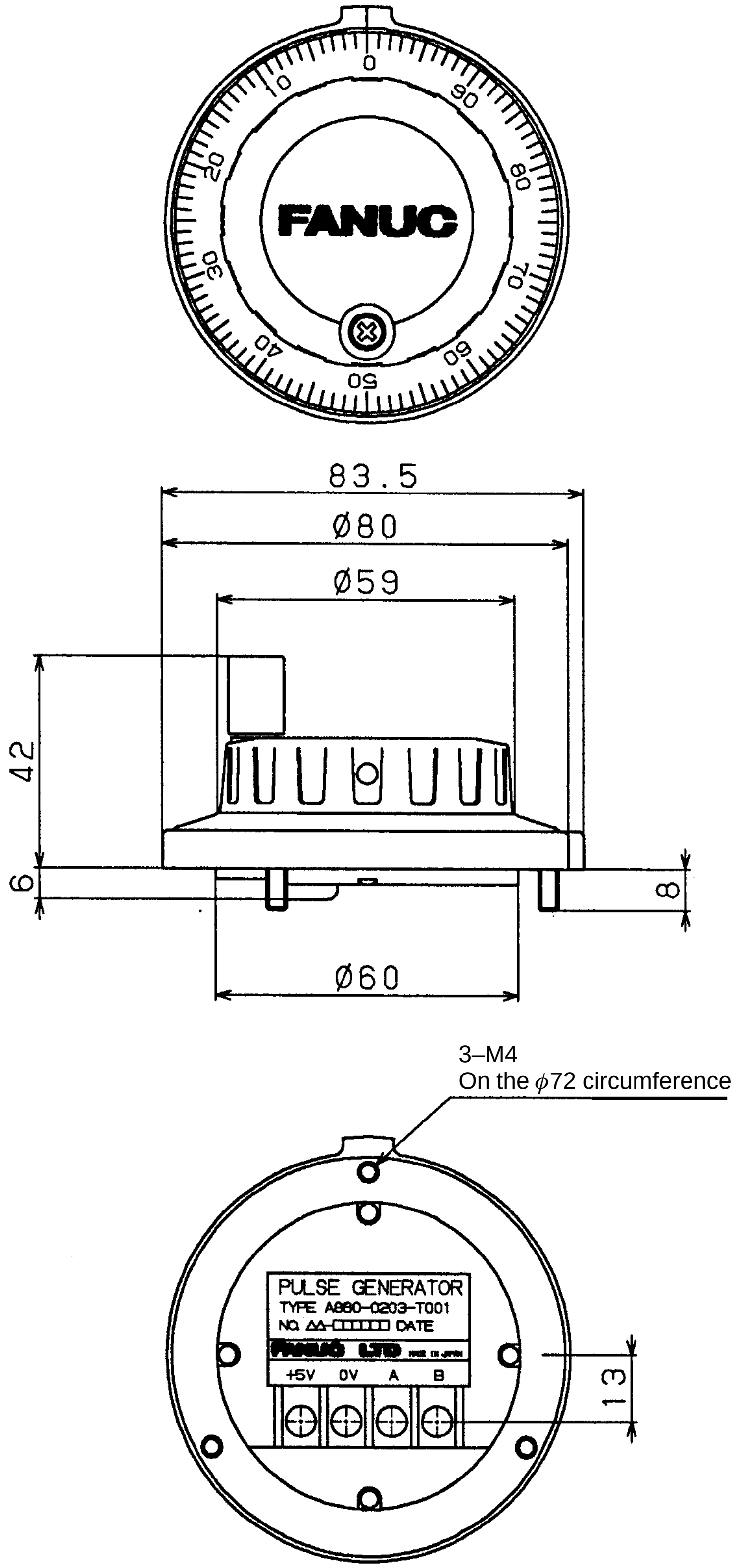
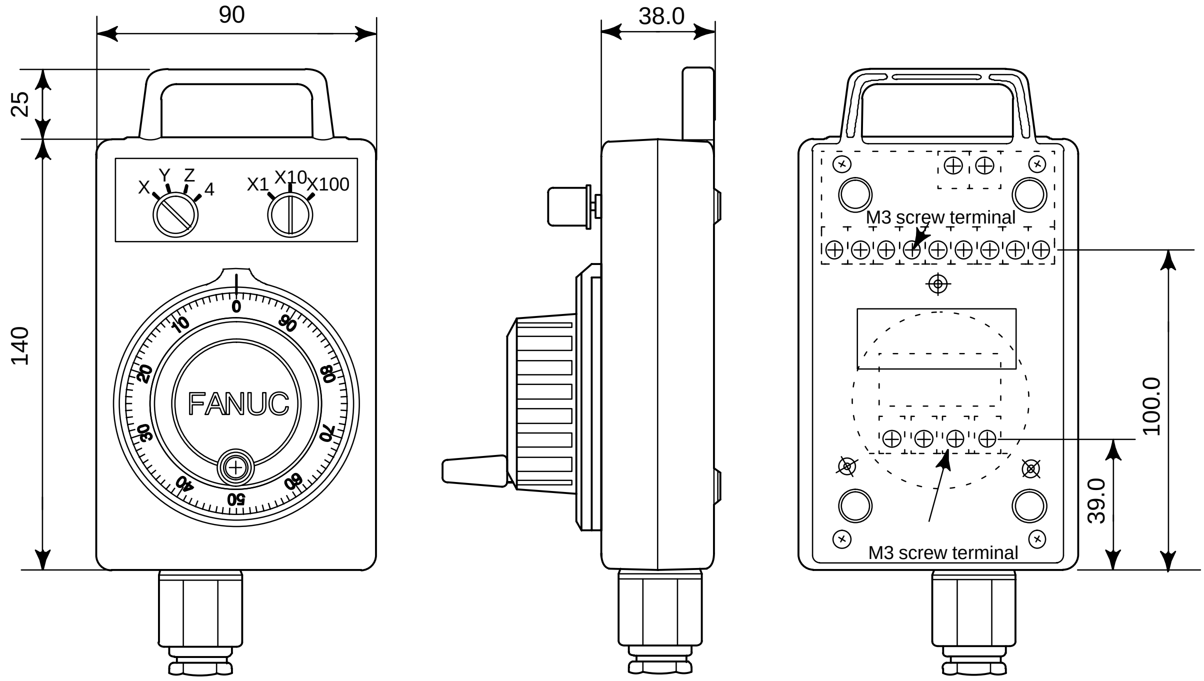


Fig. U24 External dimensions of manual pulse generator
Specification No.: A860–0203–T001

(1) A860–0203–T004 to T009



(2) A860–0203–T010 to T015

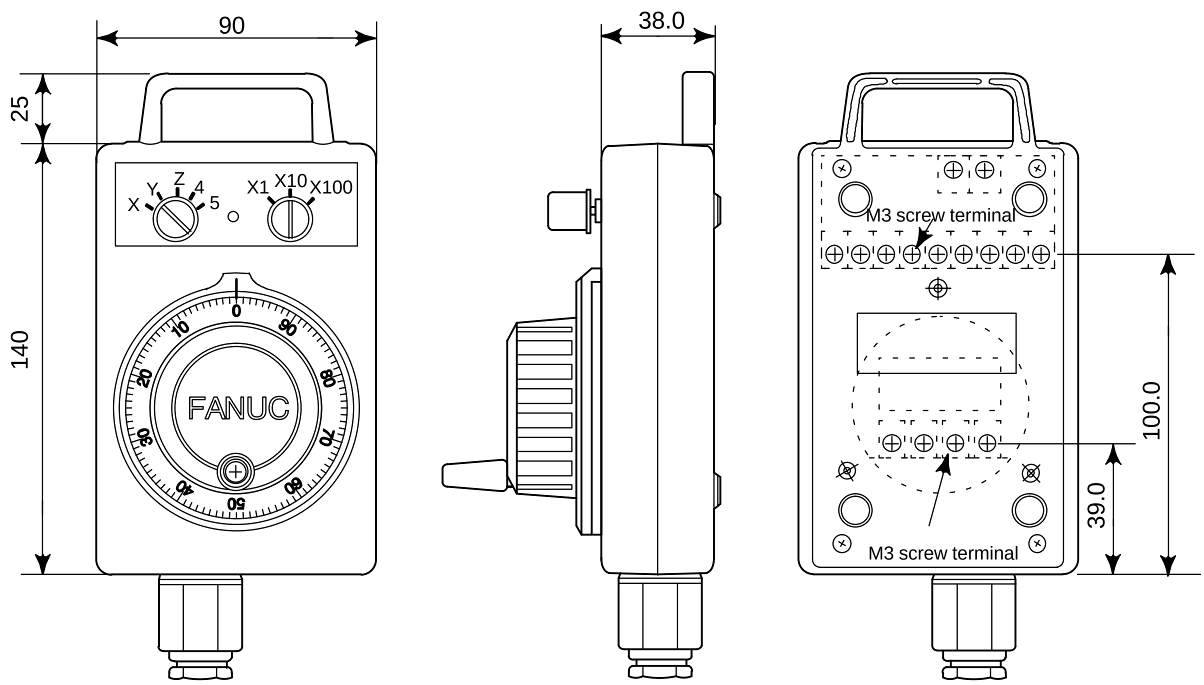
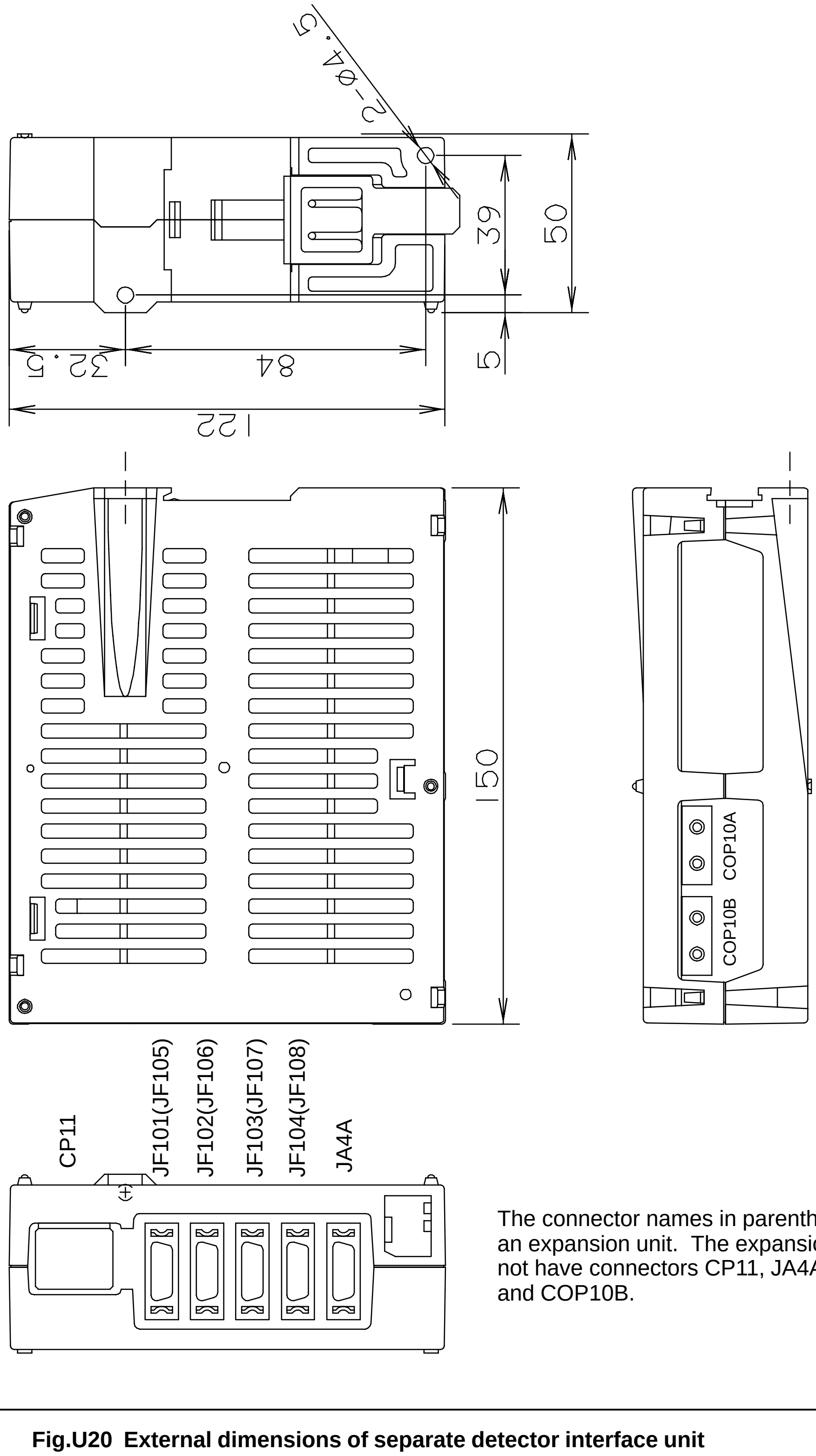


Fig.U19 Pendant type manual pulse generator
Specification No. : A860–0203–T004 to T015



The connector names in parentheses are for an expansion unit. The expansion unit does not have connectors CP11, JA4A, COP10A, and COP10B.

Fig.U20 External dimensions of separate detector interface unit

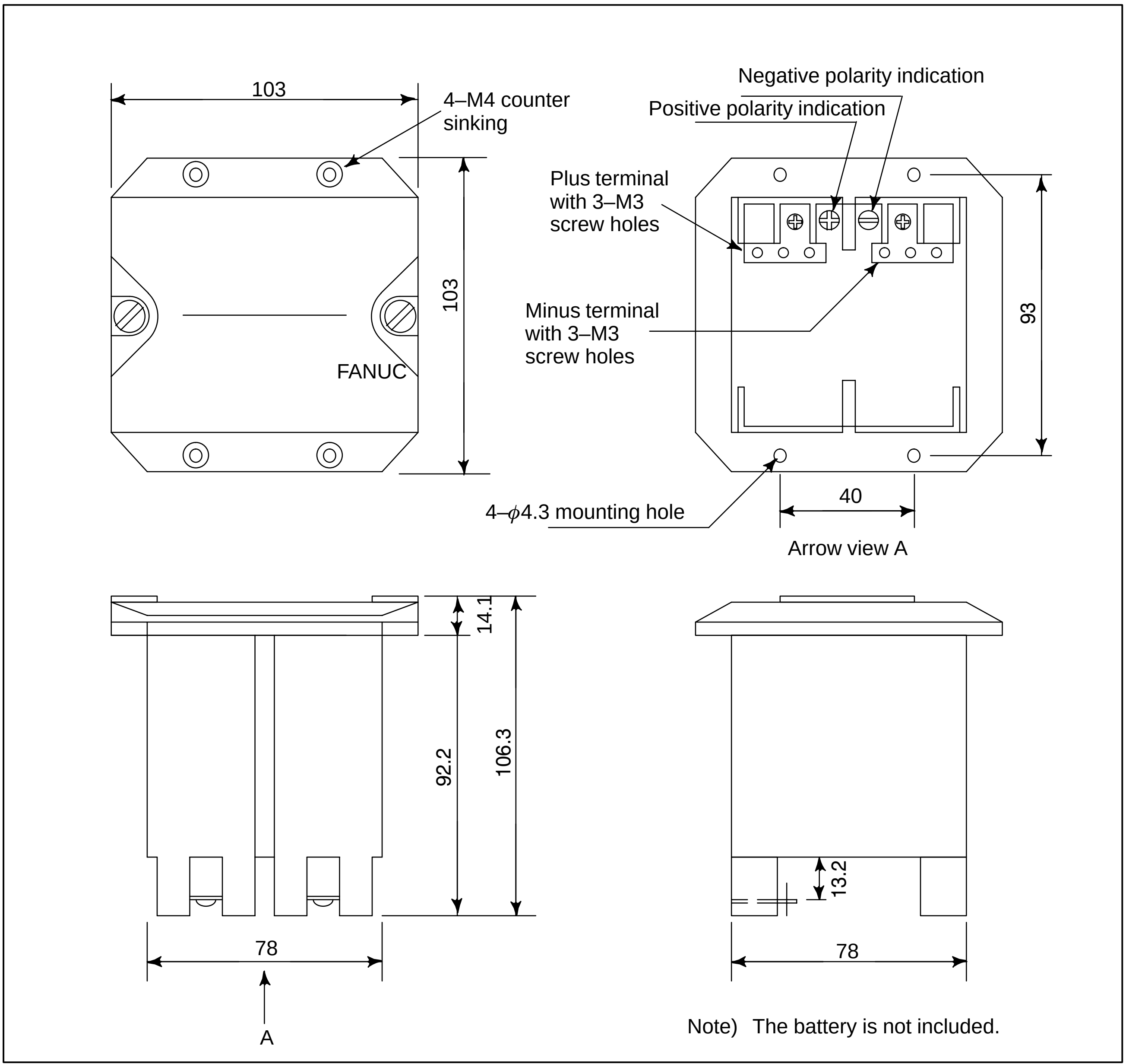


Fig.U21 External dimensions of ABS battery case for separate detector Specification No. : A06B–6050–K060

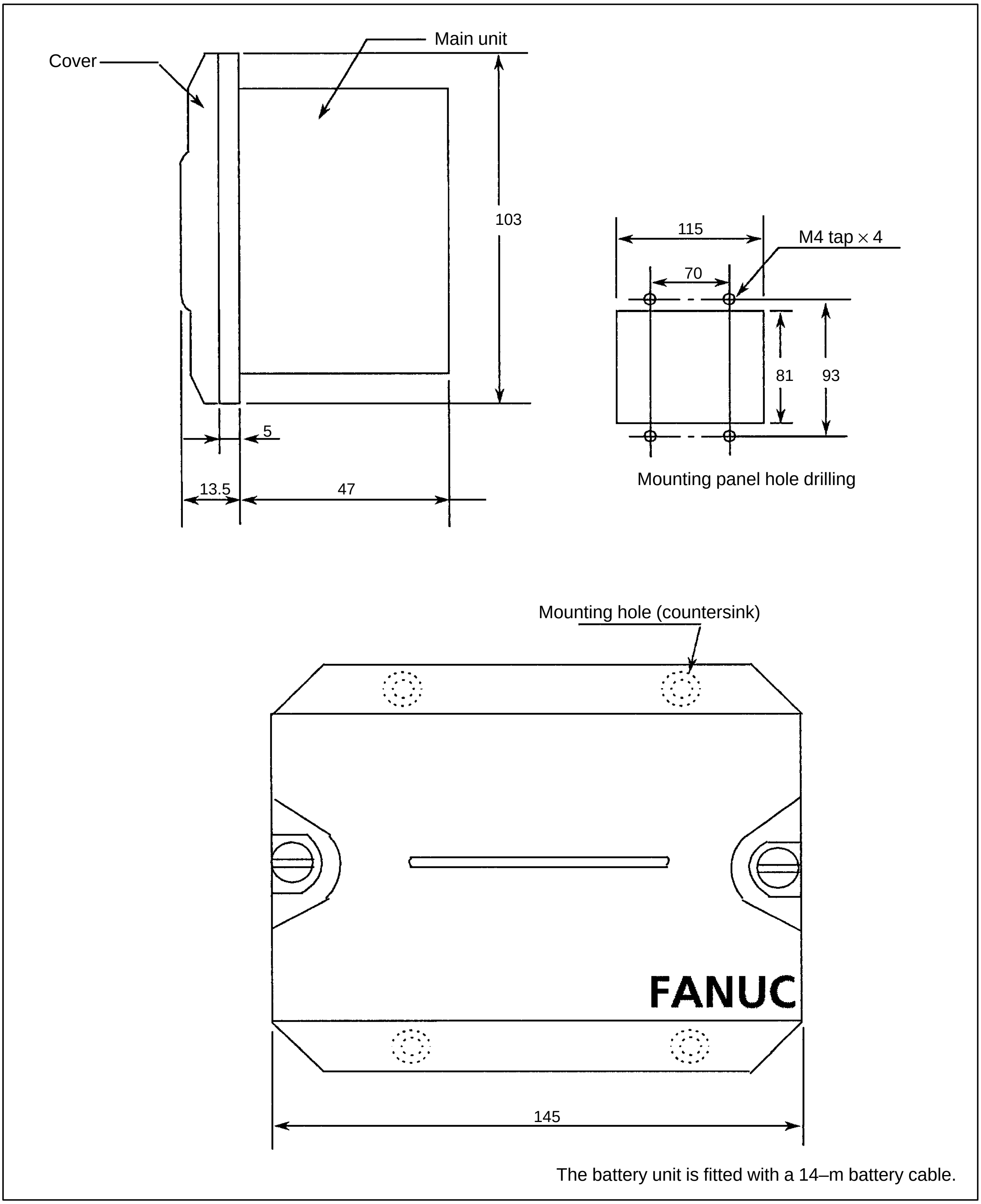


Fig. U22 External dimensions of external CNC battery unit

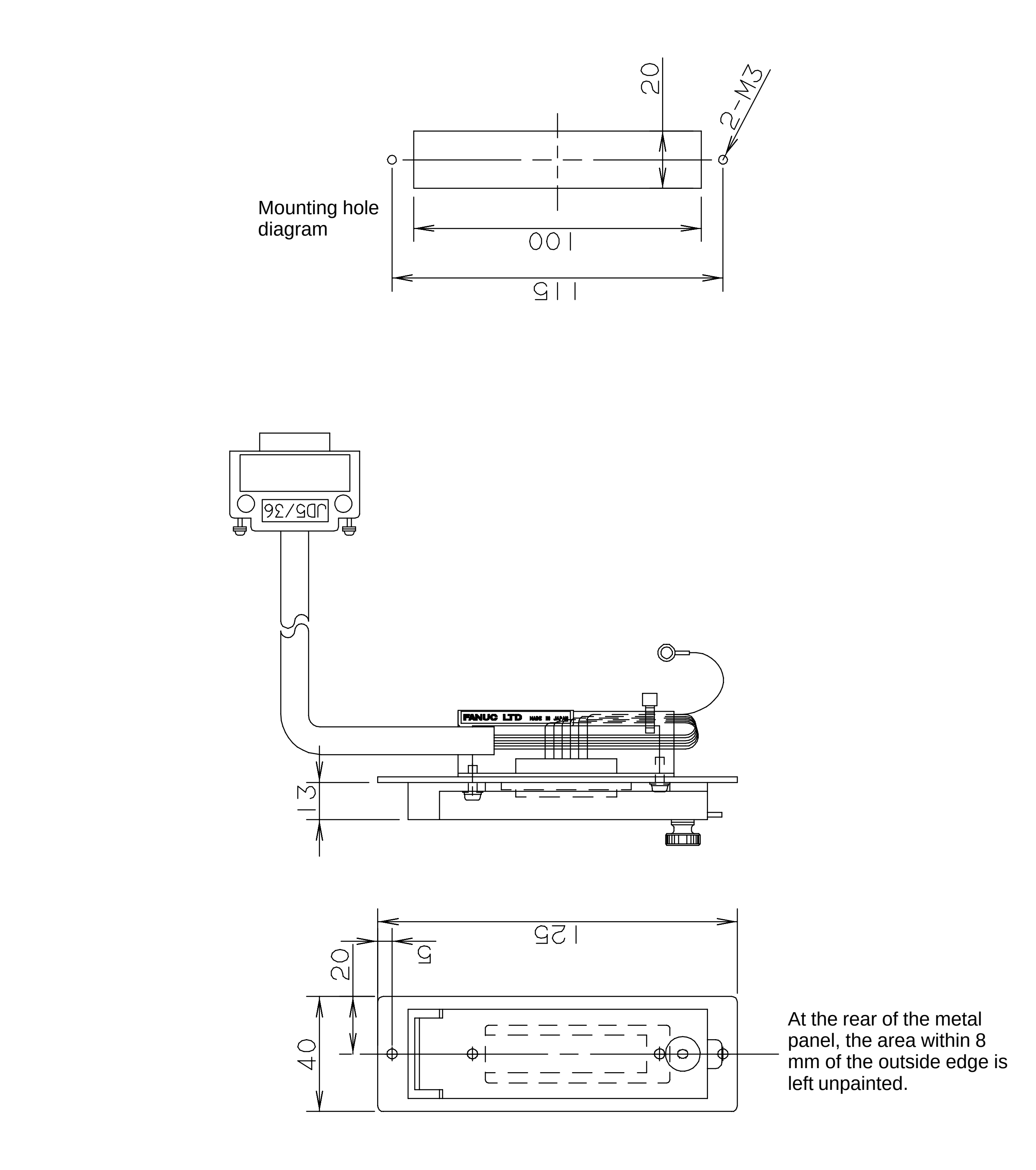
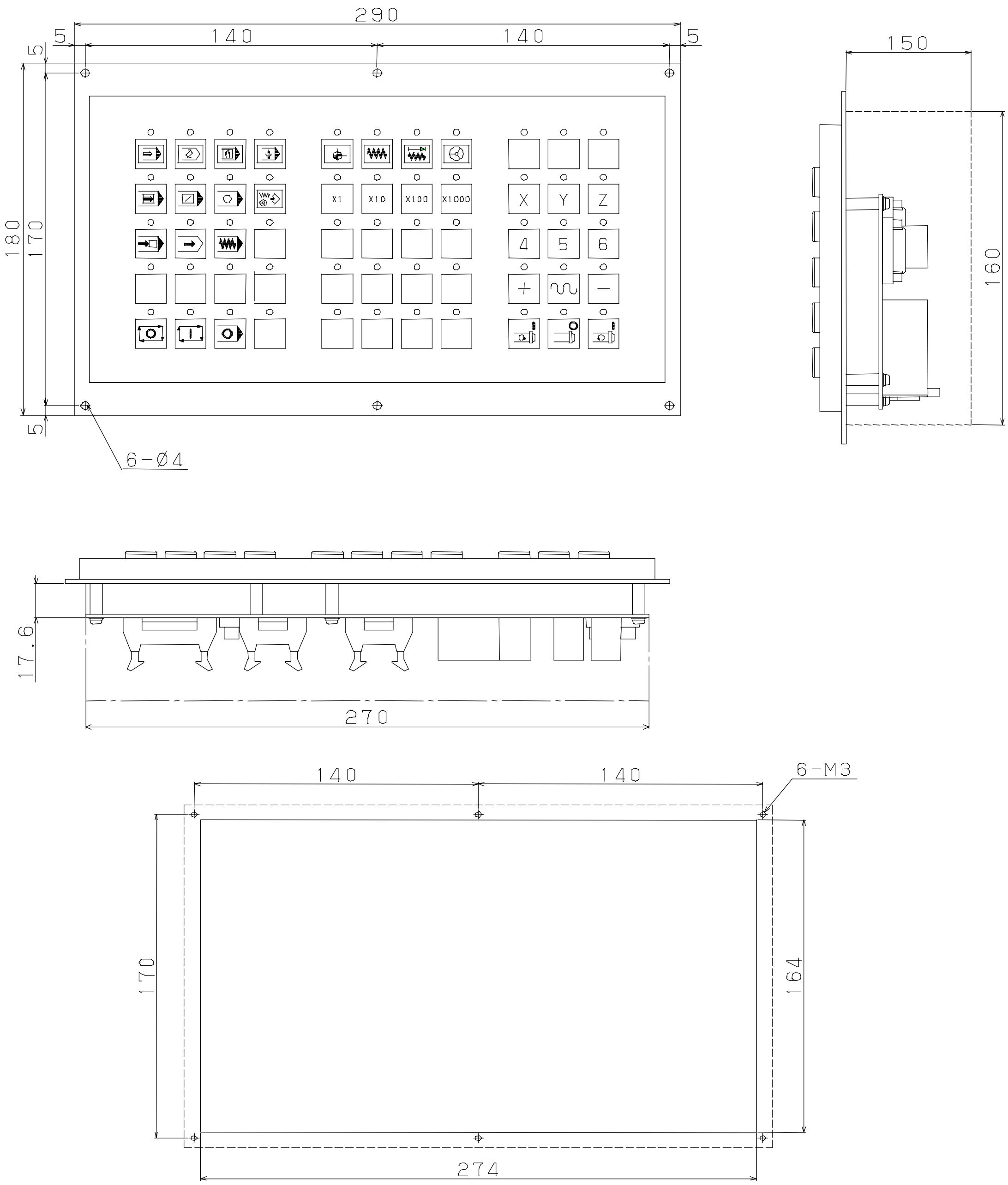


Fig. U24 External dimensions of punch panel (narrow type)



Unit = mm

Weight: 1.6kg

Panel cut drawing

Fig.U25 Machine operator's panel (Main panel B)
Specification No. : A02B–0236–C231

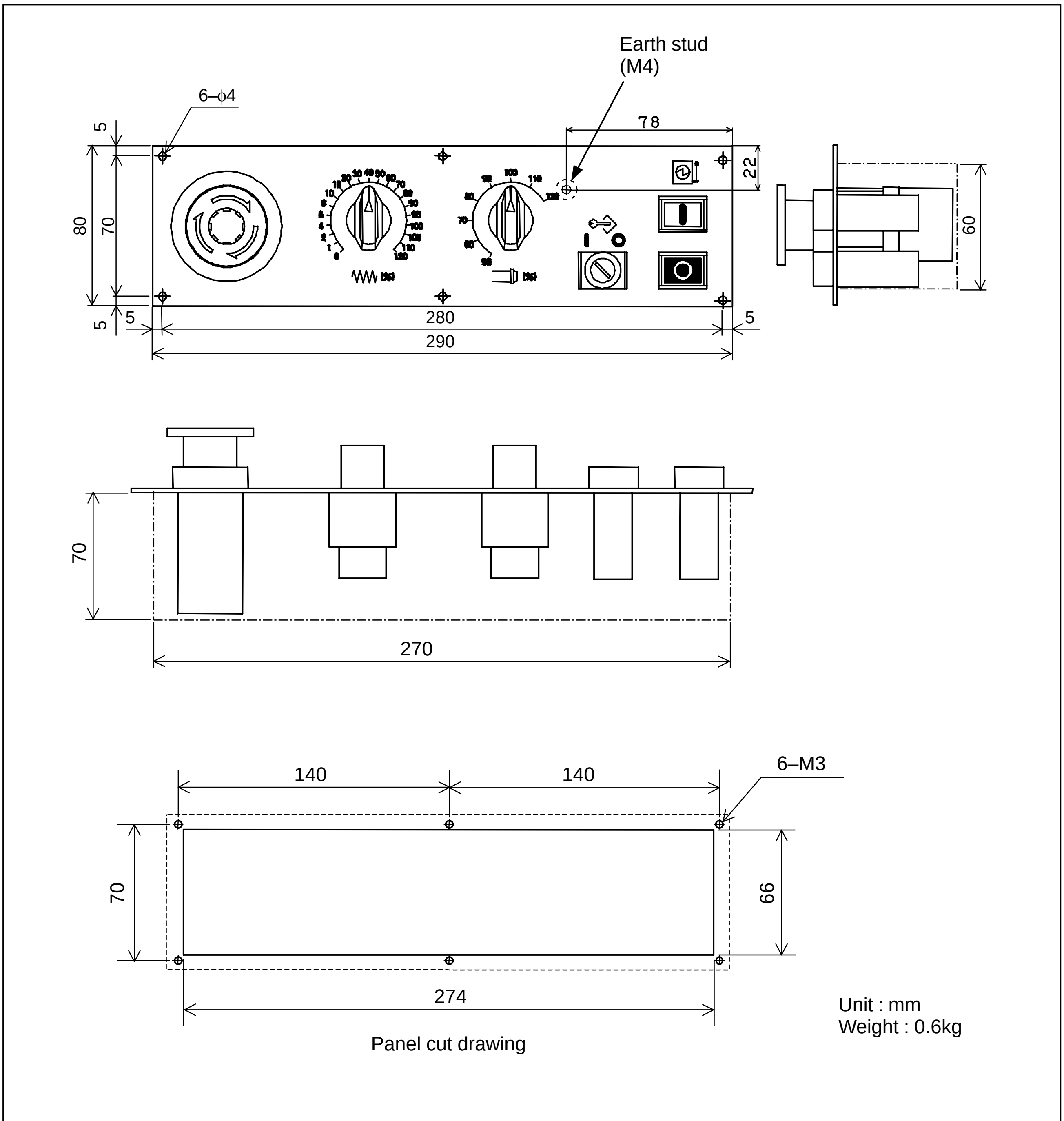


Fig.U26 Machine operator's panel (Sub panel A)
Specification No. : A02B–0236–C232

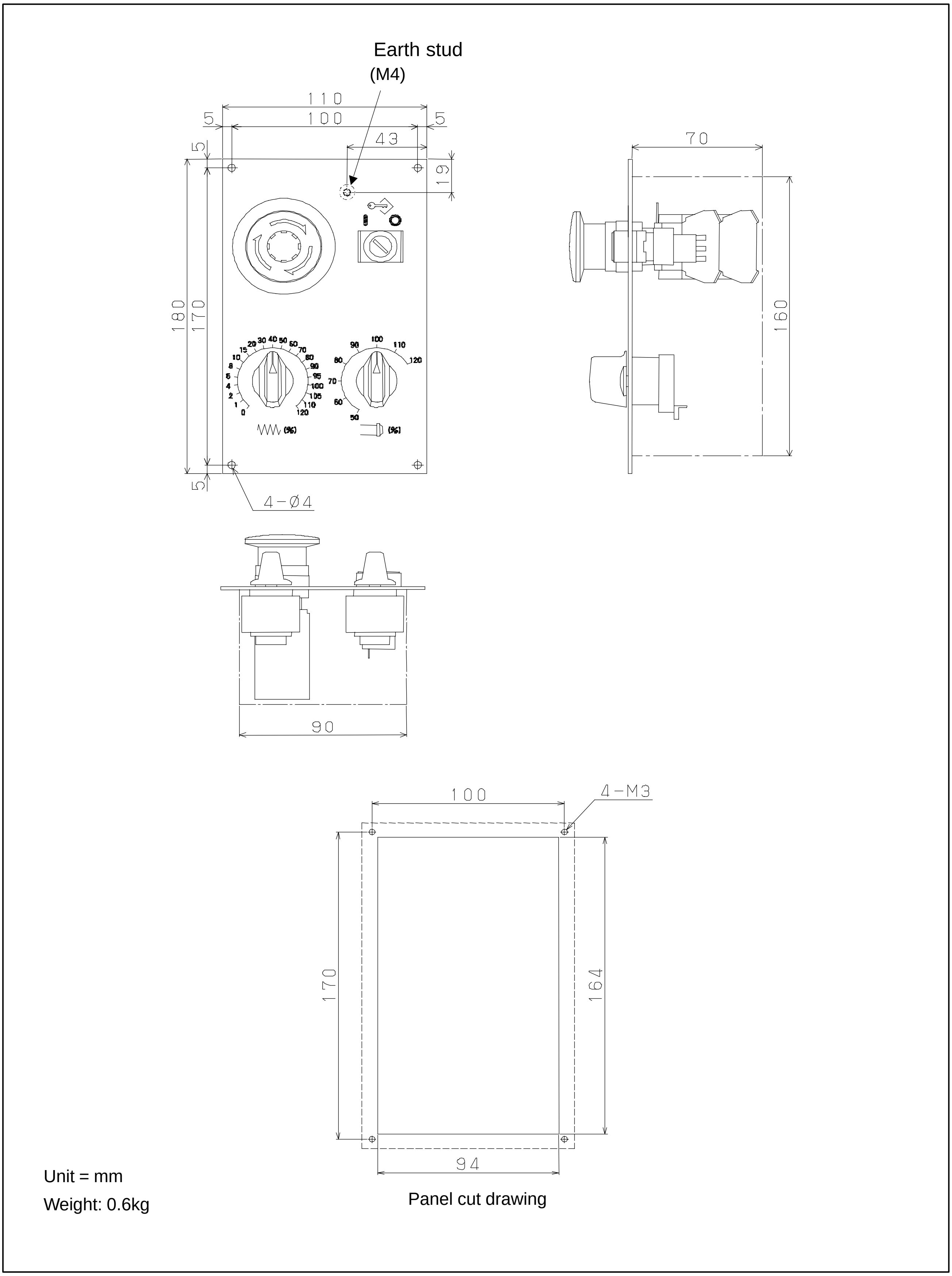


Fig.U27 Machine operator's panel (Sub panel B1)
Specification No. : A02B–0236–C235

Index

[Symbols]

[Numbers]

[A]

[B]

[C]

[D]

[E]

[F]

[G]

[H]

[I]

[J]

[K]

[L]

[M]

[N]

[O]

[P]

[R]

[S]

[T]

[U]

[V]

[W]

Revision Record

FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C DESCRIPTIONS (B–64112EN)

Edition	Date	Contents	Edition	Date	Contents
01	Jun., 2004				