FANUC

GE Fanuc Automation

Computer Numerical Control Products

Series 16*i* / 18*i* / 160*i* / 180*i* – Model A
Series 21*i* / 210*i* – Model A

Descriptions Manual

GFZ-63002EN/01

April 1997

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.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

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 damage 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.

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

II. NC FUNCTION

14. TOOL COMPENSATION FUNCTION 173

21. AXES CONTROL 238

22. FUNCTIONS SPECIFIC TO 2–PATH CONTROL 253

23. MANUAL OPERATION 262

24. AUTOMATIC OPERATION 268

25. PROGRAM TEST FUNCTIONS 278

26. SETTING AND DISPLAY UNIT 280

27. DISPLAYING AND SETTING DATA 290

III. AUTOMATIC PROGRAMMING FUNCTION

APPENDIX

I. GENERAL

1

GENERAL

The FANUC Series 16*i*, 160*i*, 18*i*, 180*i*, 21*i*, and 210*i* are super–compact ultra–thin CNC models with built–in liquid crystal displays. Each CNC unit is a mere 60 mm deep and features, immediately behind the liquid crystal display, a small CNC printed circuit board developed by utilizing state–of–the–art LSI and surface–mount technologies.
Super–compact ultra–thin open CNC models are also available. The CNC printed circuit board incorporates PC functions that are fully compatible with the IBM PC(*).
The amount of cabling in the electrical unit of the machine can be significantly reduced by using a high–speed serial servo bus, which connects the CNC control unit and multiple servo amplifiers by a single optical fiber cable. Another innovation which simplifies the electrical unit of the machine tool is the use of compact distributed I/O modules, which can be separately mounted on the machine operator's panel and control panel.

* IBM PC is a registered trademark of International Business Machines Corporation.

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

Model name	**Abbreviation**	
FANUC Series 16*i*–TA	Series 16*i*–TA	Series 16*i*
FANUC Series 16*i*–MA	Series 16*i*–MA	
FANUC Series 160*i*–TA	Series 160*i*–TA	Series 160*i*
FANUC Series 160*i*–MA	Series 160*i*–MA	
FANUC Series 18*i*–TA	Series 18*i*–TA	Series 18*i*
FANUC Series 18*i*–MA	Series 18*i*–MA	
FANUC Series 180*i*–TA	Series 180*i*–TA	Series 180*i*
FANUC Series 180*i*–MA	Series 180*i*–MA	
FANUC Series 21*i*–TA	Series 21*i*–TA	Series 21*i*
FANUC Series 21*i*–MA	Series 21*i*–MA	
FANUC Series 210*i*–TA	Series 210*i*–TA	Series 210*i*
FANUC Series 210*i*–MA	Series 210*i*–MA	

For ease of understanding, the models may be categorized as follows:
T series: 16*i*–TA, 160*i*–TA, 18*i*–TA, 180*i*–TA, 21*i*–TA, 210*i*–TA
M series: 16*i*–MA, 160*i*–MA, 18*i*–MA, 180*i*–MA, 21*i*–MA, 210*i*–MA

Related manuals

The following table lists the manuals related to the FANUC Series 16*i*, 160*i*, 18*i*, and 180*i*. This manual is indicated by an asterisk(*).

Table 1(a) Manuals Related to the Series 16*i*, 160*i*, 18*i*, and 180*i*

Manual name	Order No.	
Descriptions	B–63002EN	*
Connection Manual (Hardware)	B–63003EN	
Connection Manual (Function)	B–63003EN–1	
Operator's Manual (for Lathe)	B–63004EN	
Operator's Manual (for Machining Center)	B–63014EN	
Maintenance Manual	B–63005EN	
Parameter Manual	B–63010EN	
Macro Compiler/Macro Executor, Programming Manual	B–61803E–1	
FAPT Macro Compiler (for PCs), Programming Manual	B–66102E	
FANUC Super CAP T/Super CAP II T, Operator's Manual	B–62444E–1	
FANUC Super CAP M, Operator's Manual	B–62154E	
FANUC Super CAP M, Programming Manual	B–62153E	
Graphic Conversation I for Lathe, Operator's Manual	B–61804E–1	
Graphic Conversation II for Lathe, Operator's Manual	B–61804E–2	
Graphic Conversation I for Machining Center, Operator's Manual	B–61874–1	
FANUC Symbolic CAP T Basic Module V1, Operator's Manual	B–62824EN	
FANUC Symbolic CAP T C/Y–Axis Module V1, Operator's Manual	B–62824EN–1	
FANUC Symbolic CAP M Basic Module V1, Operator's Manual	B–62984EN	

The following table lists the manuals related to the FANUC Series 21*i* and 210*i*. This manual is indicated by an asterisk (*).

Table 1(b) Manuals Related to the Series 21*i* and 210*i*

Manual name	Order No.	
Descriptions	B–63002EN	*
Connection Manual (Hardware)	B–63083EN	
Connection Manual (Function)	B–63003EN–1	
Operator's Manual (for Lathe)	B–63084EN	
Operator's Manual (for Machining Center)	B–63094EN	
Maintenance Manual	B–63085EN	
Parameter Manual	B–63090EN	
Macro Compiler/Macro Executor, Programming Manual	B–61803E–1	
FAPT Macro Compiler (for PCs), Programming Manual	B–66102E	
FANUC Symbolic CAP T Basic Module V1, Operator's Manual	B–62824EN	
FANUC Symbolic CAP T C/Y–Axis Module V1, Operator's Manual	B–62824EN–1	
FANUC Symbolic CAP M Basic Module V1, Operator's Manual	B–62984EN	

Manuals related to the α–series control motor

Manual related to the α–series control motor

Manual name	Order No.
FANUC AC Servo Motor α series, Descriptions	B–65142E
FANUC AC Servo Motor α series, Parameter Manual	B–65150E
FANUC AC Spindle Motor α series, Descriptions	B–65152E
FANUC AC Spindle Motor α series, Parameter Manual	B–65160E
FANUC Control Motor Amplifier α series, Descriptions	B–65162E
FANUC Control Motor α series, Maintenance Manual	B–65165E

2

LIST OF SPECIFICATIONS

○ : Standard ● : Standard option ☆ : Option
✻ : Function included in another option

Note) The use of some combinations of options is restricted.

Item		**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
			MA	**TA**	**MA**	**TA**	**MA**	**TA**
Axis control								
Total controlled axes (Machine–controlled axes plus loader–controlled axes) (Machine–controlled axes including Cs–axis)		Up to 12 axes ((8 machine axes) × (1 path) + (4 loader axes))	☆	☆	—	—	—	—
		Up to 18 axes ((7 machine axes) × (2 paths) + (4 loader axes))	☆	☆	—	—	—	—
		Up to 10 axes ((6 machine axes) × (1 path) + (4 loader axes))	—	—	☆	☆	—	—
		Up to 14 axes ((5 machine axes) × (2 paths) + (4 loader axes))	—	—	—	☆	—	—
		Up to 4 axes (4 machine axes)	—	—	—	—	☆	☆
		Up to 8 axes ((4 machine axes) + (4 loader axes))	—	—	—	—	—	☆
Machine–controlled axes	Controlled paths	1–path	○	○	○	○	○	○
		2–paths	☆	☆	—	☆	—	—
	Controlled axes per path	2 axes	—	○	—	○	—	○
		3 axes	○	—	○	—	○	—
	Simultaneously controlled axes per path	2 axes	○	○	○	○	○	○
	Controlled axis expansion (total)	Up to 8 axes (single–path system) (including Cs–axis)	☆	☆	—	—	—	—
		Up to 7 axes (2–path system) ((6 feed axes) + (Cs–axis))	☆	☆	—	—	—	—
		Up to 6 axes (single–path system) (including Cs–axis)	—	—	☆	☆	—	—
		Up to 5 axes (2–path system) ((4 feed axes) + (Cs–axis))	—	—	—	☆	—	—
		Up to 4 axes (single–path system) (including Cs–axis)	—	—	—	—	☆	☆
	Simultaneously controlled axis expansion (total)	Up to 6 axes	☆	☆	—	—	—	—
		Up to 4 axes	—	—	☆	☆	☆	☆
	Axis control by PMC	Up to 4 axes simultaneously for each path (not supported for the Cs–axis)	☆	☆	☆	☆	☆	☆
	Cs contour control	1 axis per path	☆	—	☆	—	☆	☆
		2 axes per path	—	☆	—	☆	—	—

Item		**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
			MA	**TA**	**MA**	**TA**	**MA**	**TA**
Loader controlled axes	Controlled paths	1–path	☆	☆	☆	☆	—	☆
	Controlled axes	Up to 4 axes	☆	☆	☆	☆	—	☆
	Simultaneously controlled axes	Up to 4 axes	☆	☆	☆	☆	—	☆
	Axis control by PMC	Up to 4 axes	☆	☆	☆	☆	—	☆
Axis name		3 basic axes: X, Y, Z; Additional axes: U, V, W, A, B, or C	○	—	○	—	○	—
		With G code system A 2 basic axes: X, Z; Additional axes: Y, A, B, or C	—	○	—	○	—	○
		With G code system B or C 2 basic axes: X, Z; Additional axes: Y, U, V, W, A, B, or C	—	✻	—	✻	—	✻
Axis recomposition		For 2–path system only	—	☆	—	☆	—	—
Simple synchronous control		1 set	—	☆	—	☆	☆	☆
		3 sets	—	—	☆	—	—	—
		4 sets	☆	—	—	—	—	—
Slanted axis control			☆	☆	☆	☆	—	☆
Slanted axis control for arbitrary axis			☆	☆	☆	☆	—	—
B–axis control			—	☆	—	☆	—	☆
Tandem control			☆	☆	☆	☆	—	—
Torque control		PMC axis control required	✻	✻	✻	✻	✻	✻
Controlled axis detach			☆	☆	☆	☆	☆	☆
Chopping			☆	—	☆	—	—	—
Hobbing machine function			☆	—	☆	—	—	—
Simple electric gearbox			☆	—	☆	—	—	—
Minimum input increment		0.001mm, 0.001deg, 0.0001 inch	○	○	○	○	○	○
Increment system 1/10		0.0001mm, 0.0001deg, 0.00001 inch	☆	☆	☆	☆	☆	☆
Flexible feed gear		Optional DMR	○	○	○	○	○	○
Learning control			☆	☆	☆	☆	—	—
Preview repetitive control			☆	☆	☆	☆	—	—
Dual position feedback			☆	☆	☆	☆	☆	☆
Fine acceleration/deceleration			○	○	○	○	○	○
HRV control			○	○	○	○	○	○
Inch/metric switching			☆	☆	☆	☆	☆	☆
Interlock		All axes/each axis/each axial direction/block start block/cutting block start	○	○	○	○	○	○
Machine lock		All axes/each axis	○	○	○	○	○	○
Emergency stop			○	○	○	○	○	○
Overtravel			○	○	○	○	○	○
Stored stroke check 1			○	○	○	○	○	○
External stroke limit setting			☆	—	☆	—	☆	—
Stored stroke check 2			☆	—	☆	—	☆	—
Stored stroke check 3			☆	—	☆	—	—	—
Stored stroke check 2, 3			—	☆	—	☆	—	☆
Stroke limit check before travel			☆	☆	☆	☆	—	—
Chuck/tailstock barrier			—	☆	—	☆	—	☆
Mirror image		Each axis	○	○	○	○	○	○

Item	**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
Follow–up		○	○	○	○	○	○
Servo–off/mechanical handle feed		○	○	○	○	○	○
Chamfering on/off		—	○	—	○	—	○
Backlash compensation		○	○	○	○	○	○
Separate backlash compensation for rapid traverse and cutting feed		○	○	○	○	○	○
Stored pitch error compensation		☆	☆	☆	☆	☆	☆
Gradient compensation		☆	☆	☆	☆	—	—
Straightness deviation compensation		☆	☆	☆	☆	—	—
Position switch		☆	☆	☆	☆	☆	☆
Tool post interference check (between 2 paths)	For 2–path system only *1	☆	☆	—	☆	—	—
Abnormal load detection		☆	☆	☆	☆	☆	☆
Fine torque sensing	*1	☆	☆	☆	☆	—	—

Operation

Automatic operation (memory)		○	○	○	○	○	○
DNC operation	Reader/punch interface required	✻	✻	✻	✻	✻	✻
MDI operation		○	○	○	○	○	○
Scheduling function	For single–path system only *1	✻	✻	✻	✻	✻	✻
Program number search		○	○	○	○	○	○
Sequence number search		○	○	○	○	○	○
Sequence number collation stop		☆	☆	☆	☆	☆	☆
Program restart		☆	☆	☆	☆	☆	☆
Tool retract and return		☆	☆	☆	☆	—	—
Manual intervention and return		○	○	○	○	○	○
Machining return and restart		☆	—	☆	—	—	—
Rigid tapping return		☆	—	☆	—	☆	—
Buffer register		○	○	○	○	○	○
Dry run		○	○	○	○	○	○
Single block		○	○	○	○	○	○
Jog feed		○	○	○	○	○	○
Manual reference position return		○	○	○	○	○	○
Reference position return setting without dog		○	○	○	○	○	○
Butt–type reference position return setting		☆	☆	☆	☆	☆	☆
Reference position shift		☆	☆	☆	☆	—	—
Manual handle feed	1 unit per path	☆	☆	☆	☆	☆	☆
	2 units	—	☆	—	☆	—	☆
	2 or 3 units	☆	—	☆	—	☆	—
Manual handle feed magnification	×1, ×10, ×m, ×n m: 0 to 127; n: 0 to 1000	✻	✻	✻	✻	✻	✻
Handle feed in tool axis direction		☆	—	☆	—	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
Handle feed in tool axis direction B	(Tool axis direction) + (Vertical direction)	☆	—	☆	—	—	—
Manual handle interrupt		☆	☆	☆	☆	☆	☆
Incremental feed	×1, ×10, ×100, ×1000	○	○	○	○	○	○
Jogging/handle feed mode		○	○	○	○	○	○
Manual numeric command	*1	☆	☆	☆	☆	—	—
Manual linear/circular interpolation	For single–path system only	☆	☆	☆	☆	—	—

Interpolation functions

Positioning	G00 (Linear interpolation type positioning enabled)	○	○	○	○	○	○
Unidirectional positioning	G60	☆	—	☆	—	☆	—
Exact stop mode	G61	○	—	○	—	○	—
Exact stop	G09	○	—	○	—	○	—
Linear interpolation		○	○	○	○	○	○
Circular interpolation	Supported for multiple quadrants	○	○	○	○	○	○
Exponential interpolation		☆	—	☆	—	—	—
Dwell	For a specified number of seconds or rotations (To specify dwell for a specified number of rotations for MA, the threading/synchronous feed function is necessary.)	○	○	○	○	○	○
Polar coordinate interpolation		☆	☆	☆	☆	—	☆
Cylindrical interpolation		☆	☆	☆	☆	☆	☆
Helical interpolation	(Circular interpolation) + (Linear interpolation for up to 2 axes)	☆	☆	☆	☆	☆	—
Helical interpolation B	(Circular interpolation) + (Linear interpolation for up to 4 axes)	☆	—	—	—	—	—
Involute interpolation		☆	—	☆	—	—	—
Hypothetical axis interpolation		☆	☆	☆	☆	—	—
Spiral/conical interpolation		☆	—	☆	—	—	—
Smooth interpolation	High–precision contour control function required	☆	—	—	—	—	—
Threading/synchronous feed		☆	○	☆	○	☆	○
Multi–start threading		—	○	—	○	—	○
Threading retract		—	☆	—	☆	—	☆
Continuous threading		—	☆	—	☆	—	☆
Variable–lead threading		—	☆	—	☆	—	☆
Circular threading		—	☆	—	☆	—	—
Polygon turning		—	☆	—	☆	—	☆
Polygon turning between spindles		—	☆	—	☆	—	—
Skip	G31	○	○	○	○	○	○
High–speed skip		☆	☆	☆	☆	☆	☆
Continuous high–speed skip		☆	—	☆	—	—	—
Multi–step skip		☆	☆	☆	☆	—	☆
Torque–limit skip		—	○	—	○	—	○
Reference position return	G28	○	○	○	○	○	○

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	TA	MA	TA	MA	TA
Reference position return check	G27	○	○	○	○	○	○
2nd reference position return		○	○	○	○	○	○
3rd/4th reference position return		☆	☆	☆	☆	☆	☆
Floating reference position return		☆	☆	☆	☆	—	—
Normal–direction control		☆	—	☆	—	☆	—
Gentle–curve normal–direction control		☆	—	☆	—	—	—
Continuous dressing	For grinder	☆	—	☆	—	—	—
In–feed control	For grinder	☆	—	☆	—	—	—
Balance cut	For 2–path system only	—	☆	—	☆	—	—
Index table indexing		☆	—	☆	—	☆	—
High–speed cycle machining	For single–path system only	☆	☆	☆	☆	—	—
High–speed cycle machining retract		☆	☆	☆	☆	—	—
High–speed linear interpolation		☆	☆	☆	☆	—	—

Feed functions

Rapid traverse	Up to 240 m/min (1 μm)	○	○	○	○	○	○
	Up to 100 m/min (0.1 μm)	✻	✻	✻	✻	✻	✻
Rapid traverse override	Fo, 25, 50, 100%	○	○	○	○	○	○
Feed per minute		○	○	○	○	○	○
Feed per rotation	Threading/synchronous feed function required for the M series	✻	○	✻	○	✻	○
Feed per rotation without position coder		—	○	—	○	—	—
Constant tangential speed control		○	○	○	○	○	○
Cutting feedrate clamp		○	○	○	○	○	○
Automatic acceleration/deceleration	Rapid traverse: Linear Cutting feed: Exponential	○	○	○	○	○	○
Rapid traverse bell–shaped acceleration/deceleration		☆	☆	☆	☆	☆	☆
Positioning by optimum acceleration		☆	☆	☆	☆	—	—
Linear acceleration/deceleration after cutting feed interpolation		☆	☆	☆	☆	☆	☆
Bell–shaped acceleration/deceleration after cutting feed interpolation		☆	☆	☆	☆	☆	☆
Linear acceleration/deceleration before cutting feed interpolation	Feed per minute only	☆	☆	☆	☆	—	—
Feedrate override	0 to 254%	○	○	○	○	○	○
2nd feedrate override	0 to 254%	☆	☆	☆	☆	—	—
Feed by F with one digit		☆	—	☆	—	☆	—
Inverse time feed		☆	—	☆	—	—	—
Jog override	0 to 655.34%	○	○	○	○	○	○
Override cancel		○	○	○	○	○	○
Manual feed per rotation		—	○	—	○	—	○
External deceleration		☆	☆	☆	☆	☆	☆
Feed stop		☆	☆	☆	☆	—	—
Look–ahead control		☆	—	☆	—	☆	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
Simple high–precision contour control		☆	—	☆	—	—	—
Bell–shaped acceleration/deceleration before look–ahead interpolation	Simple high–precision contour control required	☆	—	☆	—	—	—
High–precision contour control	64–bit RISC (single–path system only)	☆	—	—	—	—	—
NURBS interpolation	High–precision contour control required	☆	—	—	—	—	—

Program input

Tape code	Automatic recognition of EIA RS244 and ISO 840	○	○	○	○	○	○
Label skip		○	○	○	○	○	○
Parity check	Horizontal parity, vertical parity	○	○	○	○	○	○
Control in/out		○	○	○	○	○	○
Optional block skip	1 block	○	○	○	○	○	○
	9 blocks	☆	☆	☆	☆	☆	☆
Maximum value	± with 8 digits	○	○	○	○	○	○
Program number	O with 4 digits	○	○	○	○	○	○
	O with 8 digits *1	☆	☆	☆	☆	—	—
Sequence number	N with 5 digits	○	○	○	○	○	○
Absolute/incremental programming	Combined programming in a single block allowed	○	○	○	○	○	○
Decimal point input, pocket calculator type decimal point input	*1	○	○	○	○	○	○
Input unit (10 times)		○	○	○	○	○	○
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, G53, G54 to G59	☆	☆	☆	☆	☆	☆
Workpiece coordinate system preset		☆	☆	☆	☆	☆	☆
Addition of workpiece coordinate systems	48 sets	☆	—	☆	—	☆	—
	300 sets	☆	—	☆	—	—	—
Direct input of measured offset from workpiece origin		✻	✻	✻	✻	✻	✻
Manual absolute on/off		○	○	○	○	○	○
Direct drawing dimension programming		—	☆	—	☆	—	☆
G code system	‘A	—	○	—	○	—	○
	B/C	—	☆	—	☆	—	☆
Chamfering/corner rounding		—	☆	—	☆	—	☆

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	TA	MA	TA	MA	TA
Optional–angle chamfering/corner rounding		☆	—	☆	—	☆	—
Programmable data input	G10	☆	☆	☆	☆	☆	☆
Subprogram call	4 levels of nesting	○	○	○	○	○	○
Custom macro B		☆	☆	☆	☆	☆	☆
Addition to custom macro common variables	#100 to #199, #500 to #999	☆	☆	☆	☆	☆	☆
Macro variables common to 2 paths	For 2–path system	✻	✻	—	✻	—	—
Pattern data input	*1	☆	☆	☆	☆	☆	☆
Interrupt–type custom macro		☆	☆	☆	☆	☆	☆
Canned cycle		—	○	—	○	—	○
Multiple repetitive canned cycle		—	☆	—	☆	—	☆
Multiple repetitive canned cycle II	Pocket profile	—	☆	—	☆	—	☆
Canned cycle for drilling		☆	☆	☆	☆	☆	☆
Small–diameter peck drilling cycle		☆	—	☆	—	☆	—
Canned cycle for grinding	For grinder	☆	☆	☆	☆	—	—
Arc radius R programming		○	○	○	○	○	○
Arc radius R programming with 9 digits		—	☆	—	☆	—	—
Mirror image of facing tool posts		—	☆	—	☆	—	☆
Automatic corner override		☆	☆	☆	☆	☆	—
Automatic corner deceleration		☆	—	☆	—	—	—
Feedrate clamp by arc radius		☆	—	☆	—	☆	—
Scaling		☆	—	☆	—	☆	—
Coordinate system rotation		☆	☆	☆	☆	☆	—
Three–dimensional coordinate conversion		☆	☆	☆	☆	—	—
Programmable mirror image		☆	—	☆	—	☆	—
Figure copy		☆	—	☆	—	—	—
Retrace		☆	—	☆	—	—	—
Series 15 tape format		☆	☆	☆	☆	—	—
Series 10/11 tape format		—	—	—	—	☆	☆
Conversational programming	For single–path system only *1	☆	☆	☆	☆	—	—
Macro executor	Up to 1MB *1	—	—	—	—	☆	☆
	Up to 4MB *1	☆	☆	☆	☆	—	—
C macro executor	Up to 4MB *1	☆	☆	☆	☆	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	TA	MA	TA	MA	TA

Graphic conversation functions for machining center

Super CAP II M	*1	☆	—	☆	—	—	—
Super CAP M	*1	☆	—	☆	—	—	—
NC format output	*1	☆	—	☆	—	—	—
Conversational C programming	*1	☆	—	☆	—	—	—
Contour figure repetition	*1	☆	—	☆	—	—	—
Background drawing	*1	☆	—	☆	—	—	—
U–axis conversational function	*1	☆	—	☆	—	—	—
Symbolic CAP M	*2	☆	—	☆	—	☆	—
CAP I	*1	☆	—	☆	—	—	—

Graphic conversation functions for lathe

Super CAP II T	For single–path lathe For 2–path lathe *1	—	☆	—	☆	—	—
Super CAP T	For single–path lathe For 2–path lathe *1	—	☆	—	☆	—	—
NC format output	*1	—	☆	—	☆	—	—
C–axis conversational programming	*1	—	☆	—	☆	—	—
Y–axis conversational programming	*1	—	☆	—	☆	—	—
4000 P–code macro variables	*1	—	☆	—	☆	—	—
Back machining by subspindle	*1	—	☆	—	☆	—	—
Chuck data expansion	60 types *1	—	☆	—	☆	—	—
Tool/chuck/workpiece interference check	*1	—	☆	—	☆	—	—
T code/offset set expansion	*1	—	☆	—	☆	—	—
CAP I	For single–path lathe *1	—	☆	—	☆	—	—
CAP II	For single–path lathe For 2–path lathe *1	—	☆	—	☆	—	—
Auxiliary machining	*1	—	☆	—	☆	—	—
Automatic tool determination	*1	—	☆	—	☆	—	—
Automatic tool determination B	*1	—	☆	—	☆	—	—
Animated simulation	*1	—	☆	—	☆	—	—
Animated simulation for vertical lathe	*1	—	☆	—	☆	—	—
C–axis FAPT	*1	—	☆	—	☆	—	—
Y–axis FAPT	*1	—	☆	—	☆	—	—
Back machining	*1	—	☆	—	☆	—	—
Balance cut FAPT	For 2–path lathe only *1	—	☆	—	☆	—	—
Conversational screen display language expansion	*1	—	☆	—	☆	—	—
Sub–memory addition	*1	—	☆	—	☆	—	—
File name storage on Floppy Cassette	*1	—	☆	—	☆	—	—
Basic module of Symbolic CAP T	For single–path lathe *2	—	☆	—	☆	—	☆
C/Y–axis module of Symbolic CAP T	*2	—	☆	—	☆	—	☆
2–path control module of Symbolic CAP T	*2	—	☆	—	☆	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
Miscellaneous/spindle functions							
Miscellaneous function	M with 8 digits	○	○	○	○	○	○
Second auxiliary function	B with 8 digits	☆	☆	☆	☆	☆	☆
Miscellaneous function lock		○	○	○	○	○	○
High–speed M/S/T/B interface		○	○	○	○	○	○
Wait	For 2–path lathe only	○	○	—	○	—	—
Multiple miscellaneous–function commands	3 pieces	○	○	○	○	○	○
M code group check		☆	☆	☆	☆	—	—
Spindle function	S with 5 digits, binary output	○	○	○	○	○	○
Spindle serial output	S with 5 digits, serial output	☆	☆	☆	☆	☆	☆
3–spindle serial output units	For single–path system only	—	—	☆	☆	—	—
3–/4–spindle serial output units	For single–path system only	☆	☆	—	—	—	—
Spindle analog output	S with 5 digits, analog output	☆	☆	☆	☆	☆	☆
Constant surface speed control		☆	☆	☆	☆	☆	☆
Constant surface speed control without position coder		—	☆	—	☆	—	—
Spindle override	0% to 254%	✻	✻	✻	✻	✻	✻
Actual spindle speed output		—	☆	—	☆	—	☆
Spindle speed fluctuation detection		☆	☆	☆	☆	—	☆
1st spindle orientation		☆	☆	☆	☆	☆	☆
1st spindle output switching		☆	☆	☆	☆	☆	☆
2nd spindle orientation		☆	☆	☆	☆	☆	☆
2nd spindle output switching		☆	☆	☆	☆	☆	☆
3rd spindle orientation		—	—	☆	☆	—	—
3rd spindle output switching		—	—	☆	☆	—	—
3rd/4th spindle orientation		☆	☆	—	—	—	—
3rd/4th spindle output switching		☆	☆	—	—	—	—
Spindle synchronization		☆	☆	☆	☆	☆	☆
Simple spindle synchronization		☆	☆	☆	☆	☆	☆
Multiple spindle control		☆	☆	☆	☆	—	☆
Spindle positioning		—	☆	—	☆	—	☆
Rigid tapping		☆	☆	☆	☆	☆	☆
Three–dimensional rigid tapping		☆	☆	☆	☆	—	—
Rigid tapping by manual handle		☆	—	☆	—	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	TA	MA	TA	MA	TA

Tool functions, tool compensation functions

Tool function	T7+1/T6+2	—	○	—	○	—	○
	T with 8 digits	○	—	○	—	○	—
Tool compensation data	±with 6 digits, 32 items	○	—	○	—	○	—
	±with 6 digits, 64 items	☆	—	☆	—	☆	—
	±with 6 digits, 99 items	☆	—	☆	—	☆	—
	±with 6 digits, 200 items	☆	—	☆	—	☆	—
	±with 6 digits, 400 items	☆	—	☆	—	☆	—
	±with 6 digits, 499 items	☆	—	☆	—	—	—
	±with 6 digits, 999 items	☆	—	☆	—	—	—
	±with 6 digits, 9 or 16 sets	—	○	—	○	—	○
	±with 6 digits, 32 sets	—	☆	—	☆	—	☆
	±with 6 digits, 64 sets	—	☆	—	☆	—	☆
	±with 6 digits, 99 sets	—	☆	—	☆	—	—
Tool offset memory B	Separate memory for geometry and wear	☆	—	☆	—	☆	—
Tool offset memory C	Separate memory for geometry and wear Separate memory for length compensation and cutter compensation	☆	—	☆	—	☆	—
Tool length compensation		○	—	○	—	○	—
Tool offset		☆	○	☆	○	☆	○
Y–axis offset		—	☆	—	☆	—	☆
Cutter compensation B		☆	—	☆	—	—	—
Cutter compensation C		☆	—	☆	—	☆	—
Three–dimensional tool compensation		☆	—	☆	—	—	—
Tool–tip radius compensation		—	☆	—	☆	—	☆
Tool geometry/wear compensation		—	☆	—	☆	—	☆
Tool life management		☆	☆	☆	☆	☆	☆
Addition to tool life management sets	128 sets	—	☆	—	☆	—	—
	512 sets	☆	—	☆	—	☆	—
Extended tool life management		✻	—	✻	—	✻	—
Tool offset value counter input		—	○	—	○	—	○
7–digit tool offset value		—	☆	—	☆	—	—
Tool length measurement		☆	—	☆	—	☆	—
Automatic tool length measurement		☆	—	☆	—	☆	—
Tool length workpiece origin measurement B	*1	☆	—	☆	—	—	—
Automatic tool compensation		—	☆	—	☆	—	☆
Direct input of measured tool compensation value		—	○	—	○	—	○
Direct input of measured tool compensation value B	*1	—	☆	—	☆	—	☆
Grinding–wheel wear compensation		☆	—	☆	—	—	—
Automatic modification of tool offset		—	☆	—	☆	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	Specifications	Series 16*i* Series 160*i*		Series 18*i* Series 180*i*		Series 21*i* Series 210*i*	
		MA	TA	MA	TA	MA	TA

Editing

Part program storage length	10m	—	—	—	—	○	○
	20m	—	—	○	○	☆	☆
	40m	○	○	☆	☆	☆	☆
	80m	☆	☆	☆	☆	☆	☆
	160m	☆	☆	☆	☆	☆	☆
	320m	☆	☆	☆	☆	☆	☆
	640m	☆	☆	☆	☆	—	—
	1280m	☆	☆	☆	☆	—	—
	2560m	☆	☆	—	—	—	—
	5120m	☆	☆	—	—	—	—
Registered programs	63	○	○	○	○	○	○
	125	☆	☆	☆	☆	☆	☆
	200	☆	☆	☆	☆	☆	☆
	400	☆	☆	☆	☆	—	—
	1000	☆	☆	☆	☆	—	—
Part program editing		○	○	○	○	○	○
Program protection		○	○	○	○	○	○
Background editing		☆	☆	☆	☆	☆	☆
Expanded part program editing		☆	☆	☆	☆	☆	☆
Program copy between 2 paths	For 2–path system only	✻	✻	—	✻	—	—
Playback		☆	☆	☆	☆	☆	☆
Machining time stamp		☆	☆	☆	☆	—	—

Setting, display

Status display		○	○	○	○	○	○
Clock function		○	○	○	○	○	○
Current position display		○	○	○	○	○	○
Program display	31–character program name	○	○	○	○	○	○
Parameter setting display		○	○	○	○	○	○
Self–diagnosis function		○	○	○	○	○	○
Alarm display		○	○	○	○	○	○
Alarm history display		○	○	○	○	○	○
Operator message history display	*1	✻	✻	✻	✻	✻	✻
Operation history display	*1	○	○	○	○	○	○
Help function	*1	○	○	○	○	○	○
Online custom screen	*1	☆	☆	☆	☆	☆	☆
Remote diagnosis	Reading the information of mounted printed circuit board, parameter, CNC data, alarm status, etc.	✻	✻	✻	✻	✻	✻
Run time and parts number display		☆	☆	☆	☆	☆	☆
Actual speed display		○	○	○	○	○	○
Display of actual spindle speed and T code		✻	○	✻	○	✻	○
Floppy Cassette directory display		☆	☆	☆	☆	☆	☆
Directory display and punch for a specified group	*1	○	○	○	○	○	○
Graphic function	*1	☆	☆	☆	☆	☆	☆

Item	**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
Dynamic graphic display	For the Series 21*i*, tool path drawing only *1	☆	✻	☆	✻	☆	—
Background drawing (without CAP)	*1	☆	—	☆	—	—	—
Optional path name display	For 2–path system only	○	○	—	○	—	—
Servo adjustment screen		○	○	○	○	○	○
Spindle adjustment screen	Serial output only	✻	✻	✻	✻	✻	✻
Servo waveform display	Graphic display circuit required	✻	✻	✻	✻	✻	✻
Hardware/software system configuration display		○	○	○	○	○	○
Periodic maintenance screen	*1	○	○	○	○	○	○
Maintenance information display	*1	○	○	○	○	○	○
Software operator's panel		☆	☆	☆	☆	☆	☆
General–purpose switch on software operator's panel		☆	☆	☆	☆	☆	☆
Touch pad		☆	☆	☆	☆	—	—
Display language	English	○	○	○	○	○	○
	Japanese (kanji)	☆	☆	☆	☆	☆	☆
	German/French *1	☆	☆	☆	☆	☆	☆
	Italian *1	☆	☆	☆	☆	☆	☆
	Chinese *1	☆	☆	☆	☆	☆	☆
	Spanish *1	☆	☆	☆	☆	☆	☆
	Korean *1	☆	☆	☆	☆	☆	☆
Data protection key	4 types	○	○	○	○	○	○
Screen clear	*1	○	○	○	○	○	○

Data input/output

Reader/punch interface	Reader/punch interface (channel 1)	☆	☆	☆	☆	☆	☆
	Reader/punch interface (channel 2)	☆	☆	☆	☆	☆	☆
Simultaneous input/output operation	For single–path system only	☆	—	☆	—	—	—
Remote buffer	For single–path system only	☆	☆	☆	☆	☆	☆
High–speed remote buffer A	For single–path system only	☆	☆	☆	☆	☆	☆
High–speed remote buffer B	For single–path system only	☆	—	☆	—	☆	—
Data server	For single–path system only *1	☆	☆	☆	☆	—	—
External I/O device control		☆	☆	☆	☆	☆	☆
DNC1 control	Part program uploading/downloading, CNC data read/write, PMC data transfer, memory operation control, etc. *1	☆	☆	☆	☆	☆	☆
DNC2 control	For a single–path system only Part program uploading/downloading, CNC data read/write, PMC data transfer, memory operation control, etc. *1	☆	☆	☆	☆	☆	☆
Modem card control	*1	○	○	○	○	○	○
External tool compensation		☆	☆	☆	☆	☆	☆
External message		☆	☆	☆	☆	☆	☆

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item	**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
		MA	**TA**	**MA**	**TA**	**MA**	**TA**
External machine zero point shift		☆	☆	☆	☆	☆	☆
External data input	Including three items above	☆	☆	☆	☆	☆	☆
External key input		○	○	○	○	○	○
External programming		○	○	○	○	○	○
External workpiece number search	9999	○	○	○	○	○	○
External program number search	1 to 9999	✻	✻	✻	✻	✻	✻
Memory card input/output	*1	○	○	○	○	○	○
Screen hard copy	*1	☆	☆	☆	☆	—	—
Power motion manager	*1	☆	☆	☆	☆	☆	☆

Others

Status output signal	NC ready, servo ready, automatic operation, automatic operation start, automatic operation halt, reset, NC alarm, distribution completion, rewinding, inch input, cutting, in–position, threading, tapping, etc.	○	○	○	○	○	○
Control unit built–in display	7.2″ monochrome LCD *1	●	●	●	●	●	●
	9.5″ monochrome LCD *1	●	●	●	●	●	●
	8.4″ color LCD *1	●	●	●	●	●	●
	10.4″ color LCD	●	●	●	●	●	●
Control unit option slots (depth)	Without option slot (60 mm)	●	●	●	●	●	●
	2 optional slots (110 mm)	●	●	●	●	●	●
	4 optional slots (170 mm)	●	●	●	●	—	—
MDI unit	Separate–type MDI for 7.2″/8.4″ LCD (small) *1	●	●	●	●	●	●
	Separate–type MDI for 7.2″/8.4″ LCD (standard) *1	●	●	●	●	●	●
	Separate–type MDI for 9.5″/10.4″ LCD (standard, horizontal type) *1	●	●	●	●	●	●
	Separate–type MDI for 9.5″/10.4″ LCD (standard, vertical type) (MDI for 10.4″ LCD, for the Series 160*i*, 180*i*, and 210i)	●	●	●	●	●	●
	Separate–type MDI for 10.4″ LCD (PC key, vertical type) *2	●	●	●	●	●	●
	Touch pad (10.4″ LCD)	●	●	●	●	—	—

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

Item		**Specifications**	**Series 16*i* Series 160*i***		**Series 18*i* Series 180*i***		**Series 21*i* Series 210*i***	
			MA	**TA**	**MA**	**TA**	**MA**	**TA**
PMC system	PMC–RA1	Basic instruction: 5 μs/step Maximum ladder steps: 5,000 (The ladder editing function is limited.)	—	—	—	—	●	●
	PMC–RA5	Basic instruction: 0.085 μs/step Maximum ladder steps: 16,000	—	—	—	—	●	●
	PMC–RB5	Basic instruction: 0.085 μs/step Maximum ladder steps: 24,000	●	●	●	●	—	—
	PMC–RB6	Basic instruction: 0.085 μs/step Maximum ladder steps: 32,000 Step sequence function	●	●	●	●	—	—
	PMC–R*i*	Basic instruction: 0.085 μs/step Maximum ladder steps: 32,000 IEC1131–3	●	●	●	●	—	—
	C	Up to 2MB (PMC–RB5/RB6 required)	☆	☆	☆	☆	—	—
		Battery–powered memory expansion	☆	☆	☆	☆	—	—
Machine interface (I/O Link) Maximum DI/DO points: 1024/1024		I/O unit model A	●	●	●	●	●	●
		I/O unit model B	●	●	●	●	●	●
		Operator's panel I/O module	●	●	●	●	●	●
		Connector panel I/O module	●	●	●	●	●	●
		Power magnetics cabinet I/O module	●	●	●	●	●	●
Manual pulse generator			☆	☆	☆	☆	☆	☆
Pendant–type manual pulse generator		With axis selection switch and magnification selection switch	☆	☆	☆	☆	☆	☆
Cordless manual pulse generator		Used in Japan only	☆	☆	☆	☆	☆	☆
Applicable servo motor		FANUC AC servo motor α series (with serial interface pulse coder)	○	○	○	○	○	○
Applicable servo amplifier		FANUC servo amplifier α series	○	○	○	○	○	○
Separate position detector interface unit (for closed control)		2–phase pulse interface for separate pulse coder/linear optical scale	☆	☆	☆	☆	☆	☆
Linear scale interface with absolute addressing reference mark			☆	☆	☆	☆	—	—
Applicable spindle motor		FANUC AC spindle motor, etc.	○	○	○	○	○	○
Applicable spindle amplifier		FANUC servo amplifier α series	☆	☆	☆	☆	☆	☆
		Analog interface	☆	☆	☆	☆	☆	☆
Control unit supply voltage		24 VDC ±10%	●	●	●	●	●	●

Note) *1 For the Series 16*i*, 18*i*, and 21*i* only
*2 For the Series 160*i*, 180*i*, and 210*i* only

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

The number of all controlled axes is the sum of the number of machine controlled axes and the number of loader controlled axes. The machine controlled axes include Cs axis.

16*i*–MA/16*i*–TA/160*i*–MA/160*i*–TA (1–path)
: 12 axes (8 machine axes + 4 loader axes)
16*i*–MA/16*i*–TA/160*i*–MA/160*i*–TA (2–path)
: 18 axes (7 machine axes × 2 paths+4 loader axes)
18*i*–MA/18*i*–TA/180*i*–MA/180*i*–TA (1–path)
: 10 axes (6 machine axes + 4 loader axes)
18*i*–TA/180*i*–TA (2–path)
: 18 axes (5 machine axes × 2 paths+4 loader axes)
21*i*–MA/210*i*–MA
: 4 axes (4 machine axes)
21*i*–TA/210*i*–TA
: 8 axes (4 machine axes + 4 loader axes)

1.2
MACHINE CONTROLLED AXES

1.2.1
Number of Controlled Paths

Two–path control is available in 16*i*–MA, 16*i*–TA, and 18*i*–TA, 160*i*–MA, 160*i*–TA, 180*i*–TA
In 18*i*–MA, 180*i*–MA, 21*i*–MA, 21*i*–TA, 210*i*–MA, 210*i*–TA number of controlled paths is one.

1.2.2
Number of Basic Controlled Axes

16*i*–MA/18*i*–MA/160*i*–MA/180*i*–MA/21*i*–MA/210*i*–MA (each path) :
3 axes
16*i*–TA/18*i*–TA/160*i*–TA/180*i*–TA/210*i*–TA/210*i*–MA (each path) :
2 axes

1.2.3
Number of Basic Simultaneously Controlled Axes

16*i*–MA/18*i*–MA/160*i*–MA/180*i*–MA/21*i*–MA/210*i*–MA (each path) :
3 axes
16*i*–TA/18*i*–TA/160*i*–TA/180*i*–TA/21*i*–TA/210*i*–TA (each path) :
2 axes

1.2.4
Number of Controlled Axes Expanded (All)

16*i*–MA/16*i*–TA/160*i*–MA/160*i*–TA (1–path) :
Max. 8 axes (including Cs axis)
16*i*–MA/16*i*–TA/160*i*–MA/160*i*–TA (2–path) :
Max. 7 axes (6 feed axes + Cs axis)
18*i*–MA/18*i*–TA/180*i*–MA/180*i*–TA (1–path) :
Max. 6 axes (including Cs axis)
18*i*–TA/180*i*–TA (2–path) :
Max. 5 axes (4 feed axes + Cs axis)
21*i*–MA/21*i*–TA/210*i*–MA/210*i*–TA :
Max. 4 axes (including Cs axis)

1.2.5
Number of Simultaneously Controlled Axes Expanded (All)

16*i*–MA/16*i*–TA/160*i*–MA/160*i*–TA (each path) : Max. 6 axes
18*i*–MA/18*i*–TA/180*i*–MA/180*i*–TA (each path) : Max. 4 axes
21*i*–MA/21*i*–TA/210*i*–MA/210*i*–TA : Max. 4 axes

1.2.6
Axis Control by PMA

16*i*–MA/16*i*–TA/18*i*–MA/18*i*–TA/21*i*–MA/21*i*–TA/160*i*–MA/160*i*–TA/180*i*–MA/180*i*–TA/210*i*–MA/210*i*–TA (each path) :
Max. simultaneous 4 axes (Cs axis is disable.)

1.2.7
Cs Contour Control

16*i*–MA/16*i*–TA/18*i*–MA/18*i*–TA/21*i*–MA/21*i*–TA/160*i*–MA/160*i*–TA/180*i*–MA/180*i*–TA/210*i*–MA/210*i*–TA (each path) :
1 axis

1.3 LOADER CONTROLLED AXES

Number of controlled paths	: 1–path
Number of controlled axes	: Max. 4 axes
Number of simultaneously controlled axes	: Max. 4 axes
Number of controlled axes by PMA	: Max. 4 axes

1.4 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. For the 16-TB (2–path control), the two basic axes are always set to X and Z on each path, and additional axes can be selected freely from A, B, C, U, V, W, and Y.

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.

For two–path control, X, Y, and Z are always used as the names of the three basic axes. Additional axes can be assigned any of A, B, C, U, V, and W as their names.

1.5 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, option ”increment system 1/10” is required.

Table 1.5(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	
	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 1.5(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	
Millimeter 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.

1.5.1
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 0.001 inch
IS–C	0.001 mm, 0.001 deg, or 0.0001 deg

NOTE

The minimum input increment for inch input is not affected.

1.6
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 or Helical interpolation CW
G03	G03	G03		Circular interpolation CCW or Helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High speed cycle cutting, high–speed remote buffer A
G07	G07	G07		Hypothetical axis interpolation
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and return
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
G30.1	G30.1	G30.1		Floating reference point return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting
G35	G35	G35		Circular threading
G36	G36	G36		

G code list for T series (2/3)

G code			Group	Function
A	B	C		
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G39	G39	G39		Corner circular interpolation
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
G68	G68	G68	04	Mirror image for double turrets ON or balance cut mode
G69	G69	G69		Mirror image for double turrets OFF or balance cut mode 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

G code list for T series (3/3)

G code			Group	Function
A	B	C		
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)
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 revolution 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 (1/4)

G code	Group	Function	
G00	01	Positioning	
G01		Linear interpolation	
G02		Circular interpolation/Helical interpolation CW	
G03		Circular interpolation/Helical interpolation CCW	
G02.2, G03.2		Involute interpolation	
G02.3, G03.3		Exponential function interpolation	
G04	00	Dwell, Exact stop	
G05		High speed cycle machining, high–speed remote buffer A/B, high–precision contour control, simple high–precision contour control	
G07		Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		Look–ahead control	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G11		Programmable data input mode cancel	
G12.1	25	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation cancel mode	
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	
G25	24	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	

G code list for M series (2/4)

G code	Group	Function
G27	00	Reference position return check
G28		Return to reference position
G29		Return from reference position
G30		2nd, 3rd and 4th reference position return
G30.1		Floating reference point return
G31		Skip function
G33	01	Thread cutting
G37	00	Automatic tool length measurment
G39		Corner offset circular interpolation
G40	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
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

G code list for M series (3/4)

G code	Group	Function
G60	00	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
G72.1	00	Rotational copy
G72.2		Linear copy
G73	09	Peck drilling cycle
G74		Counter tapping cycle
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

G code list for M series (4/4)

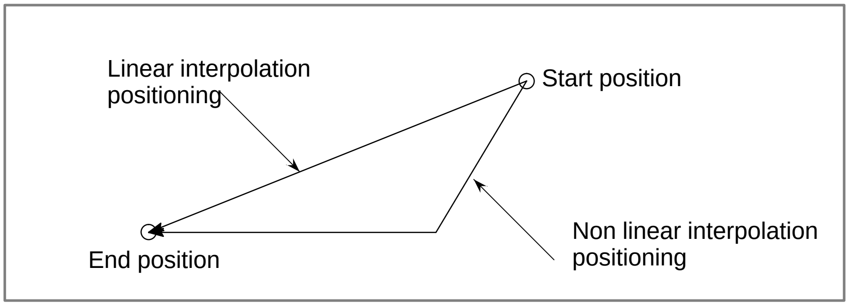
G code	Group	Function
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle
G160	20	In–feed control function cancel(for grinding machine)
G161		In–feed control function(for grinding machine)

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.



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.

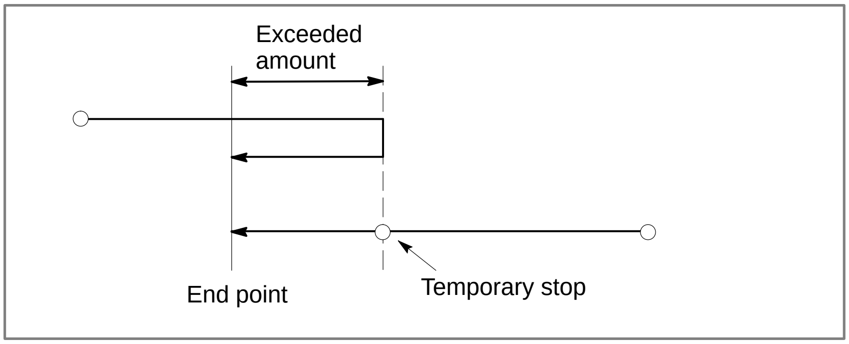
Format

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.

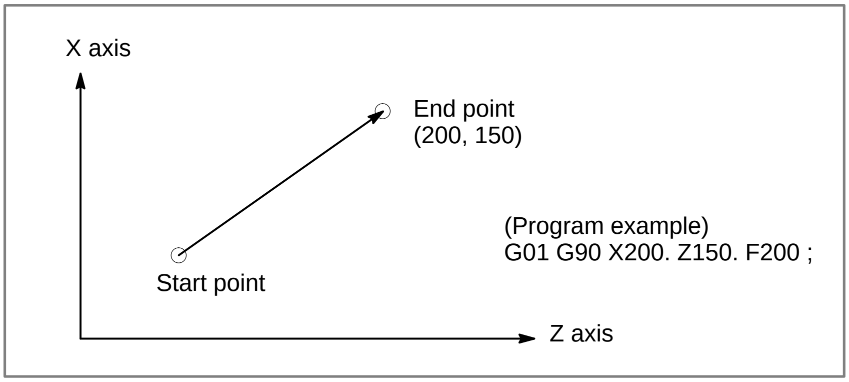


Format

G60 IP_ ;

3.3 LINEAR INTERPOLATION (G01)

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



(Program example)
G01 G90 X200. Z150. F200 ;

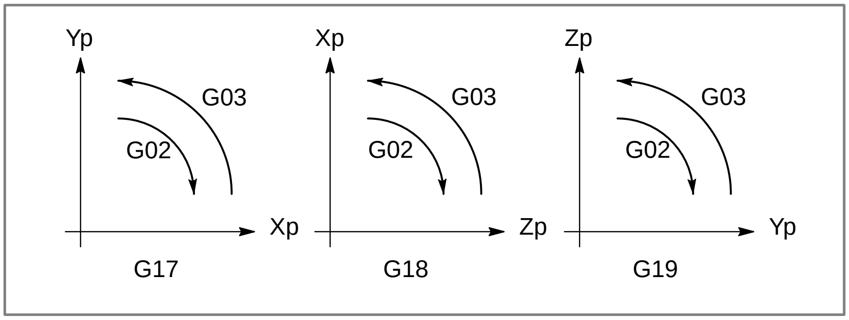
Format

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

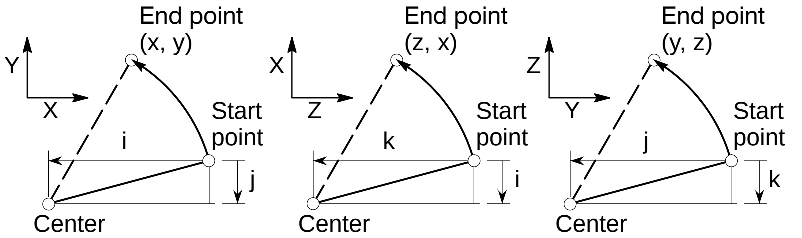
$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\\mathbf{G03}\end{matrix}\right\}\mathbf{Zp\_\ Xp\_}\left\{\begin{matrix}\mathbf{R\_}\\\mathbf{K\_\ I\_}\end{matrix}\right\}\mathbf{F\_\ ;}$$

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.)



When the option for specifying arc radius R with nine digits is selected for the T series, the valid radius range for circular interpolation is expanded as follows:

Without the option for specifying arc radius R with nine digits

		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

With the option for specifying arc radius R with nine digits

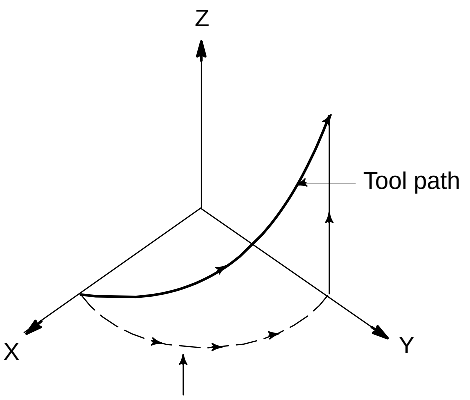
		Input increments	
		Metric input	**Inch input**
Increment system	IS–B	0.001 to 999999.999mm	0.0001 to 99999.9999inch
	IS–C	0.0001 to 99999.9999mm	0.00001 to 9999.99999inch

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:

$$F \times \frac{Lengthoflinearaxis}{Arclength}$$



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

Format

Arc on the Xp-Yp plane

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

Arc on the Zp-Xp plane

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

Arc on the Yp-Zp plane

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

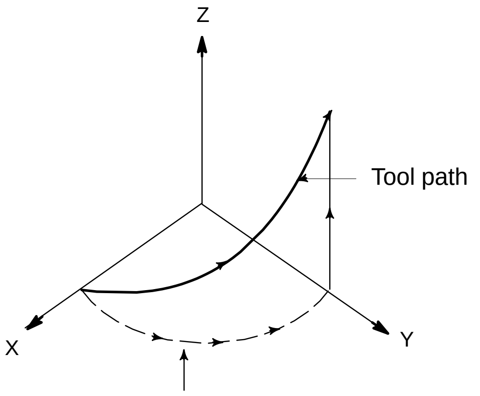
α, β: Optional axis other than the circular interpolation axes

3.6 HELICAL INTERPOLATION B (G02, G03)

M series

Helical interpolation B moves the tool in a helical manner. This interpolation can be executed by specifying the circular interpolation command together with up to four additional axes in simple high–precision contour control mode.

Basically, the command can be specified by adding two movement axes to a standard helical interpolation command (see Section 3.5). Address F should be followed by a tangential velocity, which has been determined by also taking movement along the linear axes into consideration.



The feedrate equals the tangential velocity determined by also taking movement along the linear axes into consideration.

Format

With an arc in the Xp–Yp plane

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

With an arc in the Zp–Xp plane

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

With an arc in the Yp–Zp plane

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

$\alpha\_\ \beta\_\ \gamma\_\ \delta$: Any axis to which circular interpolation is not applied. Up to four axes can be specified.

3.7

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.

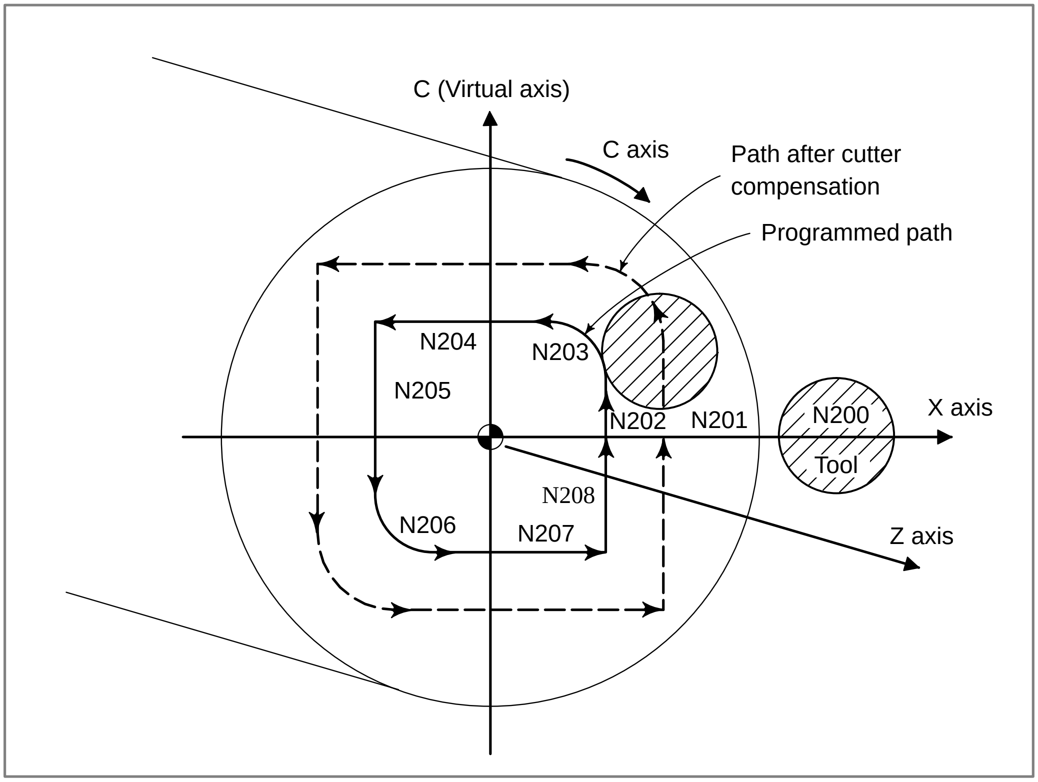
As for feedrate, specify the tangential speed (relative speed between the workpiece and the tool) on the polar coordinate interpolation plane (cartesian coordinate system) with F.

- **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;
N202 C10.0;
N203 G03 X20.0 C20.0 R10.0 ;             Contour program
N204 G01 X-40.0 ;                        (Program in cartesian coordinate
N205 G-10.0 ;                            system of X-C plane)
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
N300 Z_ ;                                interpolation
N400 X_ C_ ;
    :
M30 ;
```

3.8 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.

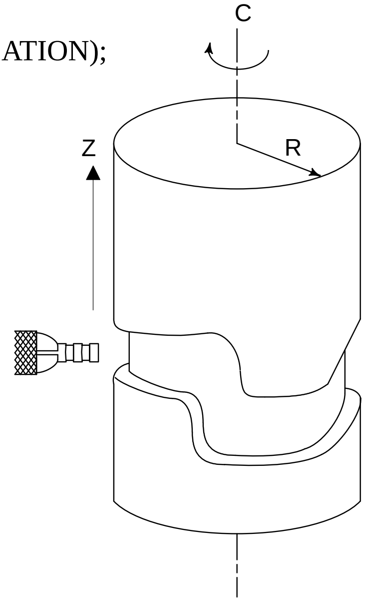
- **Cancellation mode of cylindrical interpolation**

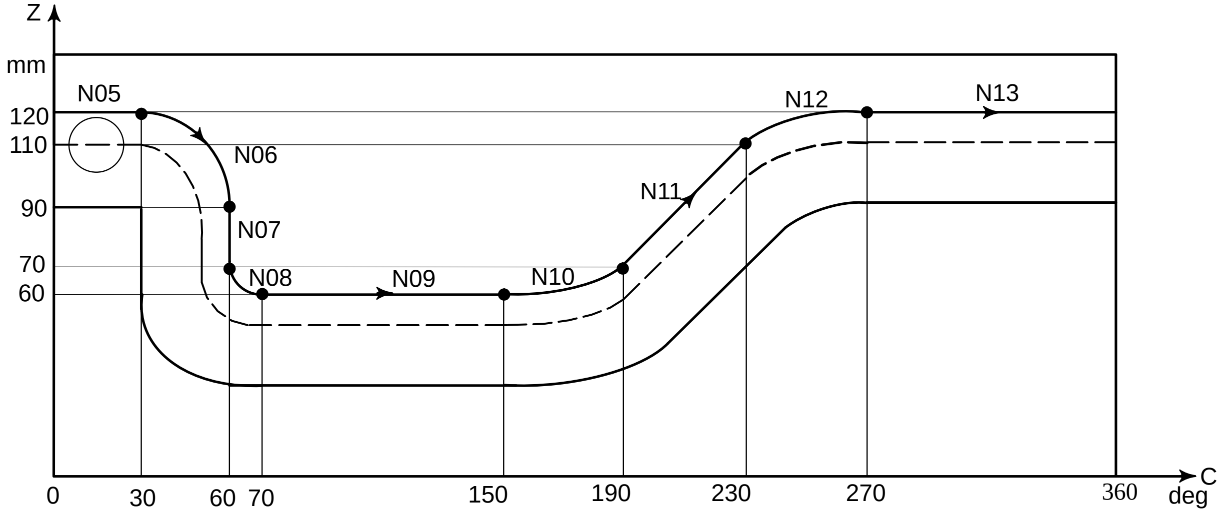
G07.1 (Name of rotary axis) 0; Cancellation mode of cylindrical interpolation is made when commanded as above.

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 G02 Z90.0 C60.0 R30.0 ;
   N7 G01 Z70.0;
   N8 G03 Z60.0 C70.0 R10.0;
   N9 G01 C150.0;
   N10 G03 Z70.0 C190.0 R75.0;
   N11 G01 Z110.0 C230.0;
   N12 G02 Z120.0 C270.0 R75.0;
   N13 G01 G360.0;
   N14 G40 Z100.0;
   N15 G07.1 C0;
   N16
   M30;
```

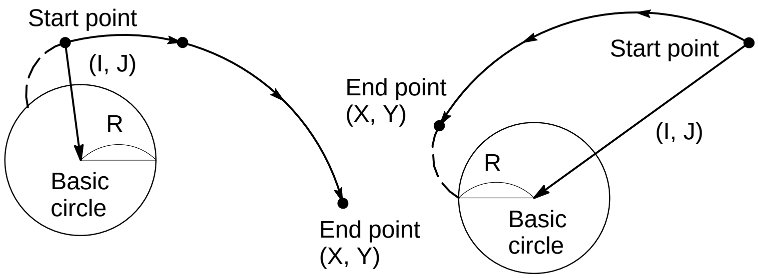




3.9 INVOLUTE INTERPOLATION (G02.2, G03.2)

M series

With the following command, the involute curve machining can be performed. Approximate involute curve with a minute straight line or arc is not needed. Therefore, the programming becomes simple and reduces the tape length. The distribution of the pulse will not be interrupted during the continuous minute block high speed operation, so fast, smooth involute curve machining is possible.



Clockwise involute interpolation **Counterclockwise involute interpolation**

Format

Xp–Yp plane

G17 $\left\{ \begin{array}{l} \textbf{G02.2} \\ \textbf{G03.2} \end{array} \right\}$ **Xp_ Yp_ I_ J_ R_ F_ ;**

Zp–Xp plane

G18 $\left\{ \begin{array}{l} \textbf{G02.2} \\ \textbf{G03.2} \end{array} \right\}$ **Zp_ Xp_ K_ I_ R_ F_ ;**

Yp–Zp plane

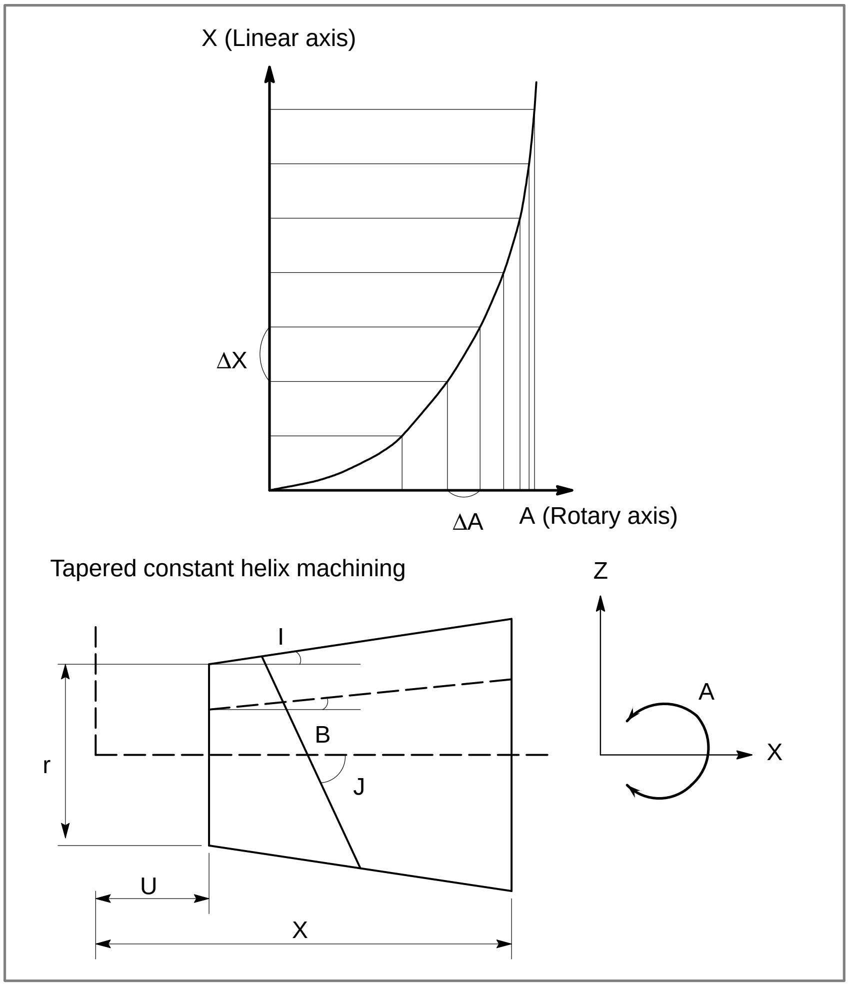
G19 $\left\{ \begin{array}{l} \textbf{G02.2} \\ \textbf{G03.2} \end{array} \right\}$ **Yp_ Zp_ J_ K_ R_ F_ ;**

G02.2 : Clockwise involute interpolation
G03.2 : Counterclockwise involute interpolation
Xp, Yp, Zp : End point coordinate value
I, J, K : Distance to the center of the basic circle of the involute curve from start point
R : Radius of basic circle
F : Cutting feedrate

3.10 EXPONENTIAL FUNCTION INTERPOLATION (G02.3, G03.3)

M series

In synchronization with the travel of the rotary axis, the linear axis (X axis) performes the exponential function interpolation. With the other axes, the linear interpolation the X axis is performed.
This function is effective for the tapered constant helix machining in the tool grinding machine.
This function is the best for the fluting with the end mill etc. and grinding.



Format

Positive rotation (ω=0)

G02.3 X_Y_ Z_ I_ J_ K_ R_ F_ Q_ ;

Negative rotation (ω=1)

G03.3 X_Y_ Z_ I_ J_ K_ R_ F_ Q_ ;

X_ : Command terminal point by Absolute or incremental
Y_ : Command terminal point by Absolute or incremental
Z_ : Command terminal point by Absolute or incremental
I_ : Command of angle I (The command unit is 0.001 deg. The range of command is 1 to ±89deg)
J_ : Command of angle J (The command unit is 0.001 deg. The range of command is 1 to ±89deg)
K_ : Amount of division of the linear axis in the exponential function interpolation (amount of span). The command range is a positive value.)
R_ : Command of constant value R in the exponential function interpolation.
F_ : Command of initial feed rate. The command is the same as the normal F code. The feed rate shall be given by the synthesized speed including the rotary axis.
Q_ : Command of feed rate at terminal point. The command unit is based on the reference axis. Within the CNC, the tool is interpolated between the initial feed rate (F_) and final feed rate (Q_) depending on the amount of linear axis travel.

Explanations

The exponential function relation expression between the linear axis and the rotary axis is defined as in the following :

$$X(\theta) = R \times (e^{\frac{\theta}{K}}-1) \times \frac{1}{tan(I)} \qquad \text{..... Travel of linear axis (1)}$$

$$A(\theta) = (-1)^{\omega} \times 360 \times \frac{\theta}{2\pi} \qquad \text{..... Travel of rotation axis (2)}$$

$$K= \frac{\tan (J)}{\tan (I)}$$

ω=0 or 1

R, I, J are constant and θ is the angle (radian) of rotation.
Also from the equation (1),

$$\theta(X) = K \times \ell n\,\{\frac{X \times \tan(I)}{R} + 1\}$$

Thus, when the tool moves from X1 to X2 along the linear axis, the angle moved about the rotation axis is calculated as follows:

$$\varDelta\theta = K \times \{\ell n\,(\frac{X_2 \times \tan(I)}{R} + 1) - \ell n\,(\frac{X_1 \times \tan(I)}{R} + 1)\ \}$$

Specify formulas (1) and (2) in commands using the format described above.

3.11 SMOOTH INTERPOLATION (G05.1)

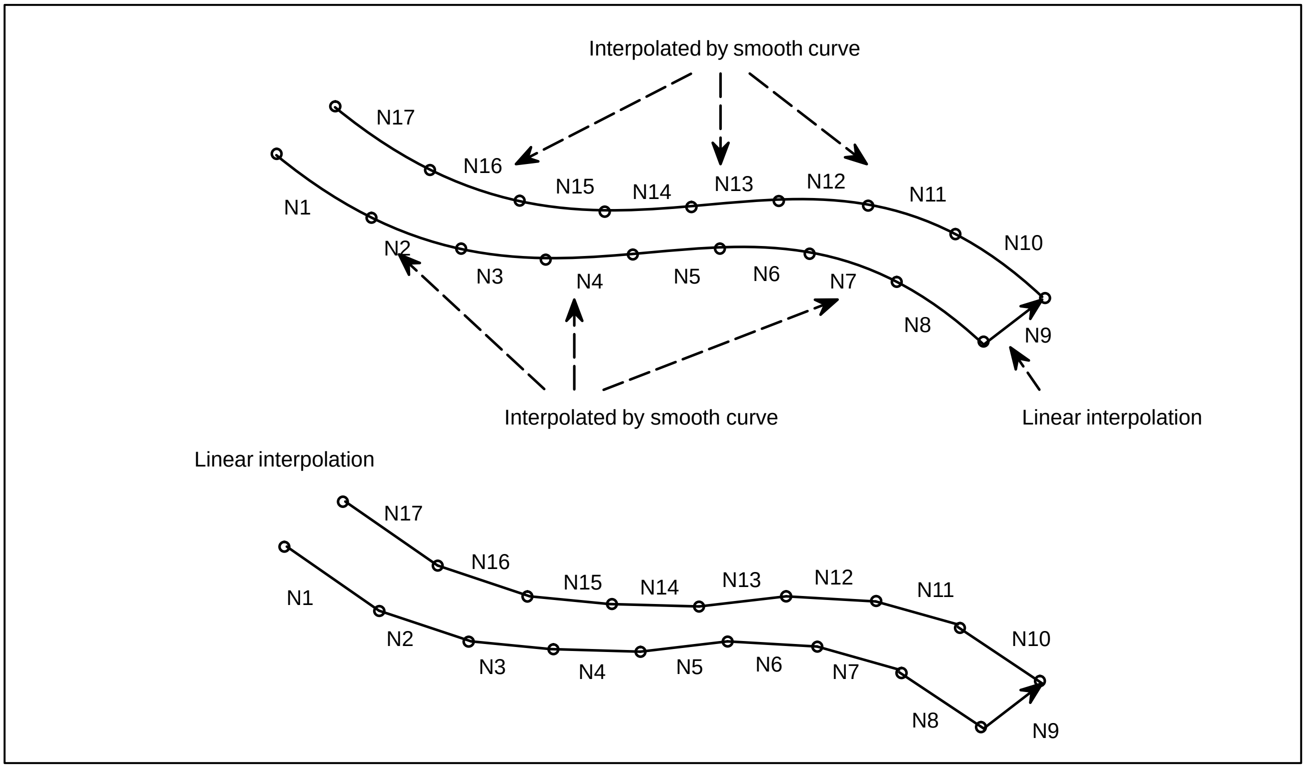
M series

Either of two types of machining can be selected, depending on the program command.

- For those portions where the accuracy of the figure is critical, such as at corners, machining is performed exactly as specified by the program command.
- For those portions having a large radius of curvature where a smooth figure must becreated, points along the machining path are interpolated with a smooth curve, calculated from the polygonal lines specified with the program command (smooth interpolation).

In smooth interpolation mode, the CNC automatically determines, according to the program command, whether an accurate figure is required, such as at corners, or a smooth figure is required where the radius of curvature is large. If a block specifies a travel distance or direction which differs greatly from that in the preceding block, smooth interpolation is not performed for that block. Linear interpolation is performed exactly as specified by the program command. Programming is thus very simple.

Examples



Smooth interpolation can be specified in high–speed contour control mode (between G05 P10000 and G05 P0). For details of high–speed contour control, see Section 20.6.

Format

Starting of smooth interpolation mode

G05.1 Q2X0Y0Z0;

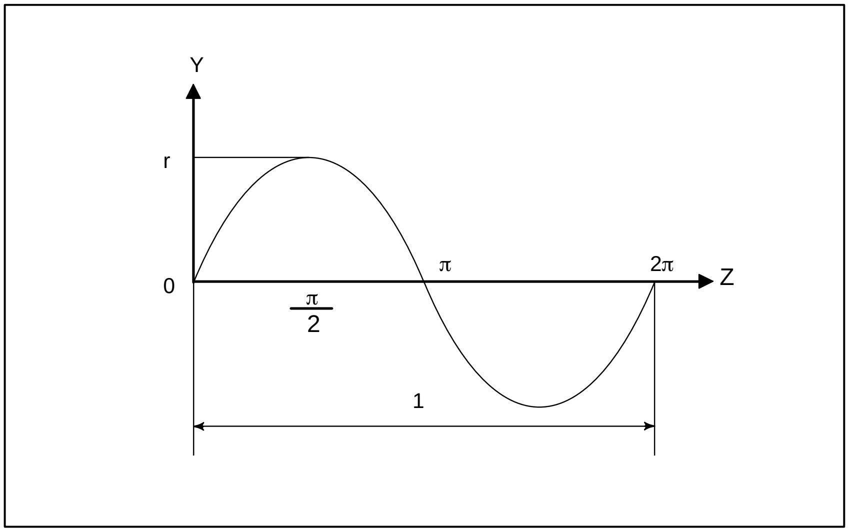
Cancelation of smooth interpolation mode

G05.1 Q 0ϕ;

3.12 HYPOTHETICAL AXIS INTERPOLATION (G07)

In helical interpolation, when pulses are distributed with one of the circular interpolation axes set to a hypothetical axis, sine interpolation is enable.

When one of the circular interpolation axes is set to a hypothetical axis, pulse distribution causes the speed of movement along the remaining axis to change sinusoidally. If the major axis for threading (the axis along which the machine travels the longest distance) is set to a hypothetical axis, threading with a fractional lead is enabled. The axis to be set as the hypothetical axis is specified with G07.



Format

G07 α 0; Hypothetical axis setting

G07 α 1; Hypothetical axis cancel

Where, α is any one of the addresses of the controlled axes.

3.13 SPIRAL INTERPOLATION, CONICAL INTERPOLATION

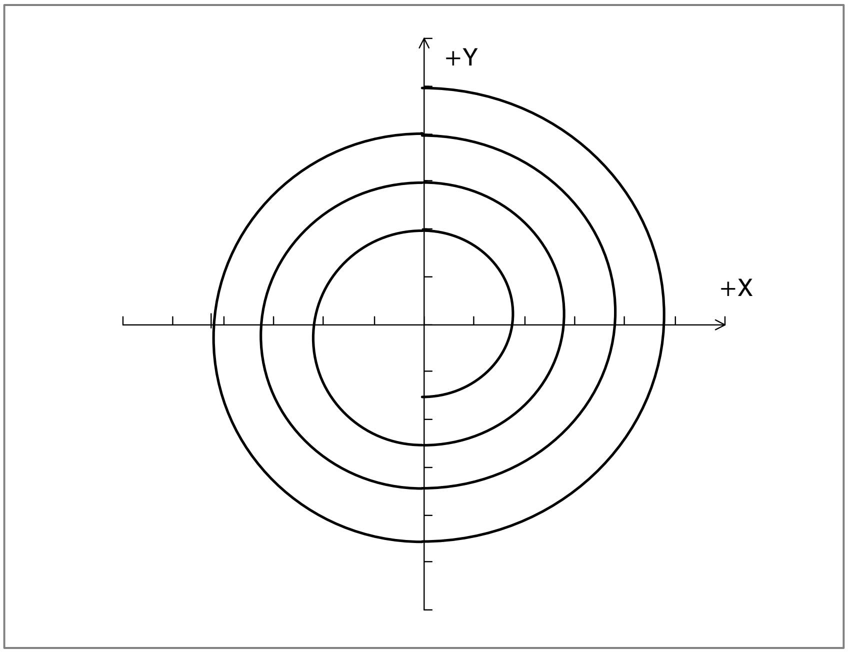
M series

Spiral interpolation is enabled by specifying the circular interpolation command together with a desired number of revolutions or a desired increment (decrement) for the radius per revolution.

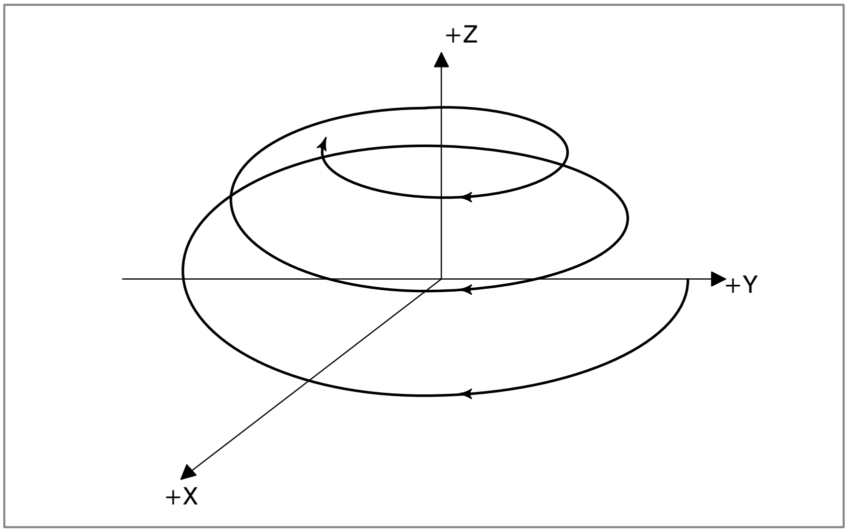
Conical interpolation is enabled by specifying the spiral interpolation command together with one or two additional axes of movement, as well as a desired increment (decrement) for the position along the additional axes per spiral revolution.

Spiral interpolation and conical interpolation do not support bell–shaped acceleration/deceleration after interpolation for cutting feed.

- **Spiral interpolation**



- **Conical interpolation**



Format

• Spiral interpolation

Xp–Yp plane

$$\mathbf{G17}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{X\_\ Y\_\ I\_\ J\_\ Q\_\ L\_\ F\_\ ;}$$

Zp–Xp plane

$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{Z\_\ X\_\ K\_\ I\_\ Q\_\ L\_\ F\_\ ;}$$

Yp–Zp plane

$$\mathbf{G19}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{Y\_\ Z\_\ J\_\ K\_\ Q\_\ L\_\ F\_\ ;}$$

X,Y,Z : Coordinates of the end point
L : Number of revolutions (positive value without a decimal point)
Q : Radius increment or decrement per spiral revolution
I, J, K : Signed distance from the start point to the center (same as the distance specified for circular interpolation)
F : Feedrate

• Conical interpolation

Xp–Yp plane

$$\mathbf{G17}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{X\_\ Y\_\ I\_\ J\_\ Q\_\ L\_\ F\_\ ;}$$

Zp–Xp plane

$$\mathbf{G18}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{Z\_\ X\_\ K\_\ I\_\ Q\_\ L\_\ F\_\ ;}$$

Yp–Zp plane

$$\mathbf{G19}\left\{\begin{matrix}\mathbf{G02}\\ \mathbf{G03}\end{matrix}\right\}\mathbf{Y\_\ Z\_\ J\_\ K\_\ Q\_\ L\_\ F\_\ ;}$$

X,Y,Z : Coordinates of the end point
L : Number of revolutions (positive value without a decimal point)
Q : Radius increment or decrement per spiral revolution
I, J, K : Two of the three values represent a signed vector from the start point to the center. The remaining value is a height increment or decrement per spiral revolution in conical interpolation
When the Xp–Yp plane is selected:
The I and J values represent a signed vector from the start point to the center.
The K value represents a height increment or decrement per spiral revolution.
F : Feedrate (determined by taking movement along the linear axes into consideration)

3.14 NURBS INTERPOLATION (G06.2)

Many computer–aided design (CAD) systems used to design metal dies for automobiles and airplanes utilize non–uniform rational B–spline (NURBS) to express a sculptured surface or curve for the metal dies.

This function enables NURBS curve expression to be directly specified to the CNC. This eliminates the need for approximating the NURBS curve with minute line segments. This offers the following advantages:

1. No error due to approximation of a NURBS curve by small line segments
2. Short part program
3. No break between blocks when small blocks are executed at high speed
4. No need for high–speed transfer from the host computer to the CNC

When this function is used, a computer–aided machining (CAM) system creates a NURBS curve according to the NURBS expression output from the CAD system, after compensating for the length of the tool holder, tool diameter, and other tool elements. The NURBS curve is programmed in the NC format by using these three defining parameters: control point, weight, and knot.

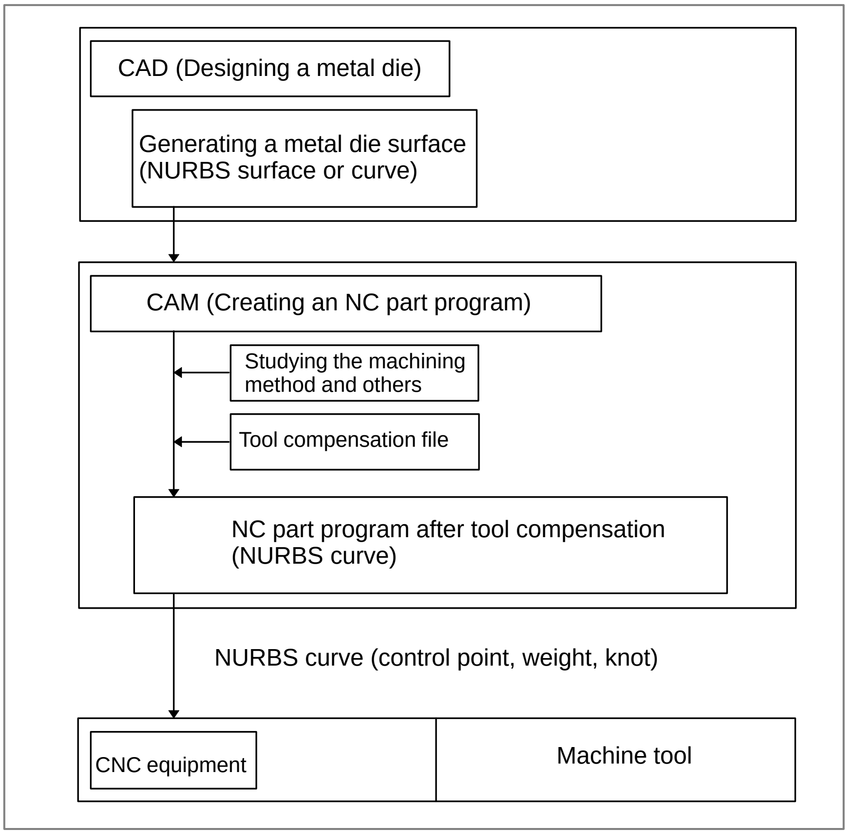


Fig. 3.14 NC part program for machining a metal die according to a NURBS curve

NURBS interpolation must be specified in high–precision contour control mode (between G05 P10000 and G05 P0). The CNC executes NURBS interpolation while smoothly accelerating or decelerating the movement so that the acceleration on each axis will not exceed the allowable maximum acceleration of the machine. In this way, the CNC automatically controls the speed in order to prevent excessive strain being imposed on the machine.

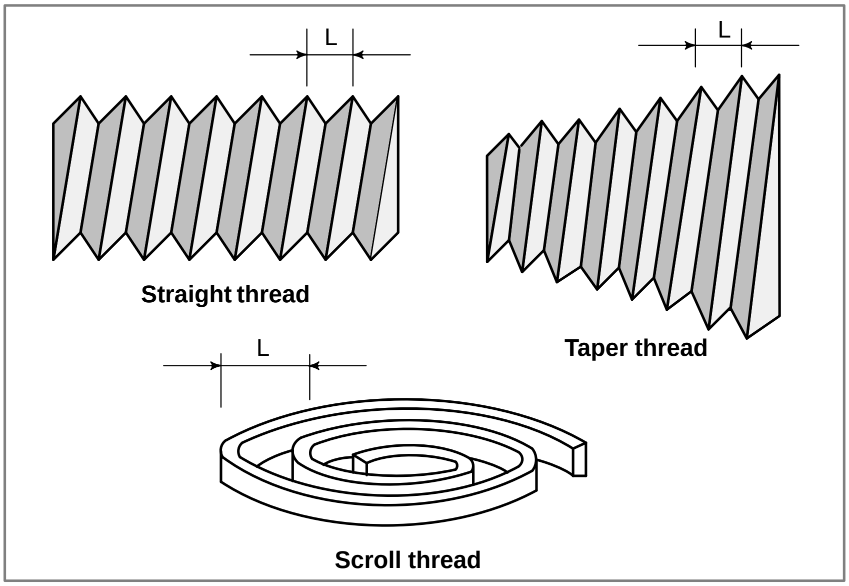
Format

```
G05 P10000 ; (Start high–precision contour control mode)
...
G06.2 [P_]   K_ X_ Y_ Z_ [R_ ] [F_ ] ;
             K_ X_ Y_ Z_ [R_ ] ;
             K_ X_ Y_ Z_ [R_ ] ;
             K_ X_ Y_ Z_ [R_ ] ;
             ...
             K_ X_ Y_ Z_ [R_ ] ;
             K_ ;
             ...
             K_ ;

G01 ...
...G05 P0   ;   (End high–precision contour control mode)

G06.2       :   Start NURBS interpolation mode
P_          :   Rank of NURBS curve
X_ Y_ Z_    :   Control point
R_          :   Weight
K_          :   Knot
F_          :   Feedrate
```

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

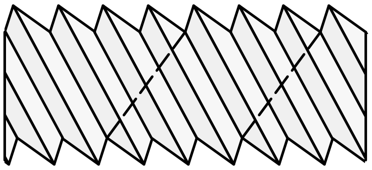
NOTE

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

4.2

T series

MULTIPLE–THREAD CUTTING (G33)



Multiple–thread screws

Format

Constant–lead threading

G33 IP_ **F_ Q_ ;**

G33 IP_ **Q_ ;**

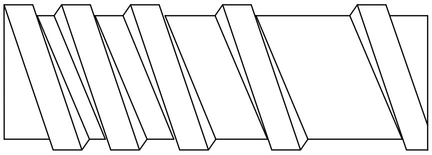
IP_ : End point
F_ : Lead in longitudinal direction
Q_ : Threading start angle

4.3

T series

VARIABLE LEAD THREAD CUTTING (G34)

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_ :**

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

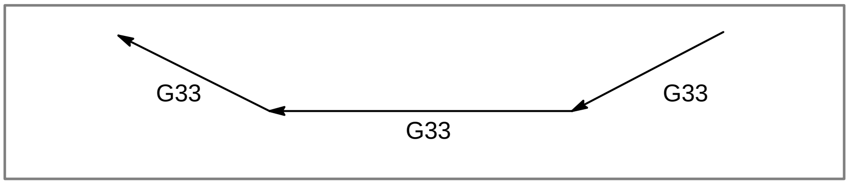
Command value range of
lead increase/decrease (K) per spindle rotation:

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

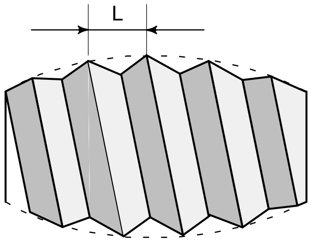
Continuous thread cutting in which thread cutting command block is continuously commanded is available. As it is controlled so that the spindle synchronism shift (occurred when shifting from one block to another) is kept to a minimum, special threads like threads which leads or shape change during the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



4.5 CIRCULAR THREADING (G35, G36)

T series

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.



Circular thread

Format

$$\left\{\begin{array}{l}\textbf{G35}\\ \textbf{G36}\end{array}\right\} \quad \textbf{X (U) \_ Z (W) \_} \quad \left\{\begin{array}{l}\textbf{I \_ K \_}\\ \textbf{R \_ \_ \_}\end{array}\right\} \quad \textbf{F \_ Q \_}$$

G35 : Clockwise circular threading command
G36 : Counterclockwise circular threading command

X (U) : Specify the arc end point (in the same way as for G02, G03).
Z (W)

I, K : Specify the arc center relative to the start point, using relative coordinates (in the same way as for G02, G03).
R : Specify the arc radius.
F : Specify the lead in the direction of the major axis.
Q : Specify the shift of the threading start angle (0 to 360° in units of 0.001°)

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

5.2.5 Inverse Time Feed (G93)

M series

Inverse time feed mode is commanded by G93, and inverse time by F code. Inverse time is commanded with the following value in a 1/min unit.

In linear interpolation	F= Speed/distance
In circular interpolation	F= Speed/radius

When F0 is commanded, alarm occurs.

5.2.6 F1–digit 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 F1-digit 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
Second Feed Rate Override

Cutting feed rate can be overrided by:

0 to 254% (per every 1%)

A second override can be performed on feed rats once overrided.

No override can be performed on functions as thread cutting and tapping in which override is inhibited.

This function is used for controlling feed rate in adaptive control, etc.

5.3.3
Rapid Traverse Override

Rapid traverse rate can be overridden by:

F0, 25%, 50%, 100%.

F0: A constant speed per axis can be set by parameter

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

5.3.4
Override Cancel

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

5.3.5
Jog Override

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

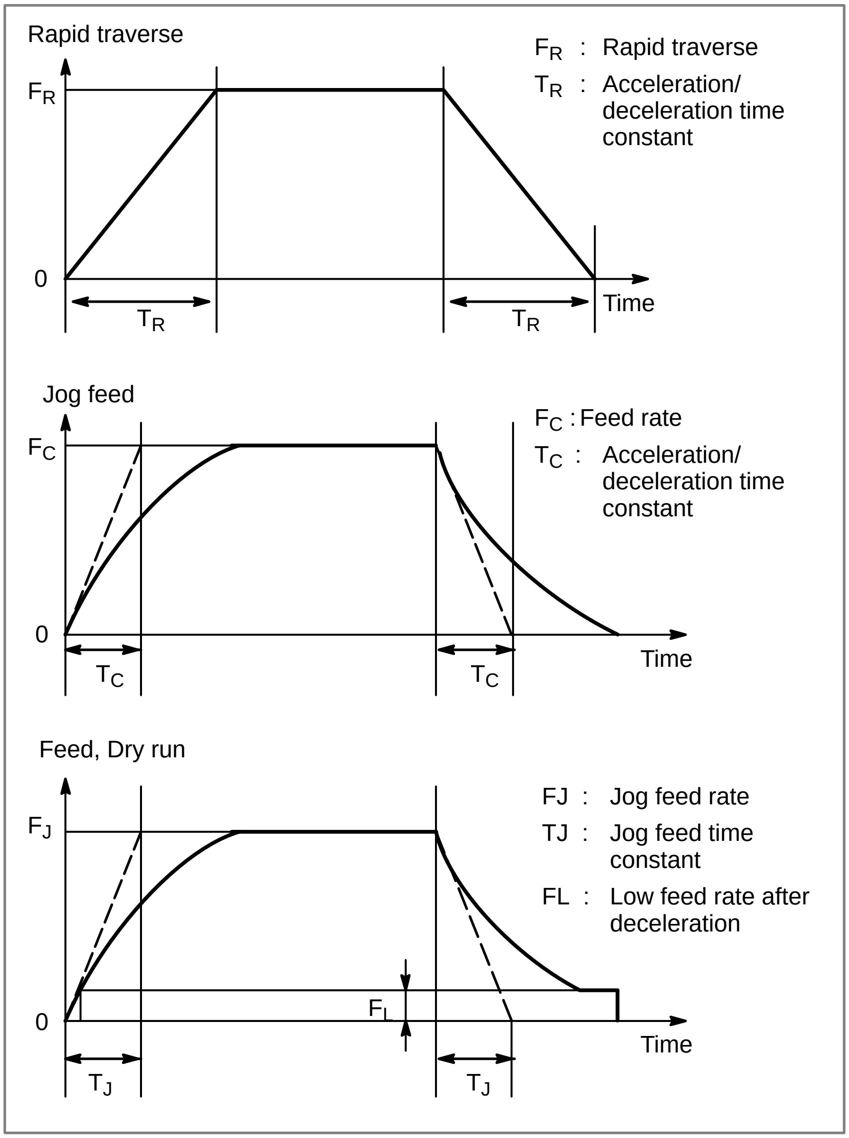
0% to 655.34% (in steps of 0.01%)

5.4
AUTOMATIC ACCELERATION/ DECELERATION

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.

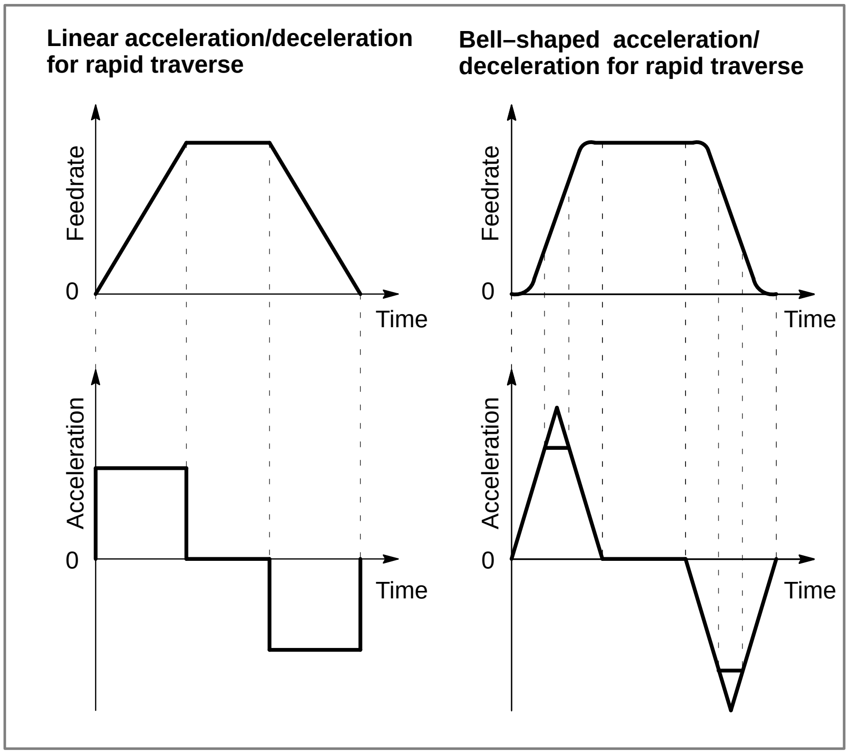
Automatic acceleration/deceleration is also performed when feed rate changes, so change in speed is also smoothly done.

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)



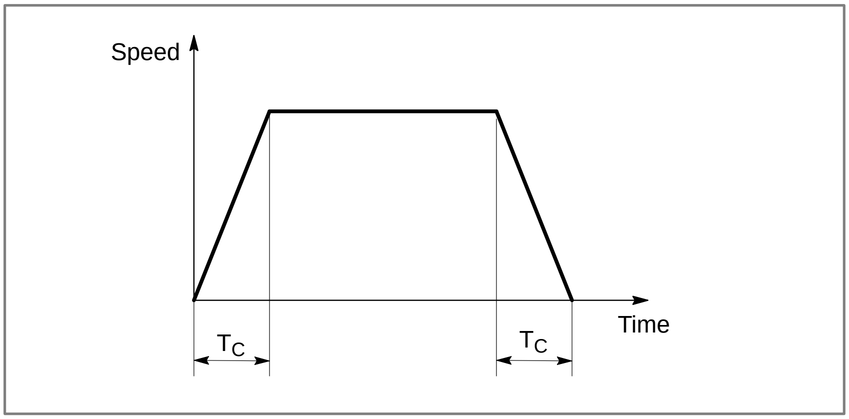
5.5
RAPID TRAVERSE BELL–SHAPED ACCELERATION/ DECELERATION

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.

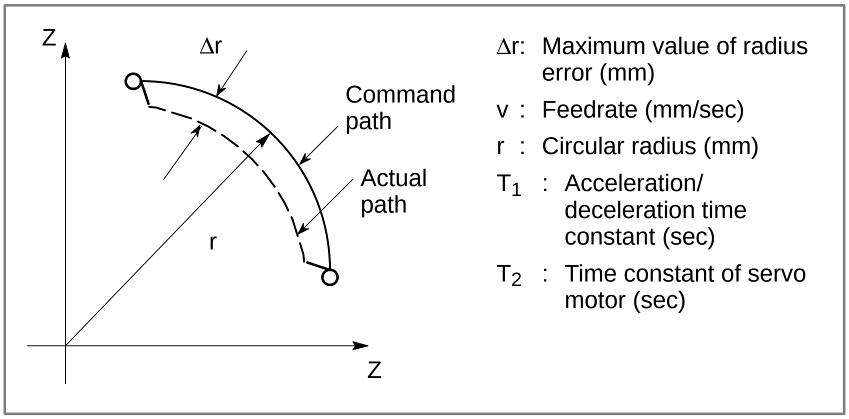


5.6

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.



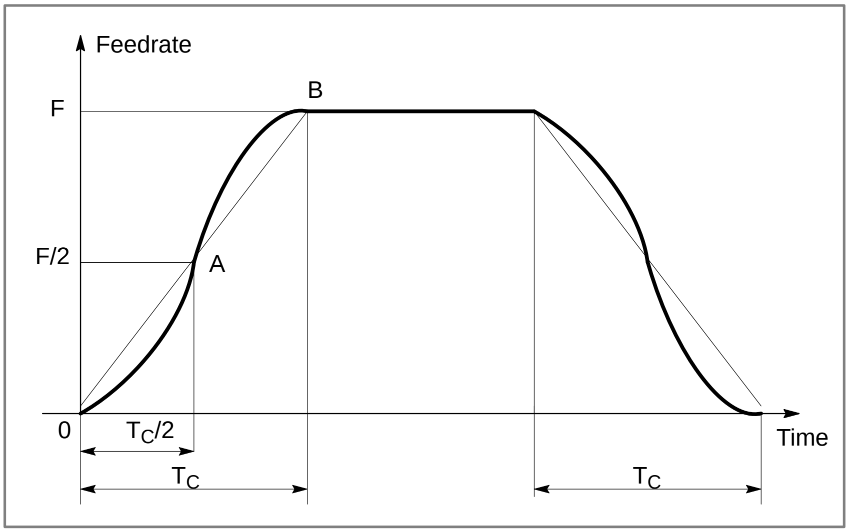
The maximum value of error in this radius direction is obtained approximately by the following equation.

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

$$\mathit{\Delta} r = (\frac{1}{24}T_1{}^2 + \frac{1}{2}T_2{}^2)\frac{V^2}{r}$$ ..... 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 :

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

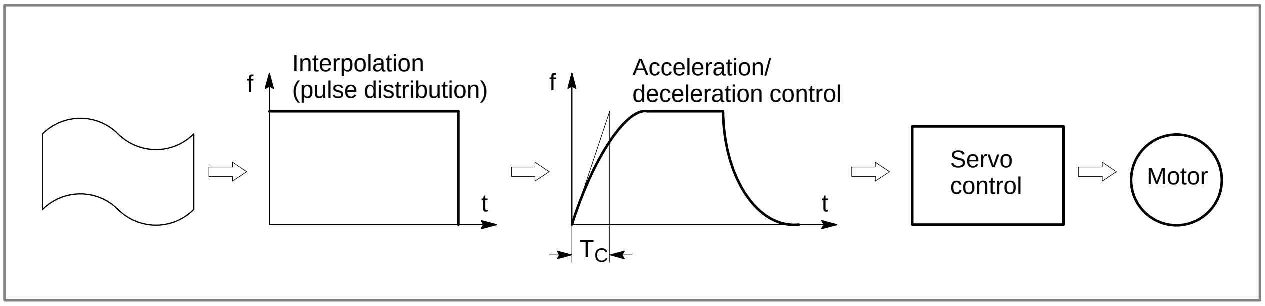
The curve AB and 0A are symmetric with respect to point A.

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

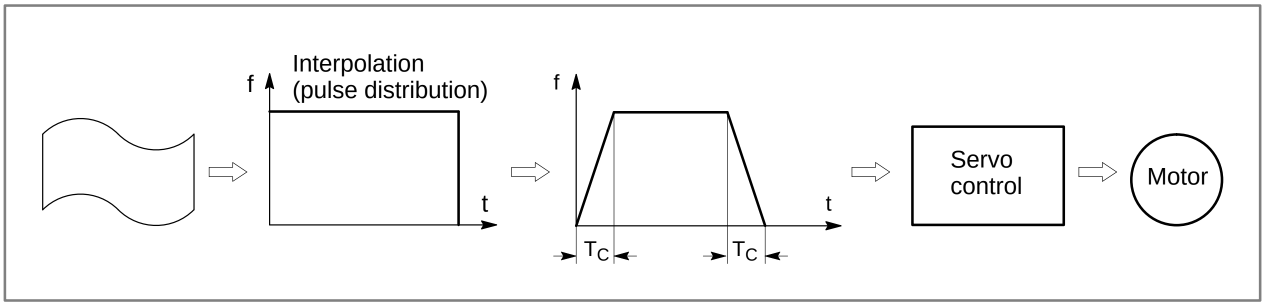
5.8 LINEAR ACCELERATION/ DECELERATION BEFORE CUTTING FEED INTERPOLATION

In response to the cutting feed command , the feedrate before interpolation, the command feedrate can be directly accelerated/ decelerated. This enables a machined shape error caused by the delay of acceleration/deceleration to be eliminated.

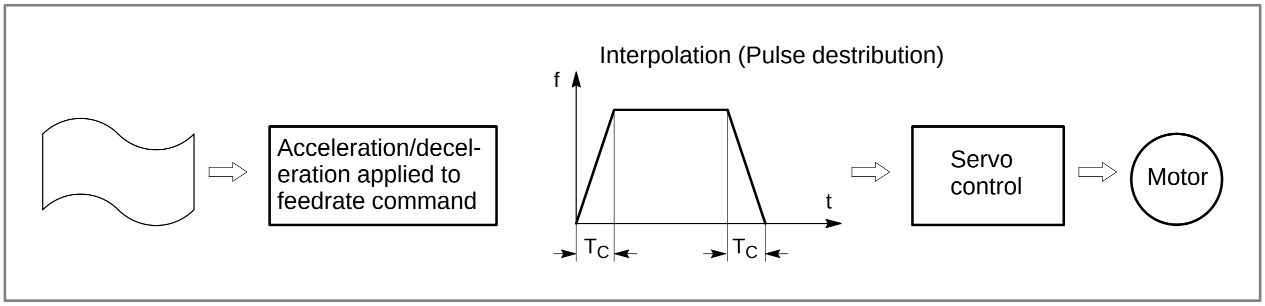
- **Exponential acceleration/deceleration after cutting feed interpolation**



- **Linear acceleration/ deceleration after cutting feed interpolation**



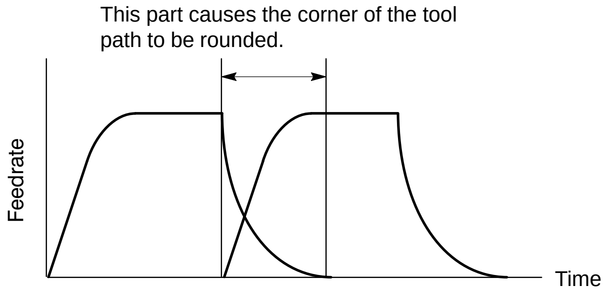
- **Linear acceleration/ deceleration before cutting feed interpolation**

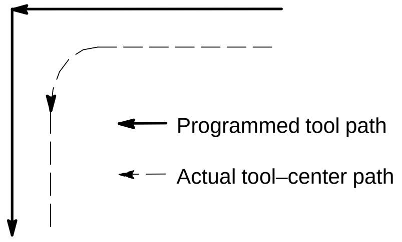


5.9 ERROR DETECTION

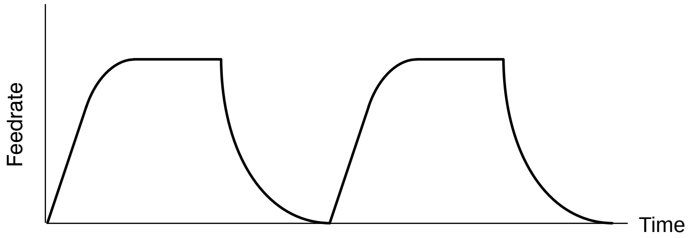
T series

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.



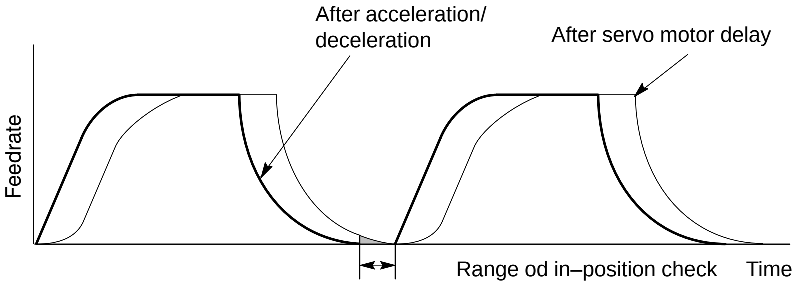


If the error detect signal is used, it is possible to specify that a block not be started until the acceleration/deceleration of the previous block has been completed.



NOTE

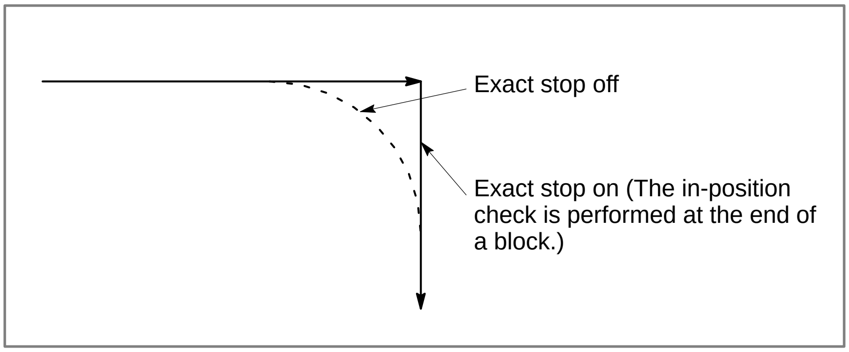
If the error detect signal is on, a cutting block is not executed until the acceleration/deceleration of the previous cutting block has been completed.
This function alone cannot prevent corner rounding due to delay caused by the servo motor, however.
To prevent corner rounding due to delay caused by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



5.10 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.



5.11 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.

5.12 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.

5.13 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.

5.14 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.

5.15
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 $\left\{\begin{matrix}\mathbf{P\_}\\ \mathbf{X\_}\end{matrix}\right\}$ **;**

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

Per revolution dwell

G95 G04 $\left\{\begin{matrix}\mathbf{P\_}\\ \mathbf{X\_}\end{matrix}\right\}$ **;**

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

5.16
POSITIONING BY OPTIMUM ACCELERATION

When a rapid traverse command is specified during automatic operation, the function for positioning by optimum acceleration can be used to adjust the rapid traverse rate, time constant, and loop gain, according to the amount of travel for the block. This reduces the time required for positioning and position check, therefore reducing the cycle time.
When rapid traverse is specified in automatic operation, the function adjusts the rapid traverse rate, time constant, and loop gain to one of seven levels, according to the amount of travel for the block. The relationship between the amount of travel and the corresponding rapid traverse rate, time constant, and loop gain is specified in parameters. This function is not effective for cutting feed.

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(ONLY FOR 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.

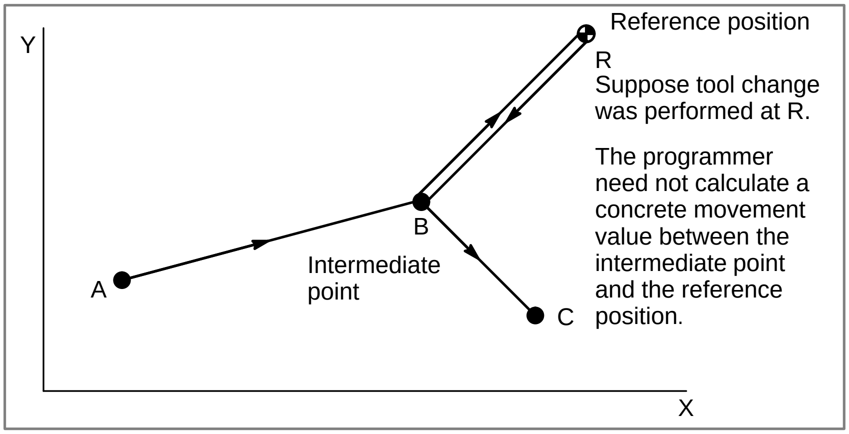
G28 IP_ ;

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_ ;



Suppose tool change was performed at R.

The programmer need not calculate a concrete movement value between the intermediate point and the reference position.

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

$$\mathbf{G30}\begin{Bmatrix}\mathbf{P2}\\ \mathbf{P3}\\ \mathbf{P4}\end{Bmatrix}\ \mathit{IP}\ \_\ ;$$

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

6.6 FLOATING REFERENCE POSITION RETURN (G30.1)

It is possible to return the tool to the floating reference position by commanding the G30.1.

The floating reference position is located on the machine and can be a reference position of some sort of machine operation. It is not always a fixed position but may vary in some cases. The floating reference position can be set using the soft keys of MDI and can be memorized even if the power is turned off.

Generally, the position where the tools can be replaced on machining center, milling machine is a set position on top of the machinery. The tools cannot be replaced at any machine angle. Normally the tool replacement position is at any of the No. 1 to No. 4 reference position. The tool can be restored to these positions easily by G30 command. However, depending on the machine, the tools can be replaced at any position as long as it does not contact the work piece.

In lathes, the tool can generally be changed at any position unless it touches the workpiece or tailstock.

For machinery such as these, in order to reduce the cycle time, it is advantageous to replace tools at a position as close as possible to the work. For this purpose, tool replacement position must be changed for each work shape and this feature can be easily realized by this function. Namely, the tool replacement position which is suitable for works can be memorized as the floating reference position and it is possible to return the tool to the tool replacement position easily by commanding the G30.1.

Format

G30.1 IP_ ;

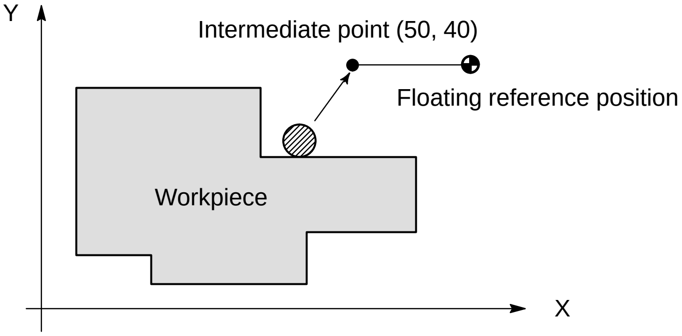
IP : It is the intermediate point to the floating reference position and is commanded by an absolute value or an incremental value.

Explanations

When the G30.1 is commanded, the axis commanded is set to the intermediate point with rapid traverse at first and then is set to the floating reference position from the intermediate point with rapid traverse. The positioning to the intermediate point or to the floating point is performed at rapid traverse for each axis (non-linear positioning). The floating reference position return completion signal is output after completing the floating reference position return.

Examples

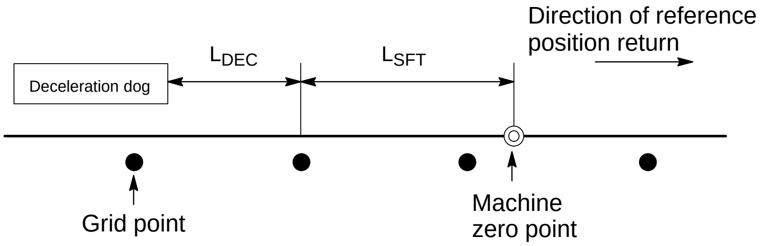
```
G30.1 G90 X50.0 Y40.0 ;
```



6.7

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.)

6.8

BUTT–TYPE REFERENCE POSITION SETTING

The butt–type reference position setting function 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.

6.9
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.

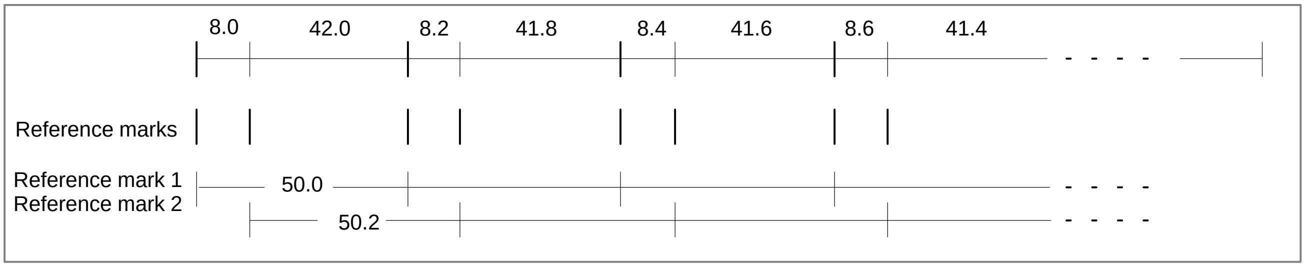


Fig. 6.9 Sample linear scale with absolute addressing reference marks

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_ ;
```

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 one of 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)

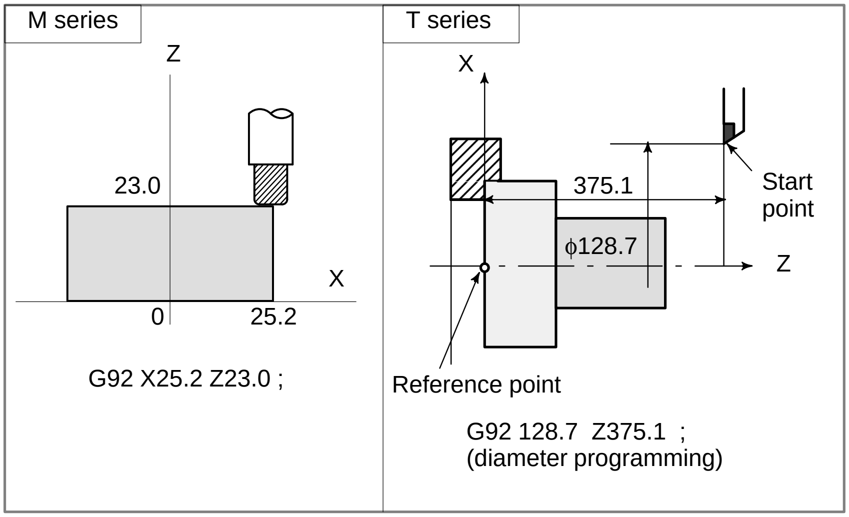
Format

(G90) G92 *IP* _ ;

Examples

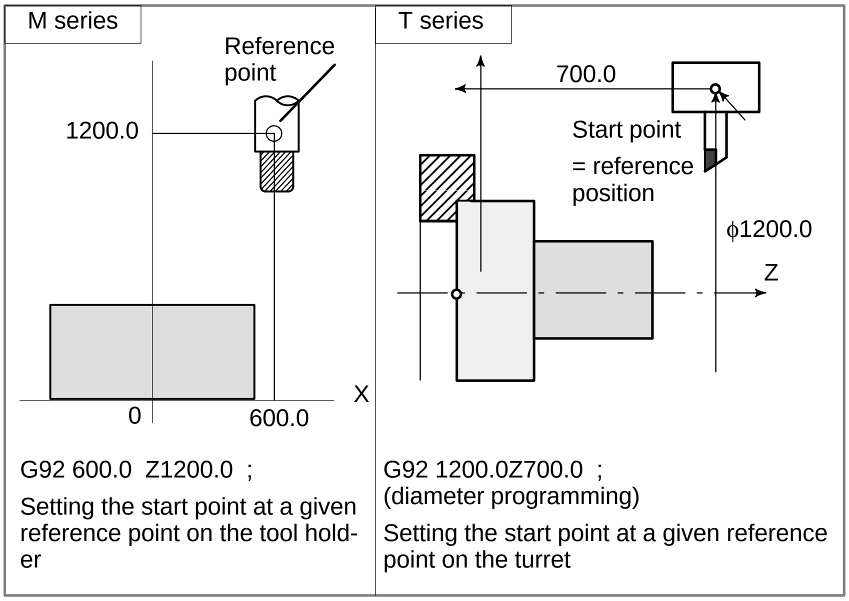
- **Example 1**

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 point 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 point is moved to a specified position. To move the tool tip to a specified position, compensate the distance between the reference point and the tool tip by using tool length compensation (for the M system) or tool offset (for the T system).



G92 600.0 Z1200.0 ;

Setting the start point at a given reference point on the tool holder

G92 1200.0Z700.0 ;
(diameter programming)

Setting the start point at a given reference point 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** **T series**

(Shift of a workpiece coordinate system)

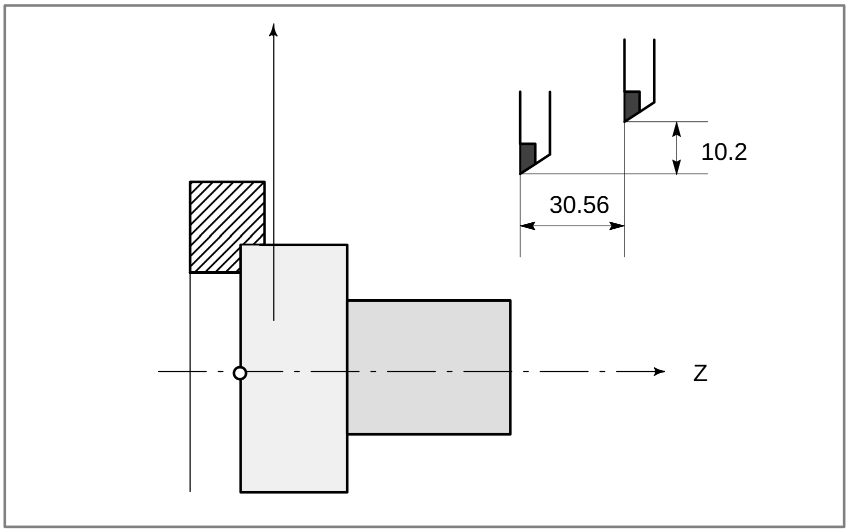
A workpiece coordinate system can be shifted by using the following command:
When this command is specified, a new coordinate system is created so that the current coordinate value (x, z) at a given point on the tool (for example, the tool tip) becomes (x+u, z+w).
For the x and u values, diameters must 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 so that the current tool position at the reference position becomes a desired position which is set using a parameter in advance. This functions as if G92IP__; were specified at the reference position.

This function can be used when the workpiece coordinate system function is not provided.

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

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

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.

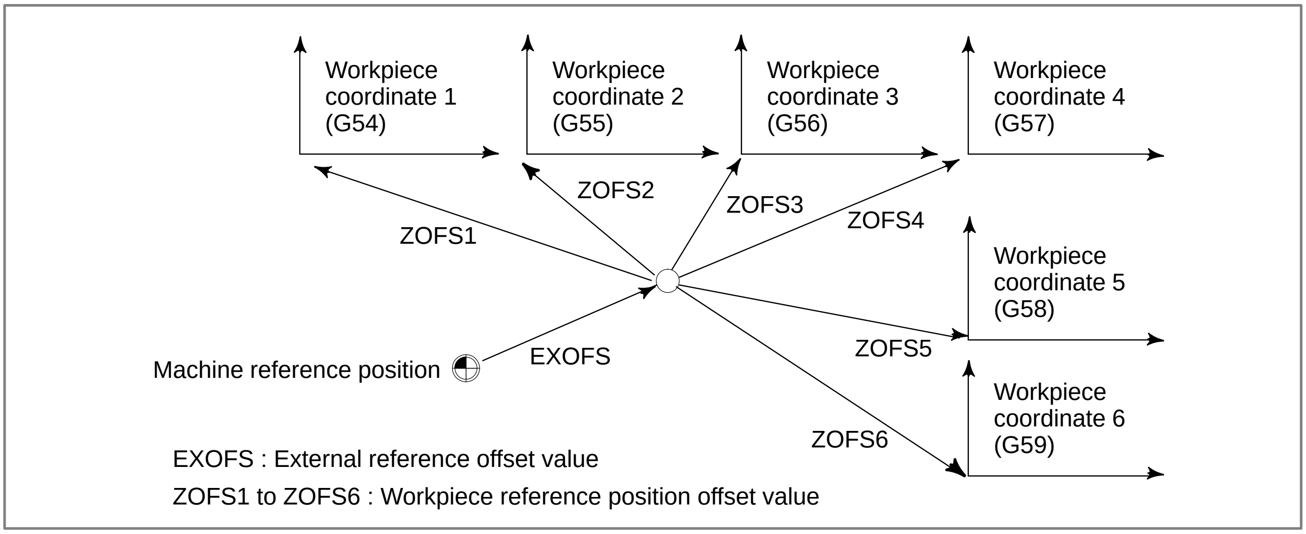
There are two setting methods.

- **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).

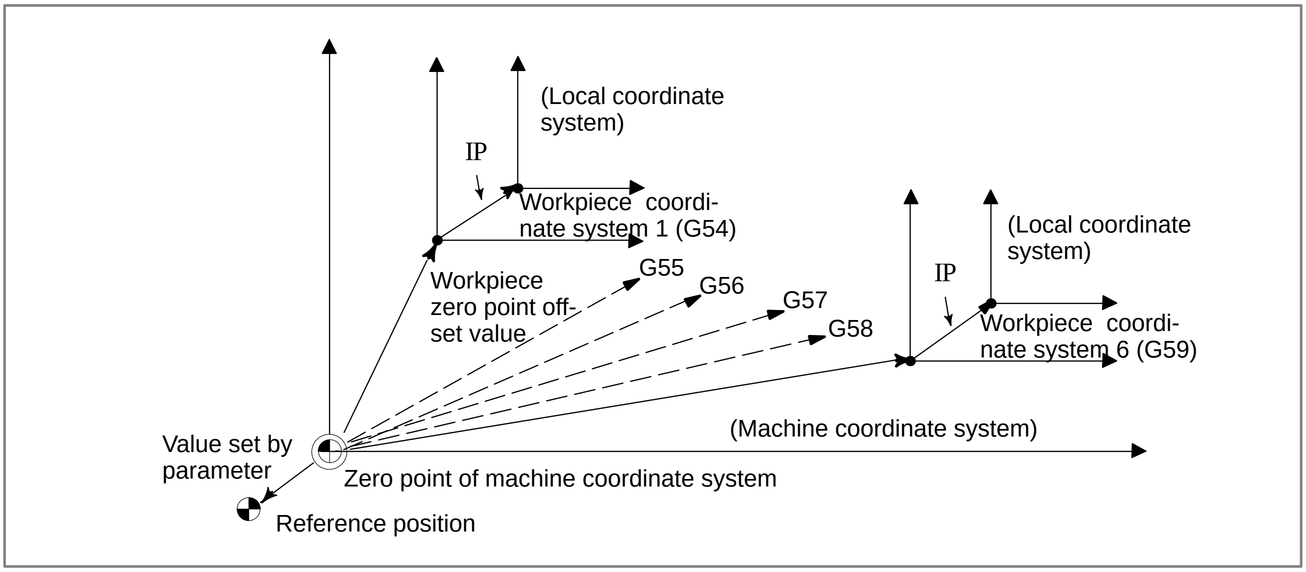


EXOFS : External reference offset value

ZOFS1 to ZOFS6 : Workpiece reference position offset value

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 (G54.1 OR G54)

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.
Up to 300 additional workpiece coordinate systems can be used.

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.

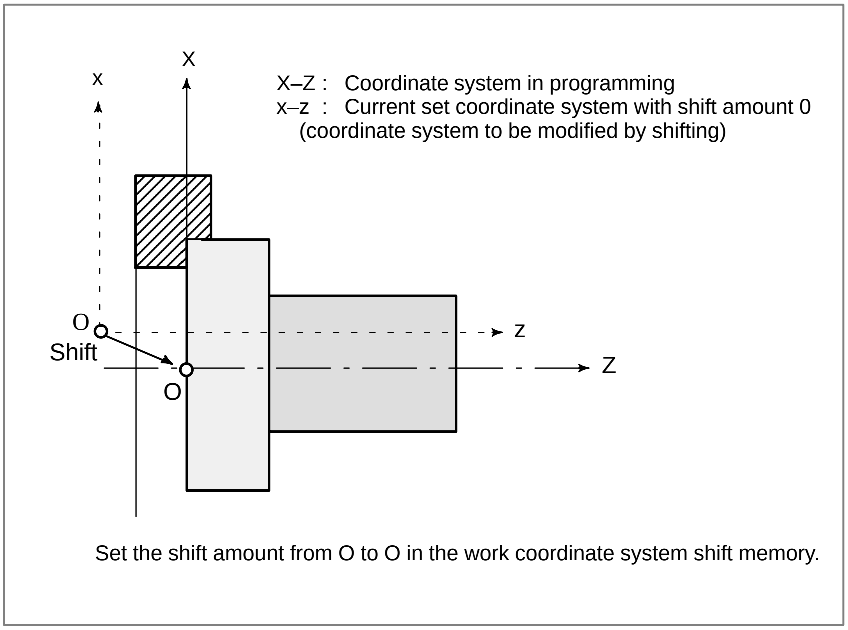
- **Workpiece coordinate system preset by MDI operation**

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 work 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

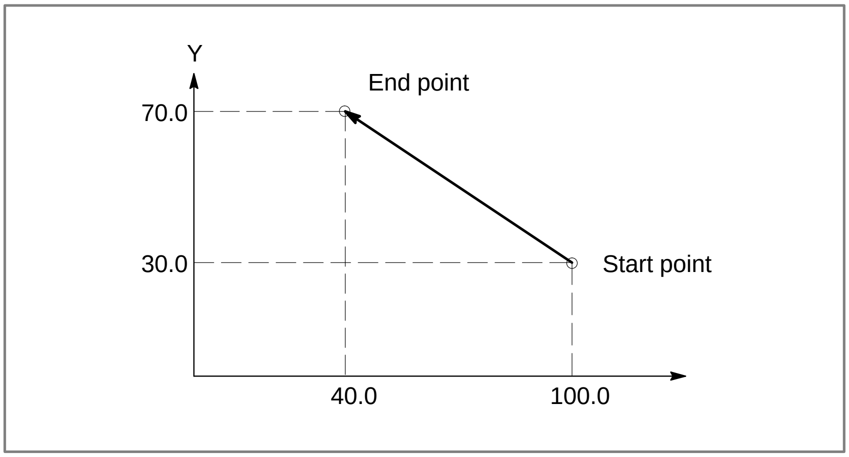
COORDINATE VALUE AND DIMENSION

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 X60.0 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

X_ W_ ;

Incremental command (Z axis move command)
Absolute command (X axis move command)

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.

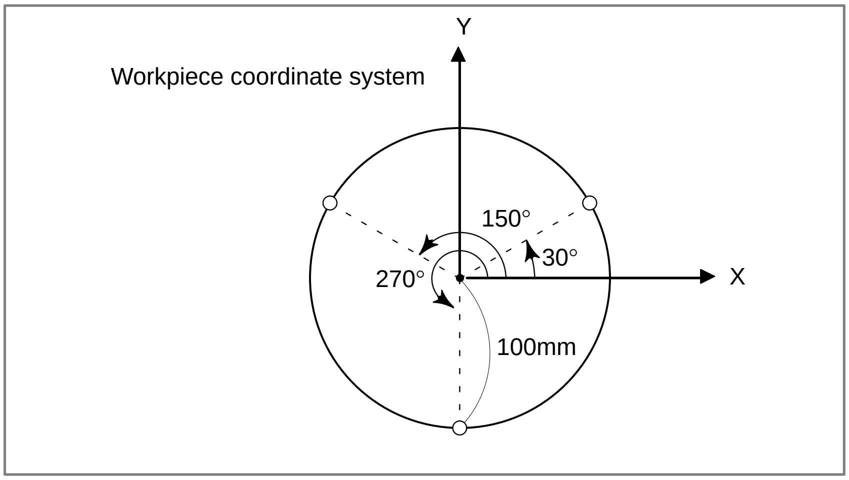
G15 : Polar coordinate system command cancel
G16 : Polar coordinate system command

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.

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.

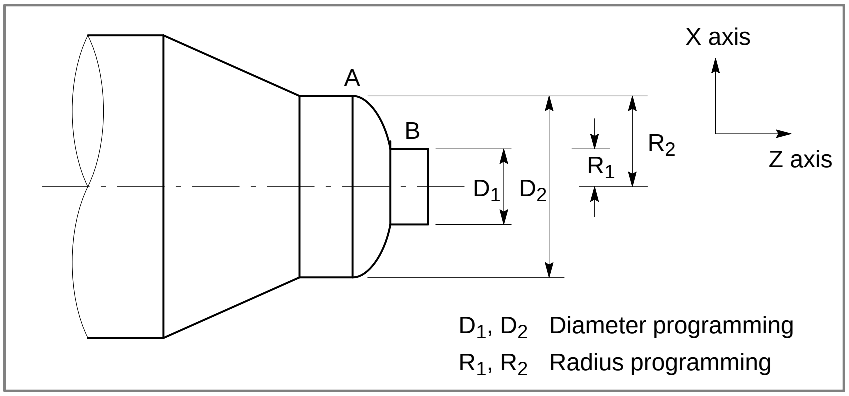
Values can be specified both with and without decimal point in a single program.

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 ROTATION 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 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:

G92 S_ ; (where, S-- is the maximum spindle speed in rpm)

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

Generally, a machine that does not have (or which does not use) a position coder cannot perform feed per rotation under constant surface speed control. When a certain parameter is set, such a machine can perform feed per rotation under constant surface speed control, assuming that S 12–bit code signals R01O to R12O represent a specified spindle speed. (Constant surface speed control without a position coder)

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

NOTE

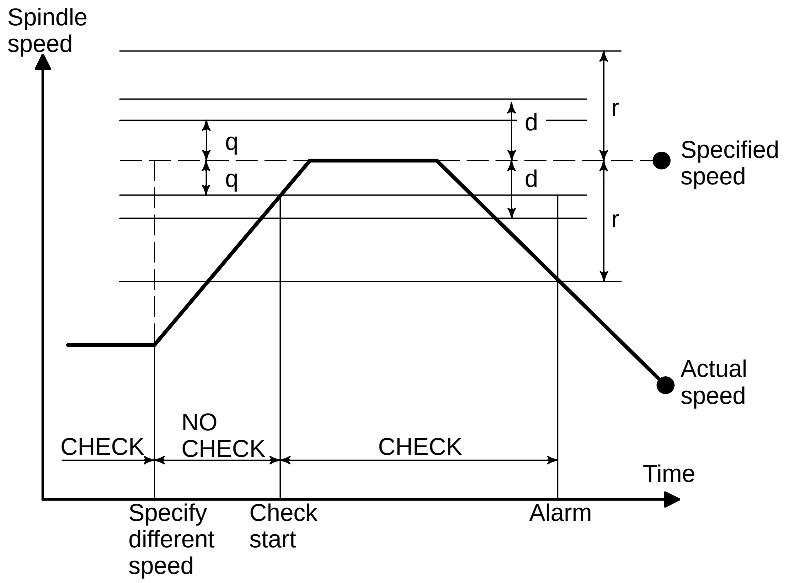
1. The value of P, Q, and R remains after the power off.
2. The actual spindle speed is calculated by the return pulses generated from the position coder attached to the spindle.

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)

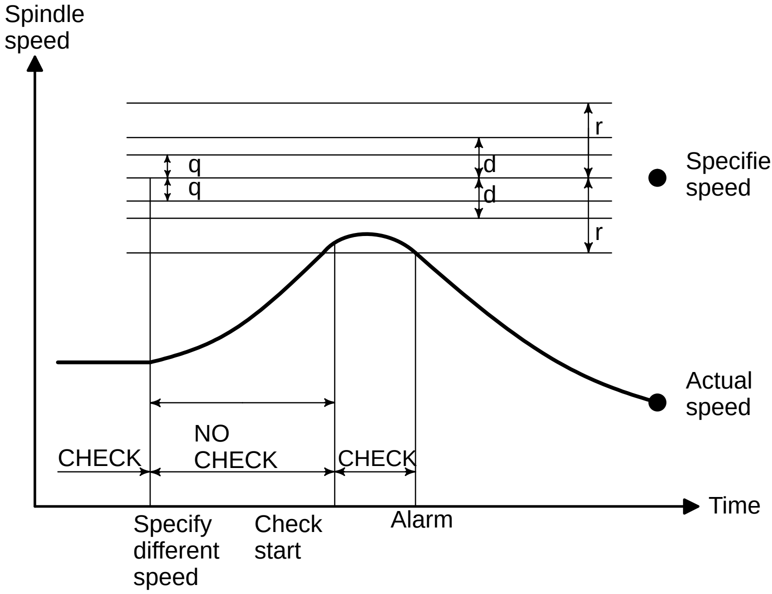
Actual speed : Speed detected by position coder

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.
With the 2–path control, two spindles can be controlled. If spindle contour control is performed for the two spindles, one spindle contour control axis must be placed for each path. Two spindle contour control axes cannot be placed for one path.

- **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 three spindles can be controlled. The three spindles are called the first, second, and third spindles. The first and second spindles are made up of serial interface spindles, and the third spindle is of an analog interface spindle. (The second or third spindle may be omitted from the configuration.)

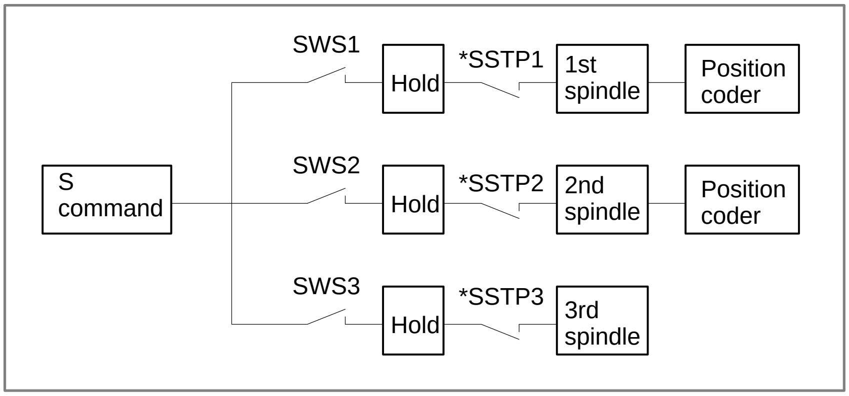
A spindle speed is specified with a 5-digit numeric value following S. This command functions on the spindle selected by spindle selection signals (SWS1 to SWS3). More than one spindle can be selected so that they can be rotated at the same time by specifying the same command.

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.
- When two–path control is performed with the M series, spindle commands and position coder feedback signals cannot be changed between the paths (spindle command select signals SLSPA and SLSPB, and spindle feedback select signals SLPCA and SLPCB are not supported).



9.12
SPINDLE SYNCHRONIZATION 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 synchronization control function is provided to provide synchronization 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.

9.15
THREE–SPINDLE SERIAL OUTPUT (ONLY FOR SINGLE–PATH CONTROL)

When single–path control is performed, three serial spindles can be connected.

The third serial spindle operates as an ordinary third analog spindle. For the third, as well as the first and second serial spindles, all the functions supported by the serial spindle control unit (spindle orientation, spindle output switching, and spindle switching) can be used.

When the third spindle orientation function is used, stop–position external–setting type orientation can also be performed for the third spindle.

9.16
SIMPLE SPINDLE SYNCHRONOUS CONTROL

In simple spindle synchronous control mode, the second spindle can be controlled as a slave axis of the first spindle.

Thus, control based on the Cs contour axis control function, rigid tapping function, and spindle positioning function (T series) can be exercised over the second spindle, under the control of the first spindle.

Note, however, that unlike spindle synchronous control, simple spindle synchronous control does not guarantee synchronization between the first and second spindles.

To realize simple spindle synchronous control, two serial spindle systems, both of which support two–spindle connection, are required. Moreover, both spindles must be fitted with the spindle–related hardware, such as detectors, required for the functions used with simple spindle synchronous control (Cs contour axis control function, rigid tapping function, and spindle positioning function (T series)).

10 TOOL FUNCTIONS

10.1
T CODE OUTPUT

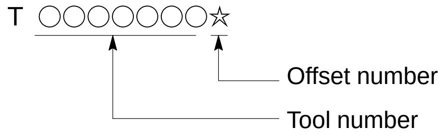
M series

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.

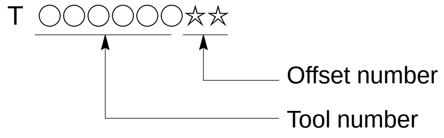
T series

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:



When the last two digits are used to specify the offset 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
Addition of Tool Pairs for Tool Life Management <512 Pairs (M series) / 128 Pairs (T series)>

The number of groups that can be registered in the tool life management function and the allowable number of tools per group can be selected from the following four combinations. One of the combinations is selected using a parameter.

M series		T series	
Number of groups	Number of tools	Number of groups	Number of tools
64	16	16	32
128	8	32	16
256	4	64	8
512	2	128	4

10.2.3

M series

Extended Tool Life Management

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.

11 MISCELLANEOUS FUNCTIONS

11.1 MISCELLANEOUS 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 MISCELLANEOUS 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.

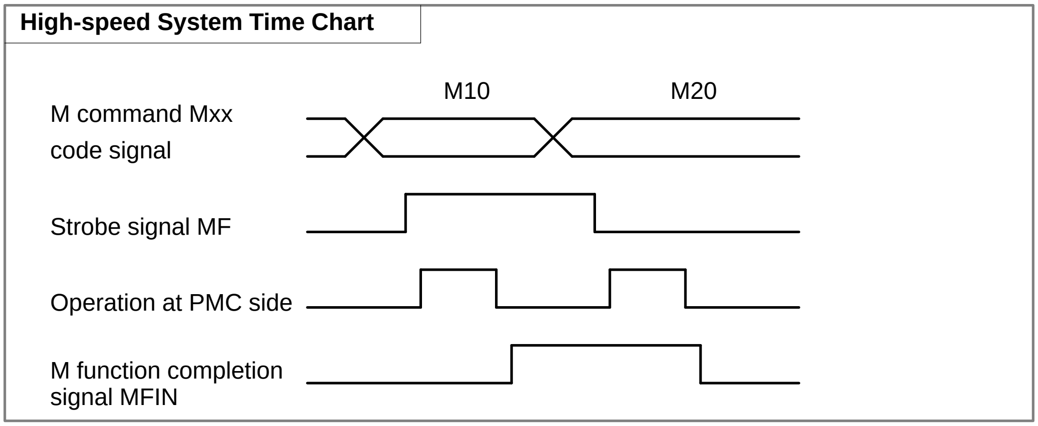
The following describes an example of auxiliary function M code command. The same applies to the T, S, and B (second auxiliary function) functions.

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.

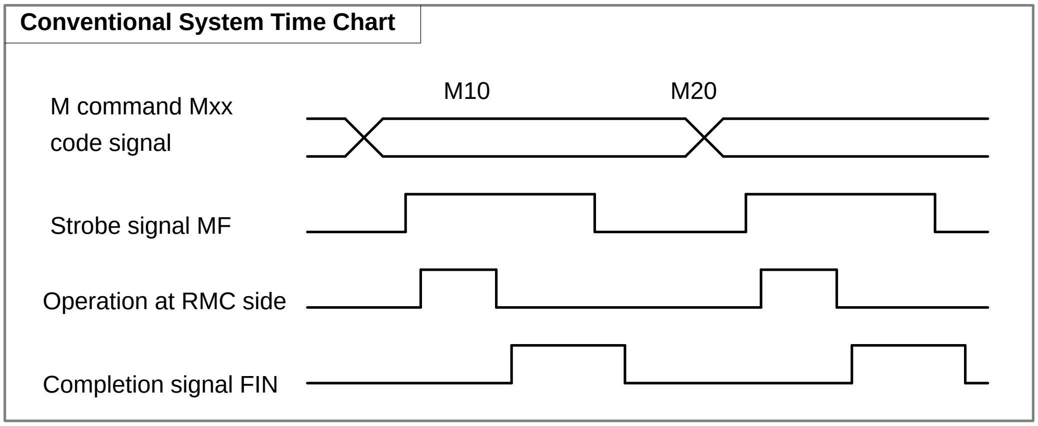
In the usual system, if the leading edge (from “0” to “1”) of the completion signal FIN of M/S/T/B is received and then the trailing edge (from “1” to “0”) of the signal FIN is received, it is considered that the operation has been completed. However, in this system, the operation is considered to have been completed by a single change of completion signal MFIN.

Example) M10;
M20;

High-speed System Time Chart



Conventional System Time Chart



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.

11.5
M CODE GROUP CHECK FUNCTION

The M code group check function checks if a combination of multiple M codes (up to three M codes) contained in a block is correct.

This function has two purpose. One is to detect if any of the multiple M codes specified in a block include an M code that must be specified alone. The other purpose is to detect if any of the multiple M codes specified in a block include M codes that belong to the same group.

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 (when the 8–digit program number option is used, however, eight digits following address O) after the address O. Program number of the program currently under execution is always displayed on the CRT screen. 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.

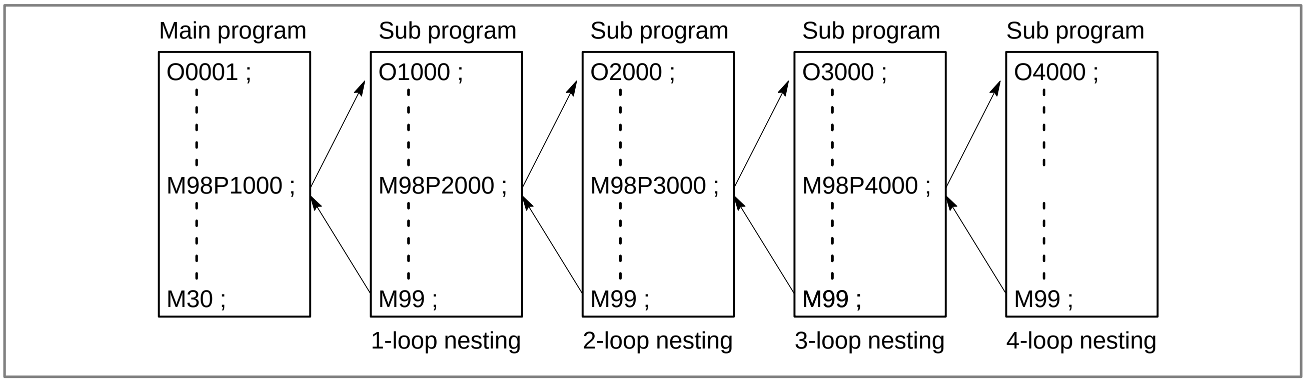
Example) 01234 (PROGRAM FOR ATC);

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.



Format

Sub program call

M98 P ○○○ ○○○○ **;**

Number of repetitive calls (○○○)

Subprogram number (○○○○)

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

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 (*1)	1–9999	1–9999
Sequence number		N	1–99999	1–99999
Preparatory function		G	0–99	0–99
Dimension word, Setting unit	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, Setting unit	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, Setting unit	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 repeats		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 (*1)	1–9999	1–9999
Sequence number		N	1–99999	1–99999
Preparatory function		G	0–99	0–99
Dimension word, Setting unit	IS–B	X, Y, Z, U, V, W, A, B, C, I, J, K, R	±99999.999mm ±99999.999deg	±9999.9999inch (Note2) ±99999.999deg
	IS–C		±99999.9999mm ±99999.9999deg	±999.99999inch (Note2) ±9999.9999deg
Feed per minute, Setting unit	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	0.0001–500.000 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, Setting unit	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 repeats		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.Y200.;

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.

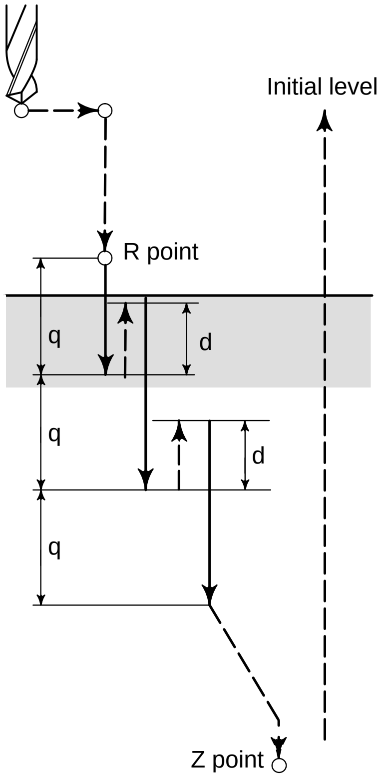
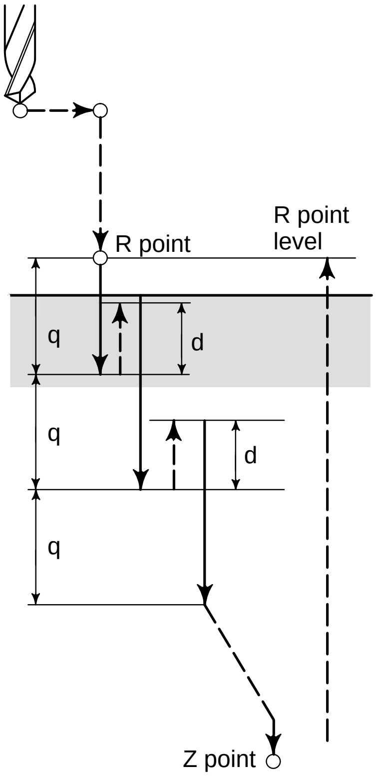
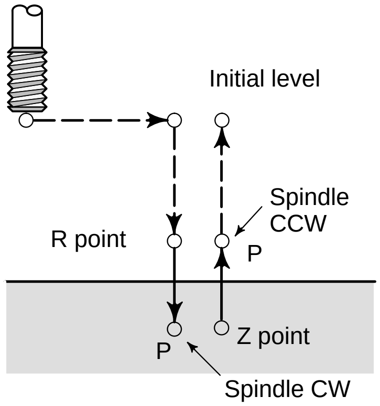
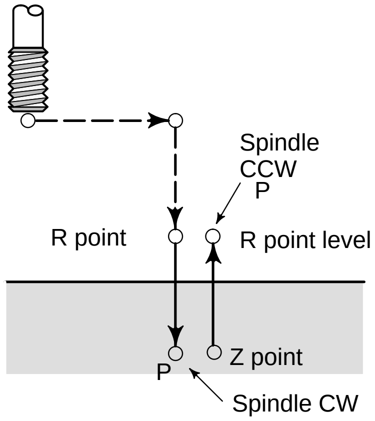
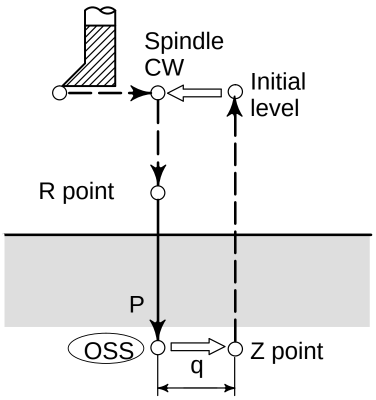
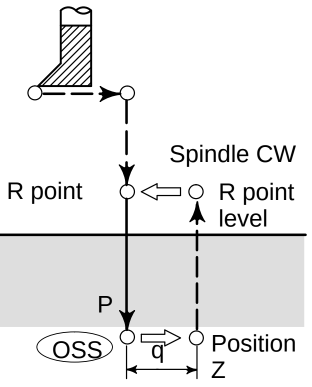
It is not always necessary to command G17, G18, G19 in the same block as G73 - G89.

NOTE

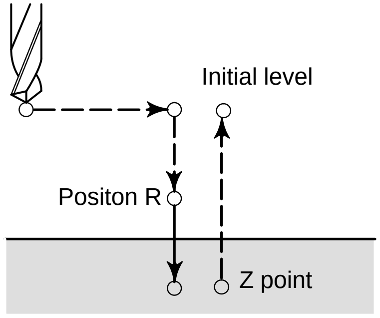
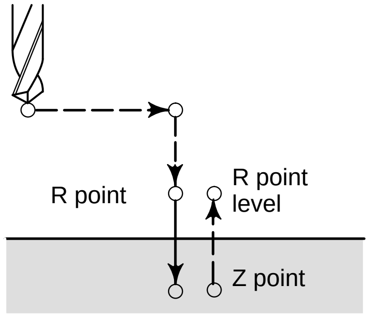
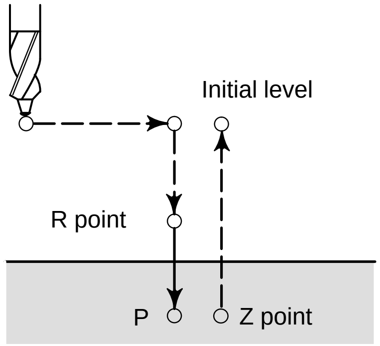
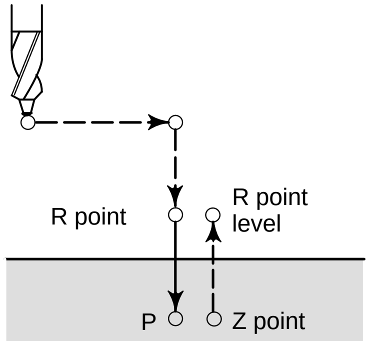
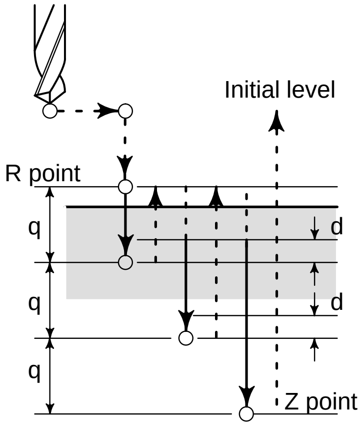
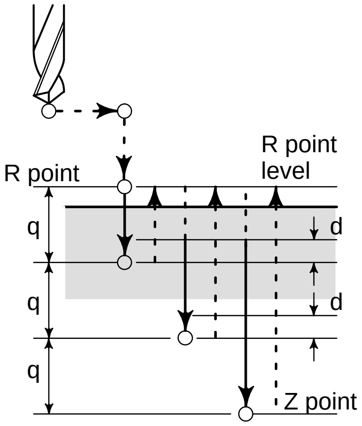
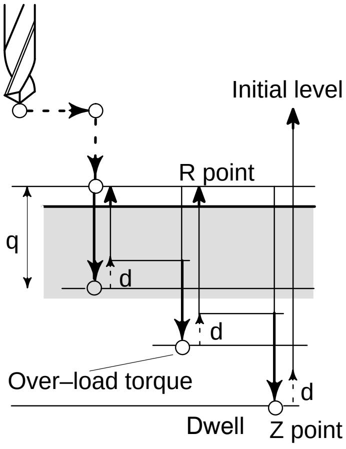
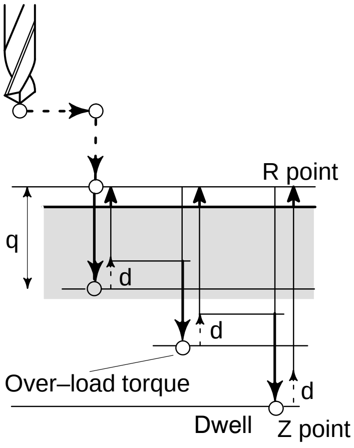
Z axis can always be appointed the drilling axis by parameter setting.

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.

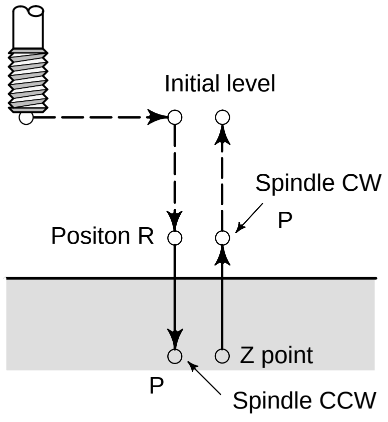
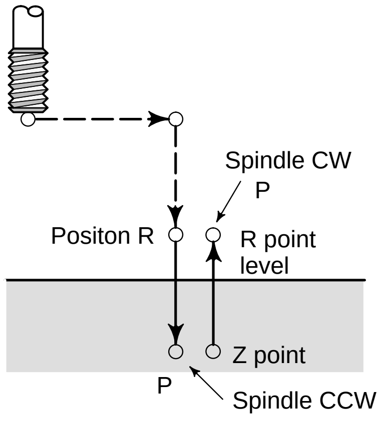
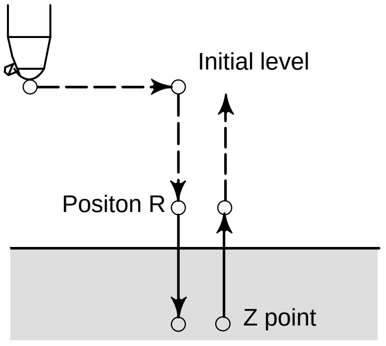
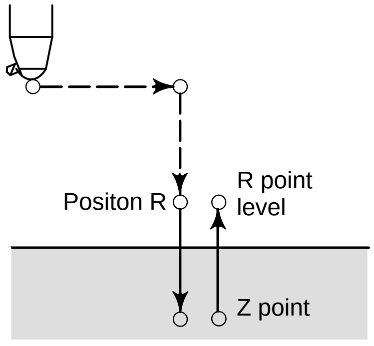
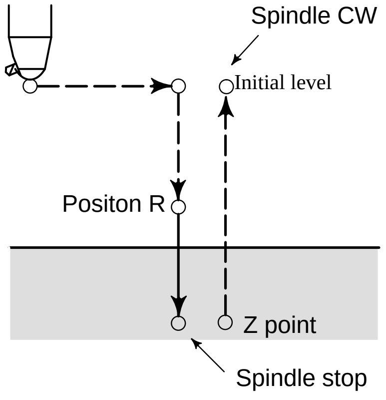
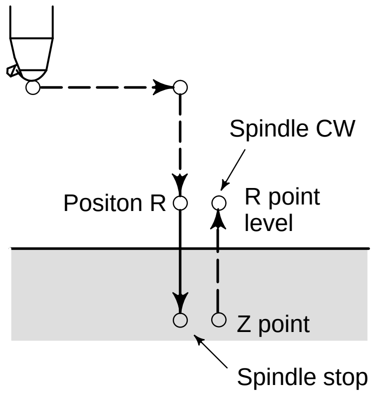
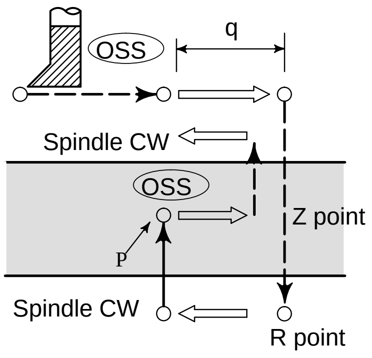
13 types of canned cycles (1/4)

<table>
<tr><th rowspan="2">G code</th><th colspan="2">Operation</th><th rowspan="2">Function</th></tr>
<tr><th>G98 mode</th><th>G99 mode</th></tr>
<tr><td>G73</td><td> </td><td> </td><td>High–speed peck drilling cycle
(Note 1)</td></tr>
<tr><td>G74</td><td> </td><td> </td><td>Counter tapping cycle</td></tr>
<tr><td>G76</td><td> </td><td> </td><td>FIne boring cycle</td></tr>
</table>

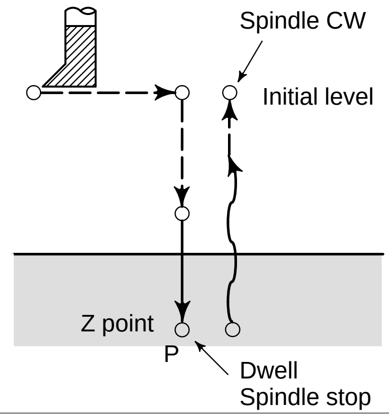
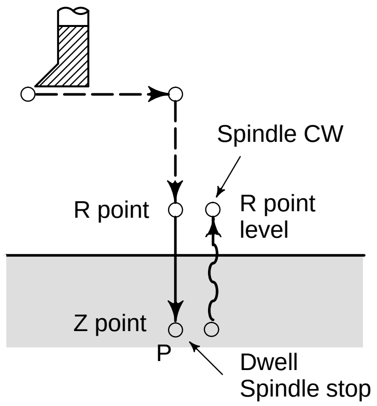
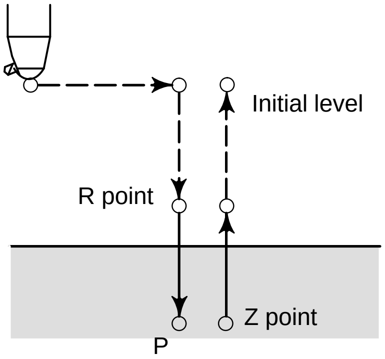
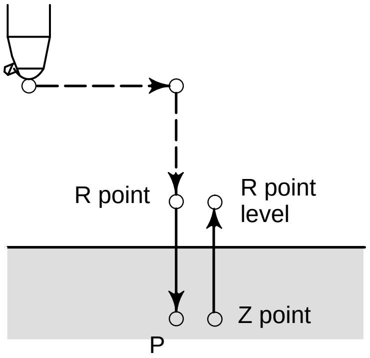
13 types of canned cycles (2/4)

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		Function
	G98 mode	G99 mode	
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	OSS	Oriented spindle stop (Spindle stops at constant rotation position)
– – →	Rapid traverse		
∿→	Manual feed	⇨	Shift
P	Dwell	I	Initial position level
Z	Z point (Hole bottom position)	R	R point

Note 1 ”d” of G73 and G84 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.

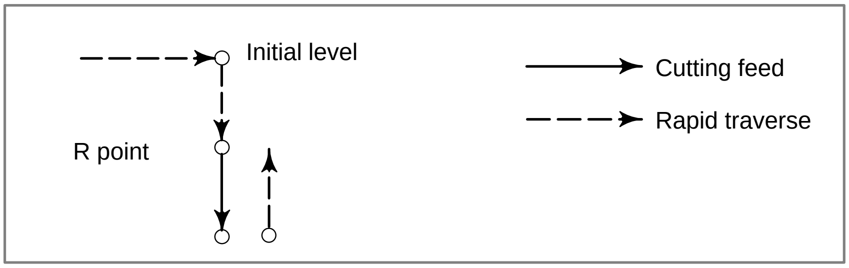
Z	:	Specify hole end position shown in the previous table.
R	:	Specify R point position shown in the previous table.
Q	:	Specify cutting quantity with G73, G83, and shift quantity with G76, G87,
P	:	Specify dwell time at the hole bottom.
K	:	Specify how may times to repeat. When specified K0, drilling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

Explanations

● R point level return (G99)

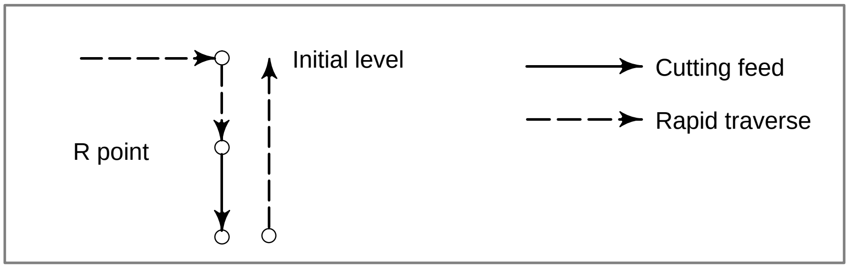
By specifying G99, return point in canned cycle is specified to R point. The drilling starts from the end point of the previous block. If the previous block has ended in the initial point, it begins from the initial point and returns to the R point.

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.



13.2 RIGID TAP

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.

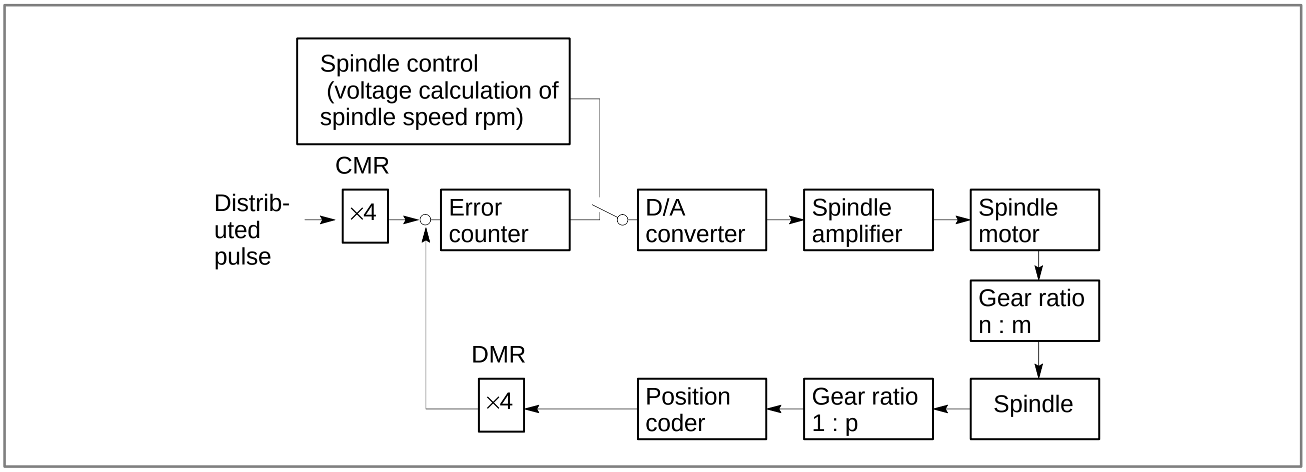
This allows the following condition to be satisfied also during acceleration/deceleration at the hole bottom and a tapping of improved accuracy to be made.

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.

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

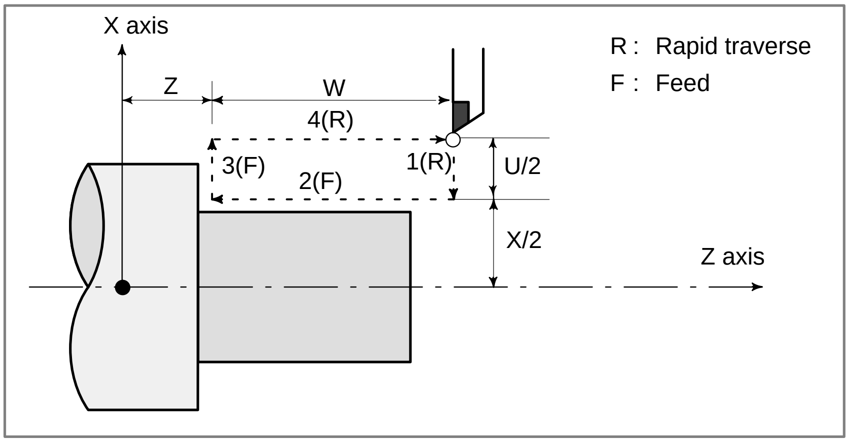
T series

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.

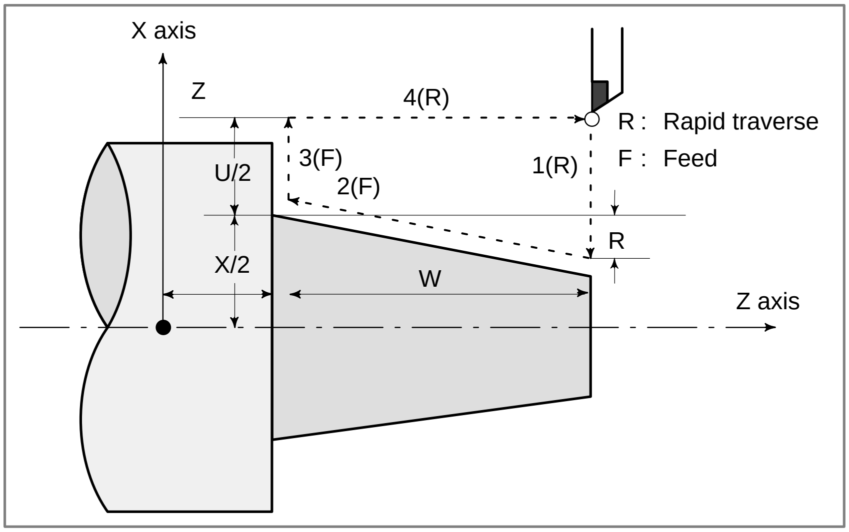


Format

```
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.



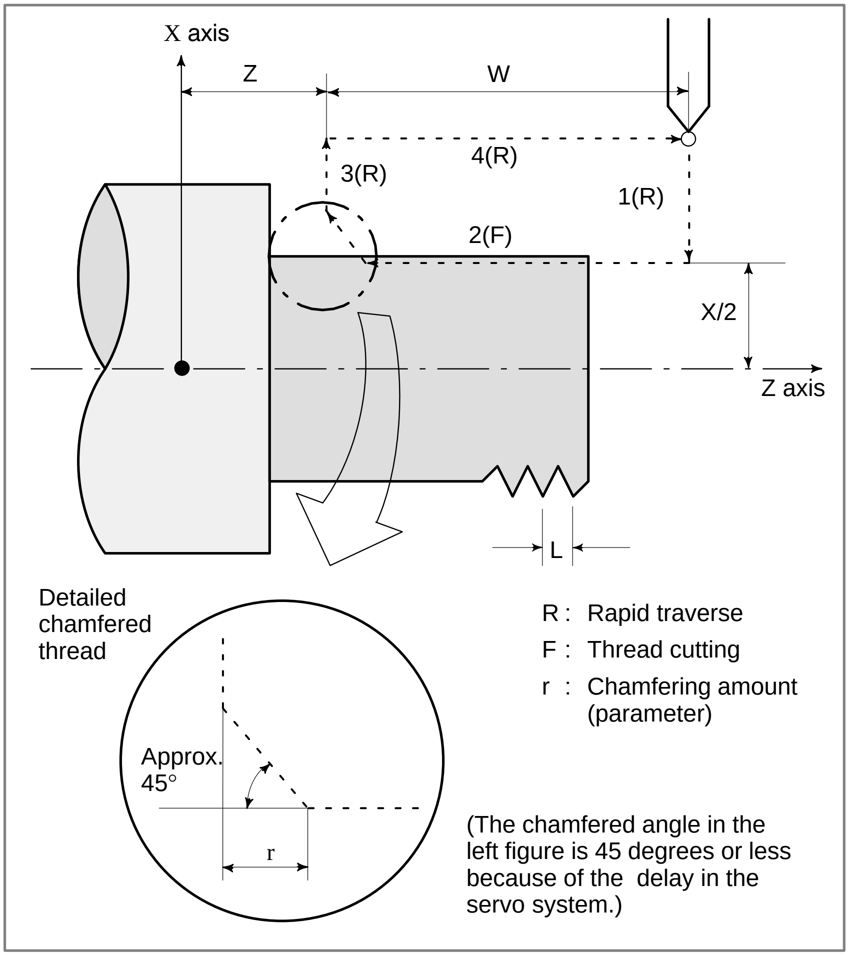
Format

```
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.



R : Rapid traverse
F : Thread cutting
r : Chamfering amount (parameter)

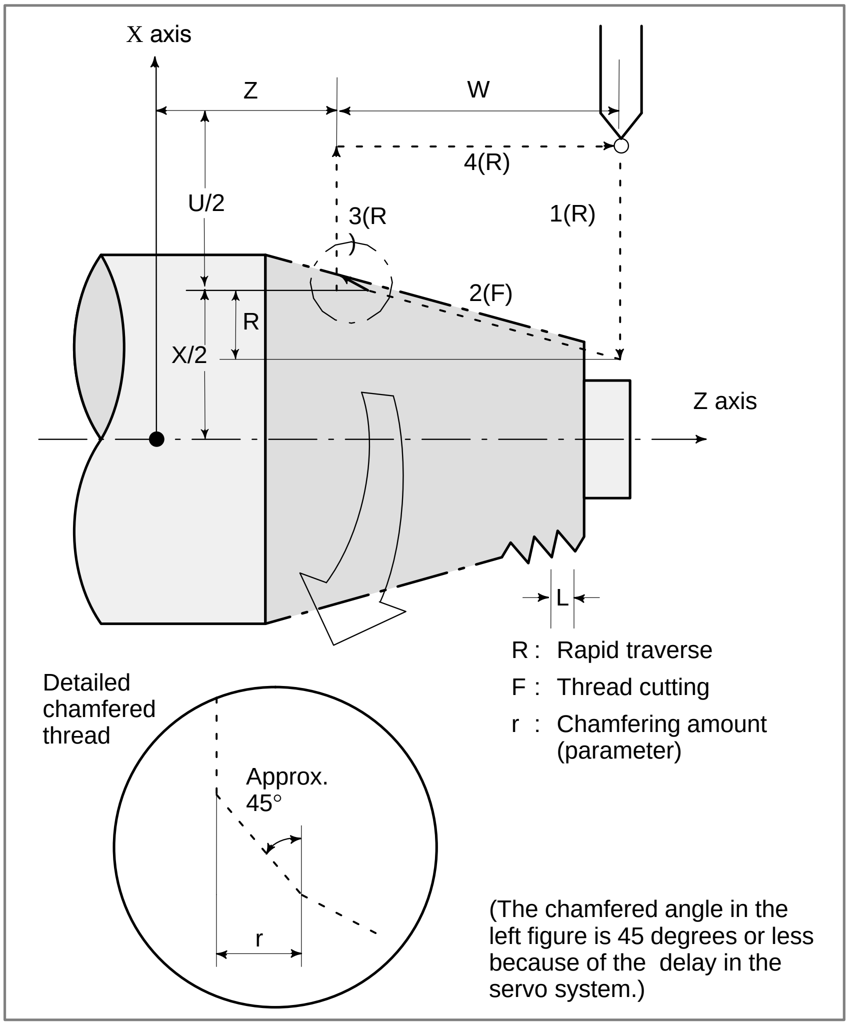
(The chamfered angle in the left figure is 45 degrees or less because of the delay in the servo system.)

Format

```
G78 X_ Z_ F_ ;
```

- **Tapered thread cutting cycle**

The command below actuates a tapered thread cutting cycle.



(The chamfered angle in the left figure is 45 degrees or less because of the delay in the servo system.)

Format

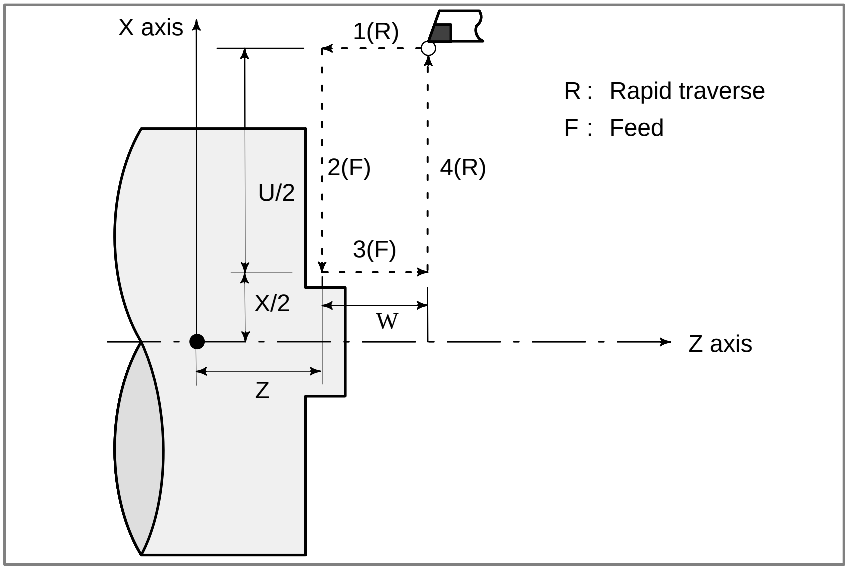
```
G78 X_ Z_ R_ F_ ;
```

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.

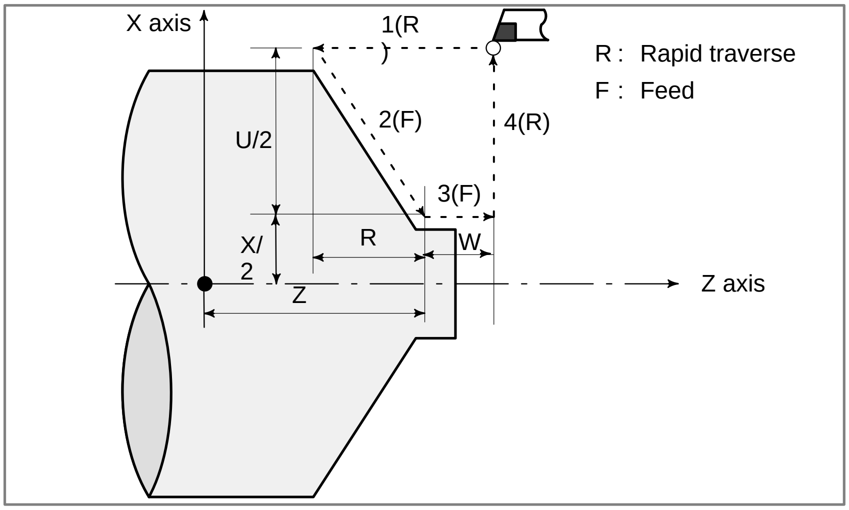


Format

```
G78 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

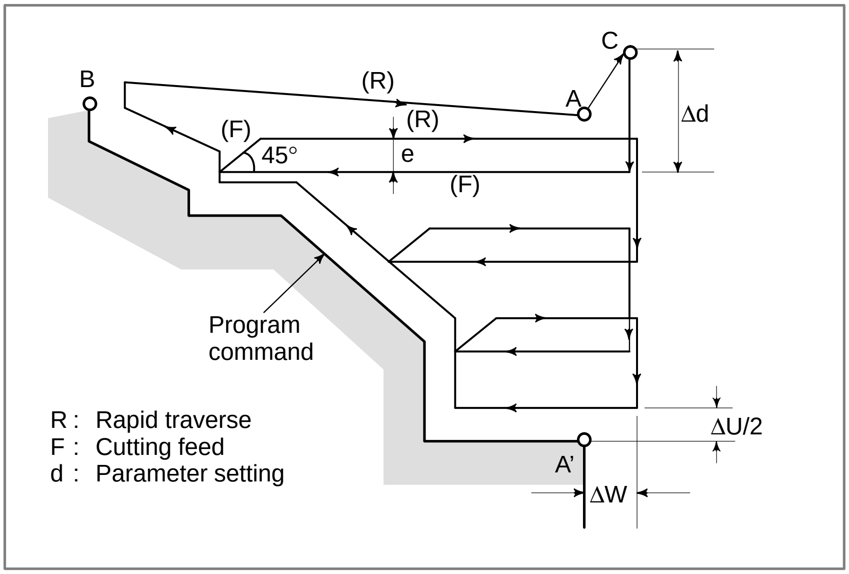
A multiple repetitive cycle is composed of several canned cycles. A tool path for rough machining, for example, is determined automatically by giving the data of the finishing work shape. A thread cutting cycle has also been prepared.

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 delta d and the finishing allowance delta U/2 and delta W.



Format

G71 U(Δd) R(e) ;

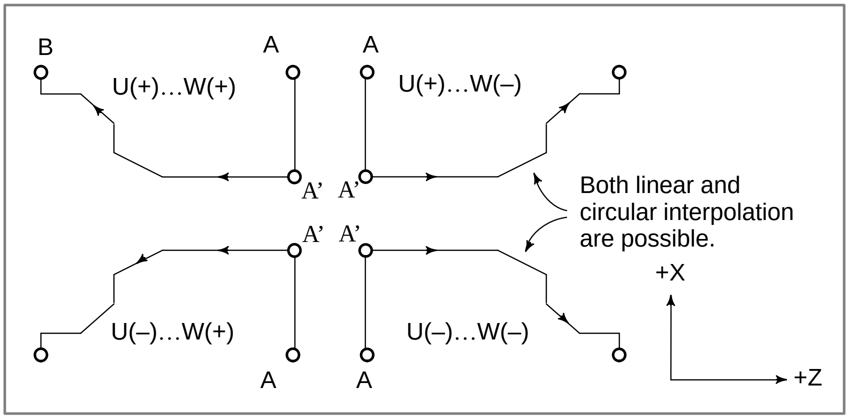
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ; (ns)

```
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.

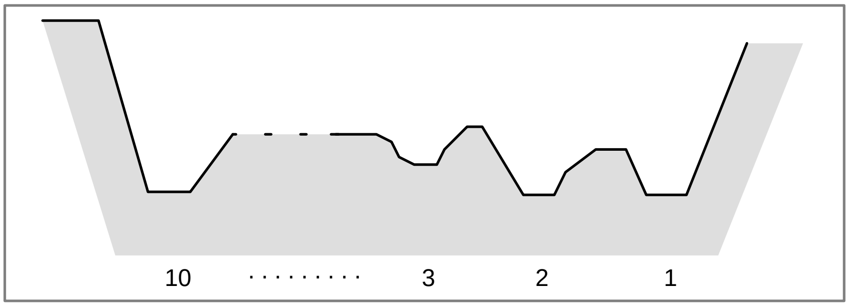
F, S, and T in the blocks of move commands from A to B are ignored and those specified in the same block as G71 or before are effective.
G96s (constant surface speed control on) and G97s (constant surface speed control off) in the blocks of move commands from A to B are ignored. A G96 or a G97 commanded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



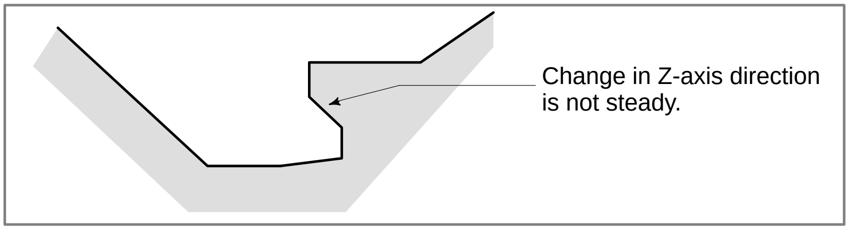
For the path from A to A', the block of sequence number ns specifies a command including G00 or G01. For the path A' to B, increase or decrease must be steady in both X-axis and Z-axis directions.
If the command for the path from A to A' is G00, the cutting along the path is performed in the G00 mode.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

- **Type II**

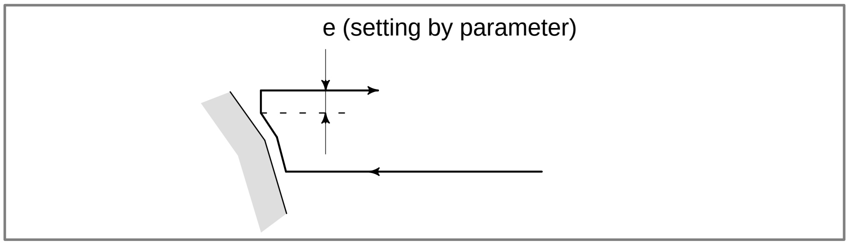
Type II differs from Type I in the following point.
Increase in X-axis direction does not need to be steady. Up to 10 pockets are allowed.



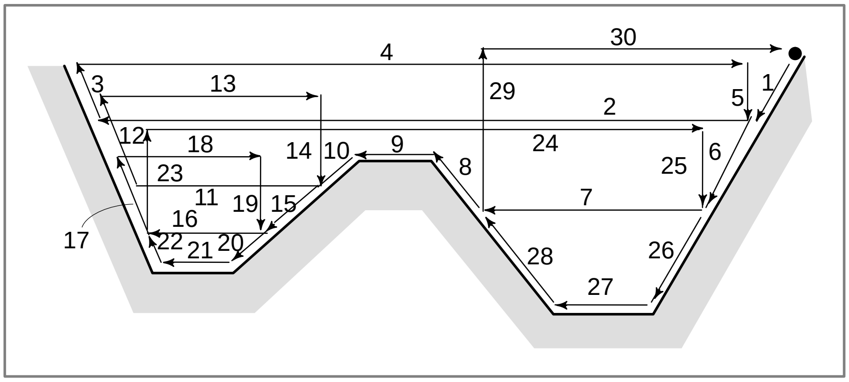
In Z-axis direction, however, increase or decrease must be steady. The following figure is not allowed for machining.



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 Δw=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

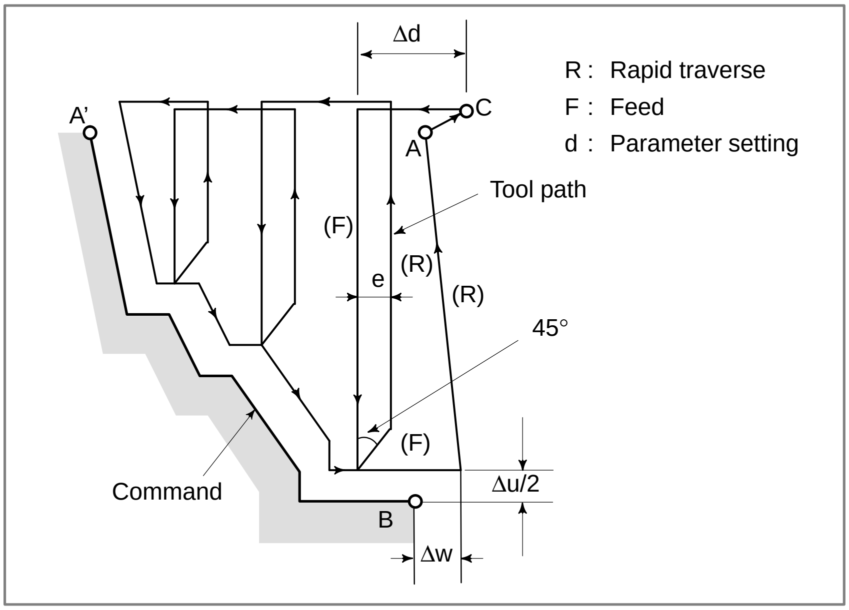
Used when two axes are specified in the first block in the repeat part.

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.



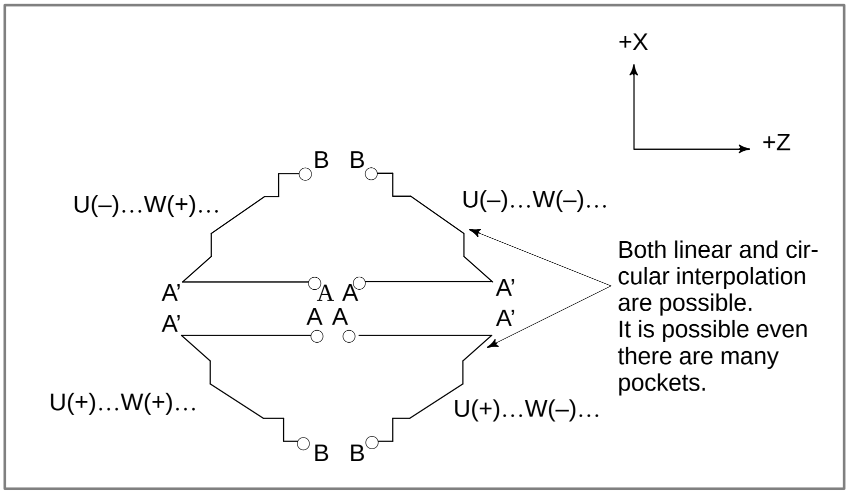
Format

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 delta U and delta W are as follows:

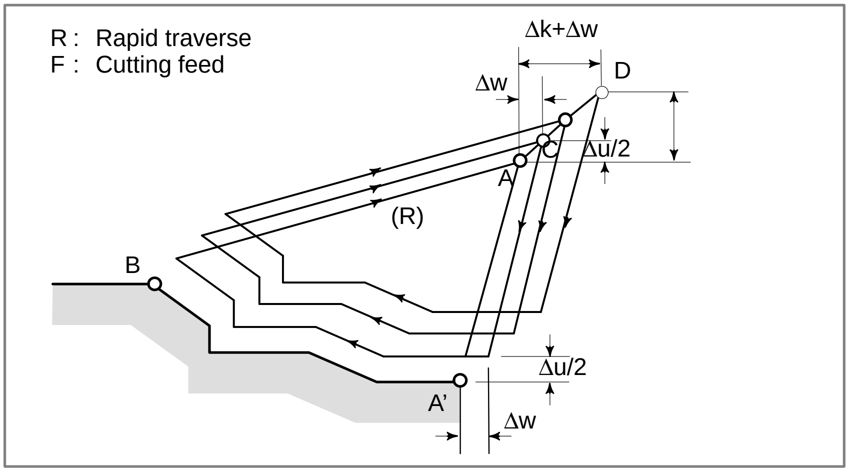


This function is effective only in memory mode.

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

NOTE

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

N_ G70 P(ns) Q(nf) ;

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

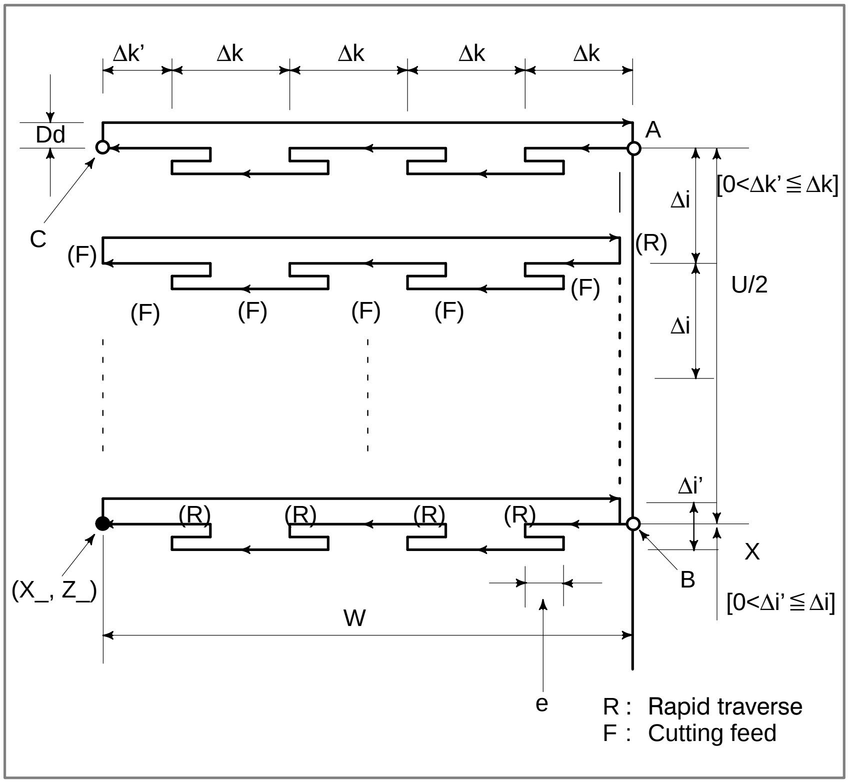
NOTE

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

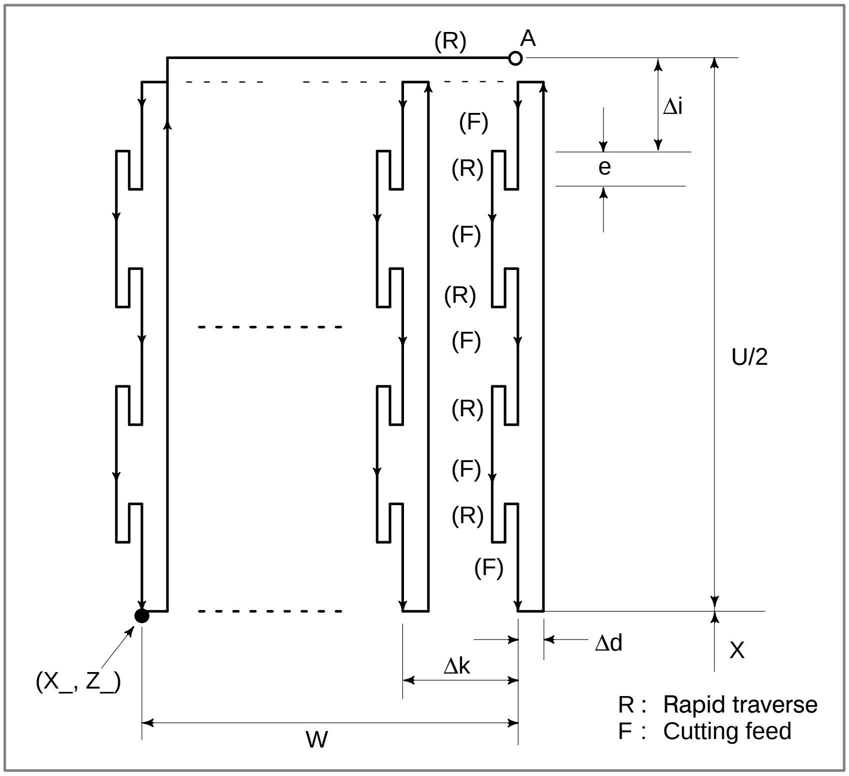
$$\textbf{G74 R}(\underline{\textbf{e}})\ ;$$

$$\textbf{G74}\begin{Bmatrix}\textbf{X\_ Z\_}\\ \textbf{U\_W\_}\end{Bmatrix}\textbf{P}(\underline{\textbf{ns}})\ \textbf{Q}(\underline{\textbf{nf}})\ \textbf{U}(\underline{\Delta\textbf{u}})\ \textbf{F}(\underline{\textbf{f}})\ ;$$

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 i 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.



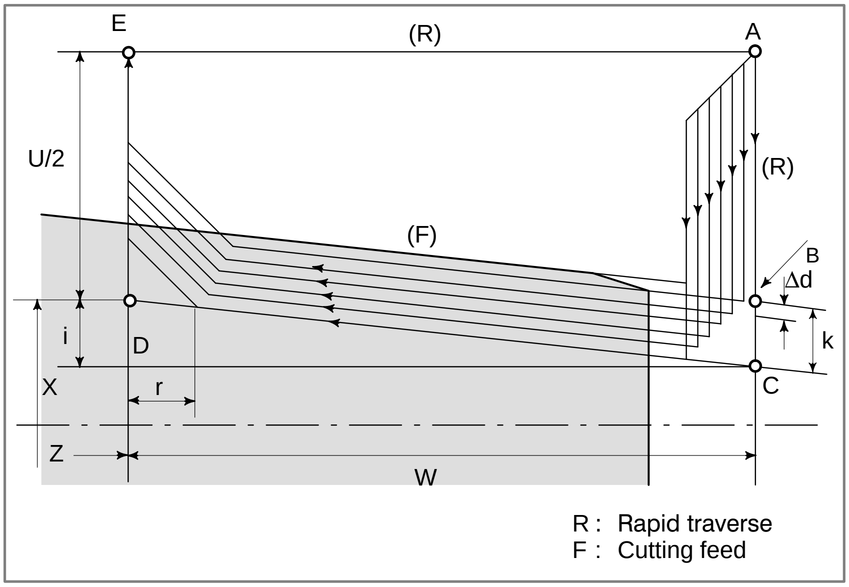
Format

$$\mathbf{G75\ R\underline{(e)}\ ;}$$

$$\mathbf{G75} \left\{\begin{matrix} \mathbf{X\_\ Z\_} \\ \mathbf{U\_W\_} \end{matrix}\right\} \mathbf{P\underline{(\Delta i)}\ Q\underline{(\Delta k)}\ R\underline{(\Delta d)}\ F\underline{(f)}\ ;}$$

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($\underline{\text{d)}}$;

$$\mathbf{G76}\left\{\begin{matrix}\mathbf{X\_\ Z\_}\\ \mathbf{U\_W\_}\end{matrix}\right\}\mathbf{R\underline{(i)\ P(k)}\ Q\underline{(\Delta d)}\ F\underline{(\ell)}\ ;}$$

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:
P02 12 60
m r a

Δbmin: Minimum depth of cut

d : Finishing allowance

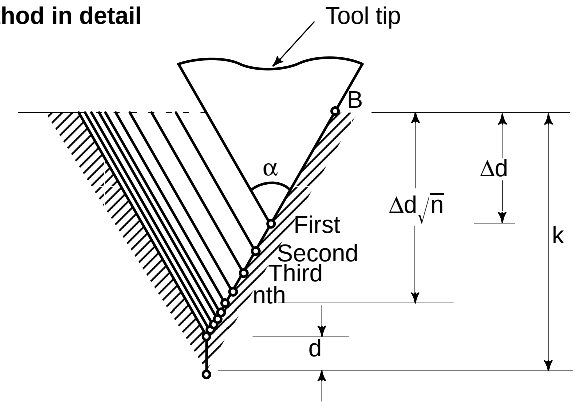
i : Difference in thread radius
Straight threading for i = 0

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)

Cutting method in detail



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

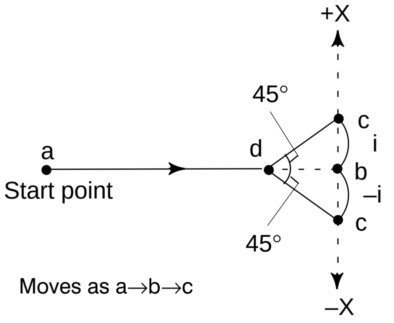
G code	Drilling axis	Drilling	Operation at hole bottom	Clearance	Use
G80					Cancel
G83	Z axis	Cutting feed Intermittent feed	Dwell	Rapid traverse	Deep drilling
G84	Z axis	Cutting feed	Spindle reverse	Cutting feed	Tapping
G85	Z axis	Cutting feed	Dwell	Cutting feed	Boring
G87	X axis	Cutting feed Intermittent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

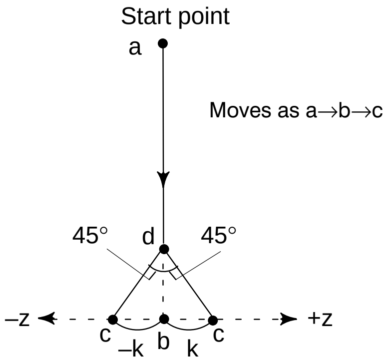
T series

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.

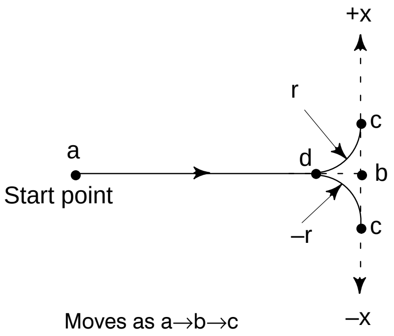
- **Chamfering Z → X**

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.	

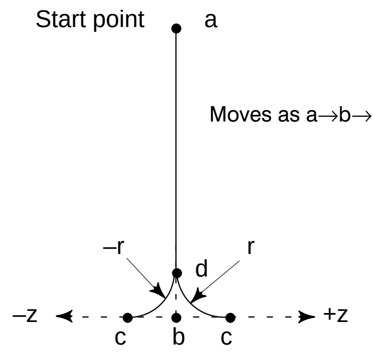
- **Chamfering X → Z**

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.	

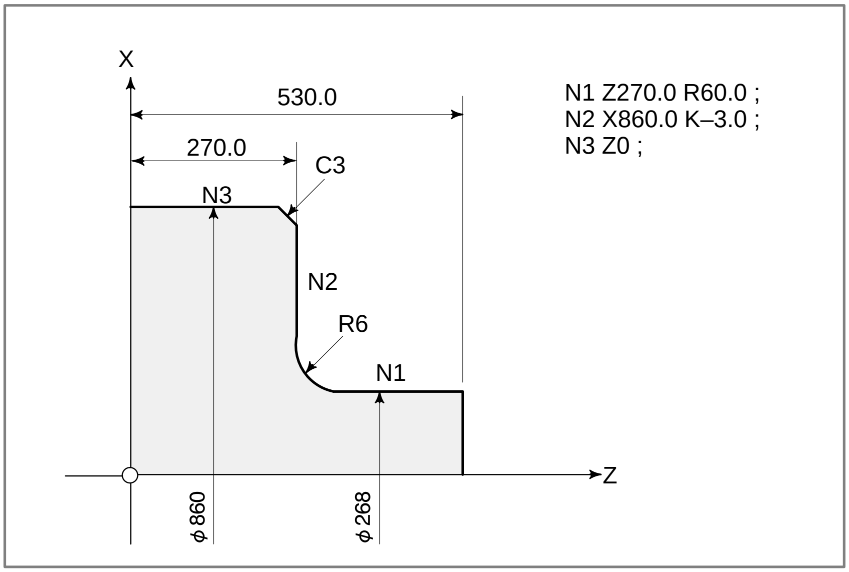
- **Corner R Z → X**

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.	

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



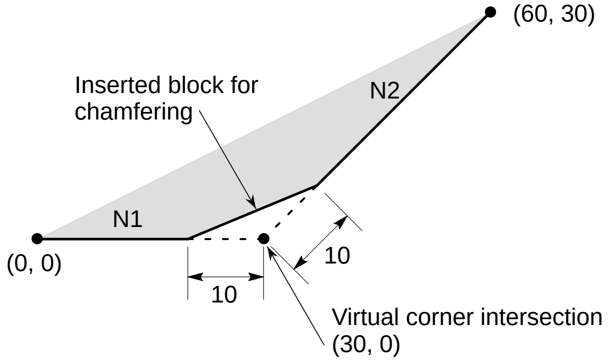
```
N1 Z270.0 R60.0 ;
N2 X860.0 K–3.0 ;
N3 Z0 ;
```

13.8 OPTIONAL ANGLE CHAMFERING/ CORNER ROUNDING

M series

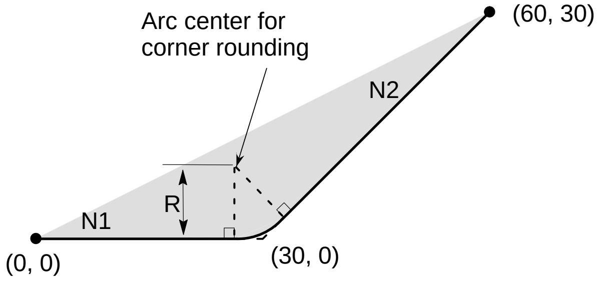
The block for chamfering or corner rounding can be inserted automatically between two optional linear interpolations, or between the linear interpolation and circular interpolation, or between two circular interpolations.
Specifying **”,C_”** inserts the block for chamfering, and specifying **”,R_”** 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.

```
N1 G91 G01 X30.0, C10.0 ;
N2 X30.0 Y30.0 ;
```



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

```
N1 G91 G01 X30.0, R10.0 ;
N2 X30.0 Y30.0 ;
```



13.9

T series

DIRECT DRAWING DIMENSIONS PROGRAMMING

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:

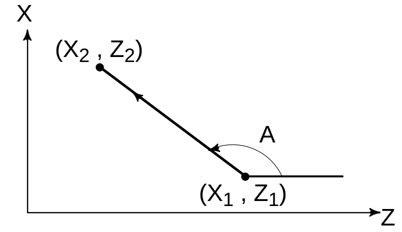
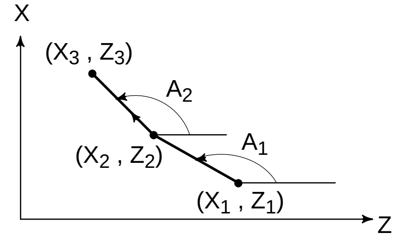
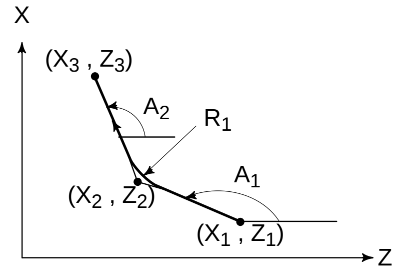
,A_
,C_
,R_

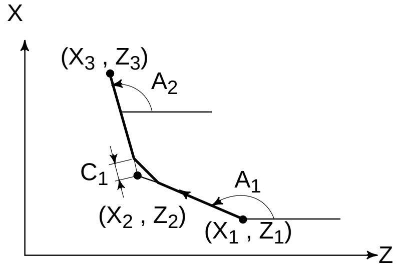
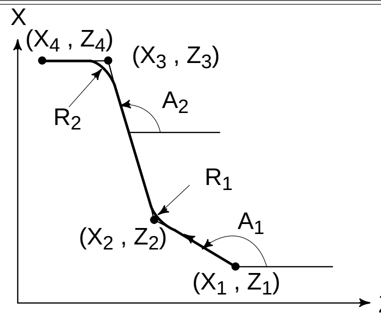
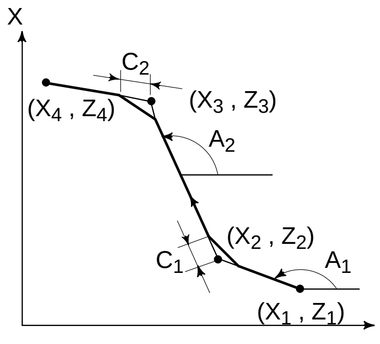
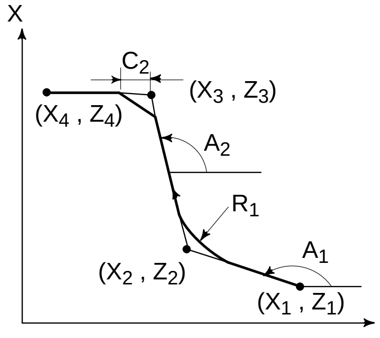
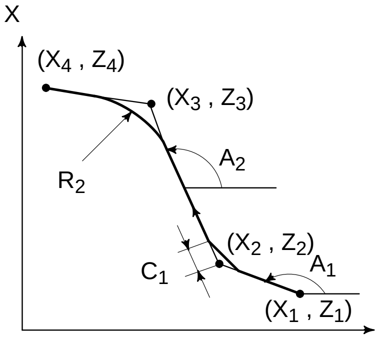
NOTE

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_ ;	

	Command	**Movement of tool**
4	X_2_ Z_2_, C_1_ ; X_3_ Z_3_ ; or , A_1_, C_1_ ; X_3_ Z_3_, A_2_ ;	
5	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_ ;	
6	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_ ;	
7	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_ ;	
8	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_ ;	

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.

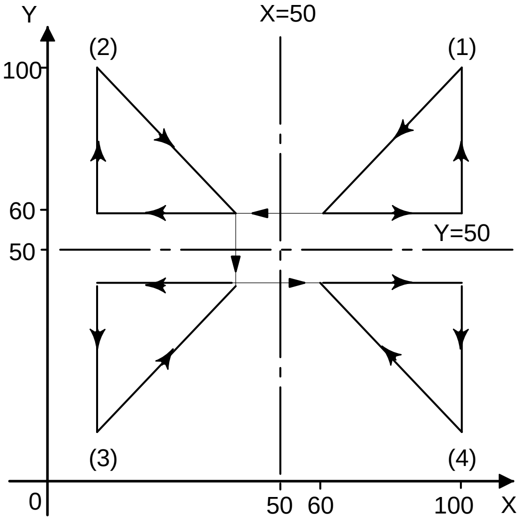
- **Setting of programmable mirror image**
  G51.1 X_ Y_ Z_ ;
  is commanded and mirror image is commanded to each axis (as if mirror was set on the axis).
- **Programmable mirror image cancel**
  G50.1 X_ Y_ Z_ ;
  is commanded and the programmable mirror image is canceled.

NOTE

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
```

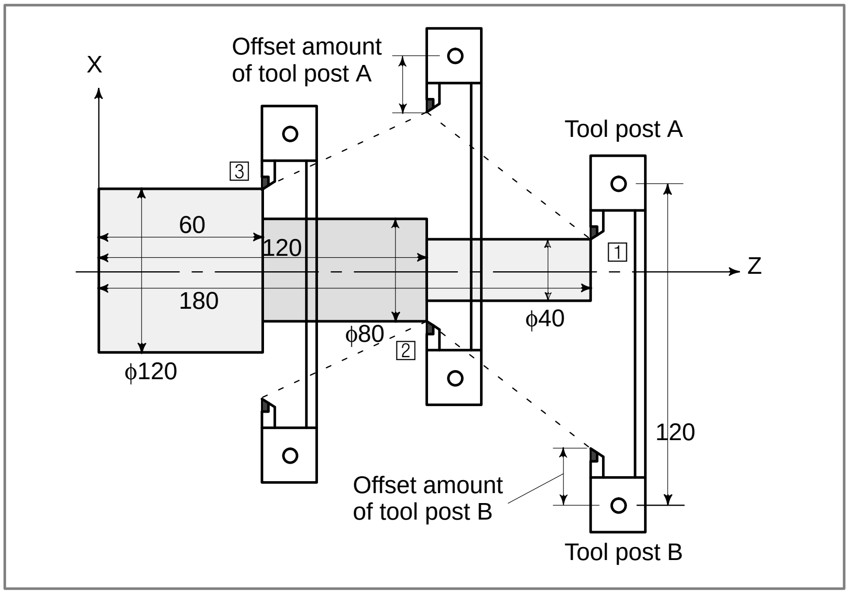
13.11 MIRROR IMAGE FOR DOUBLE TURRETS (G68, G69)

T series

Mirror image can be applied to X axis with G code.

G68 : Double turret mirror image on
G69 : Mirror image cancel

When G68 is designated, the coordinate system is shifted to the mating turret symmetrical cutting.
To use this function, set the distance between the two turrets at parameter.



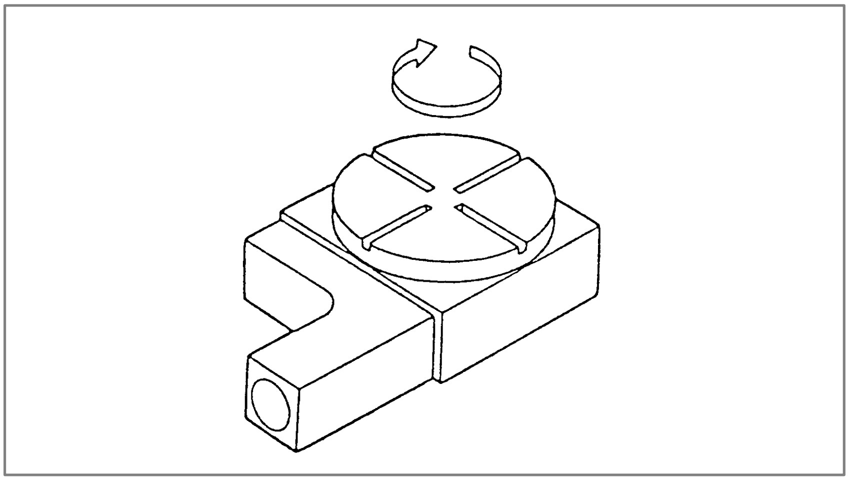
X40.0 Z180.0 T0101 ; Position turret A at $\boxed{1}$.
G68 ; Shift the coordinate system by the distance A to B (120 mm), and turn mirror image on.
X80.0 Z120.0 T0202 ; Position turret B at $\boxed{2}$.
G69 ; Shift the coordinate system by the distance B to A, and turn mirror image on.
X120.0 Z60.0 T101 ; Position turret A at $\boxed{3}$.

13.12 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.

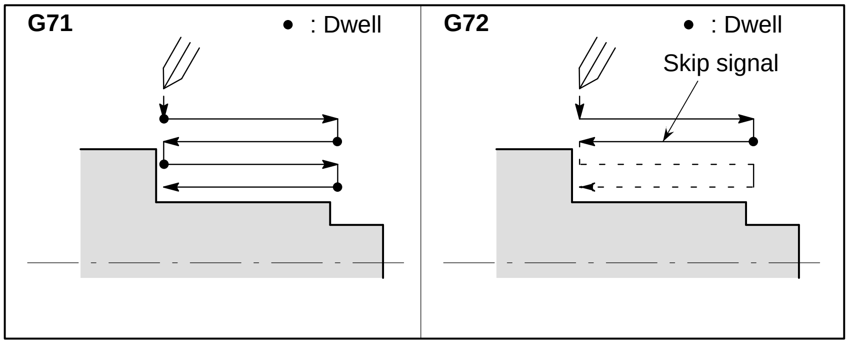


13.13 CANNED CYCLES FOR CYLINDRICAL GRINDING

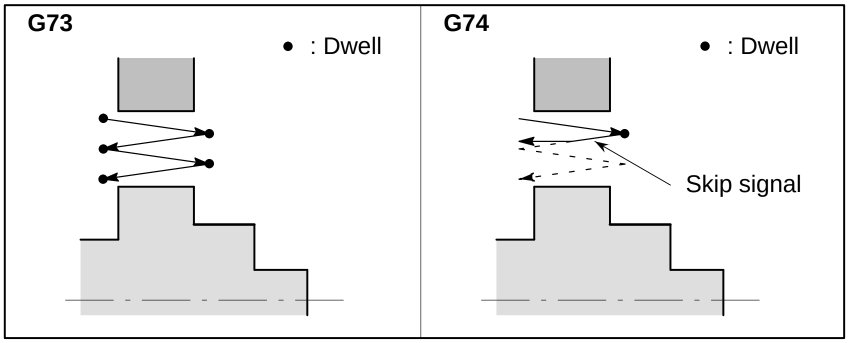
T series

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.

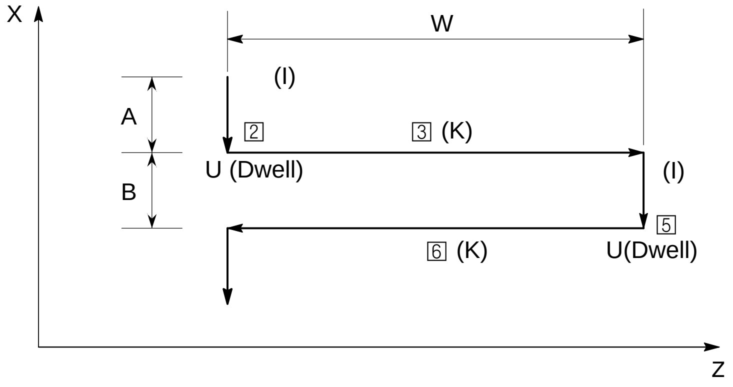
Traverse grinding cycle (G71, G72)



Oscillating grinding cycle (73, G74)



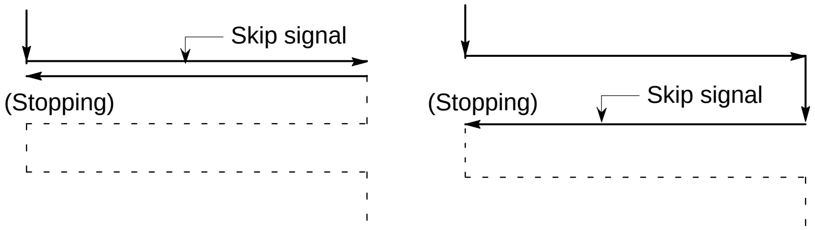
13.13.1 Traverse Grinding Cycle (G71)



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

13.13.2 Traverse Direct Gauge Grinding Cycle (G72)

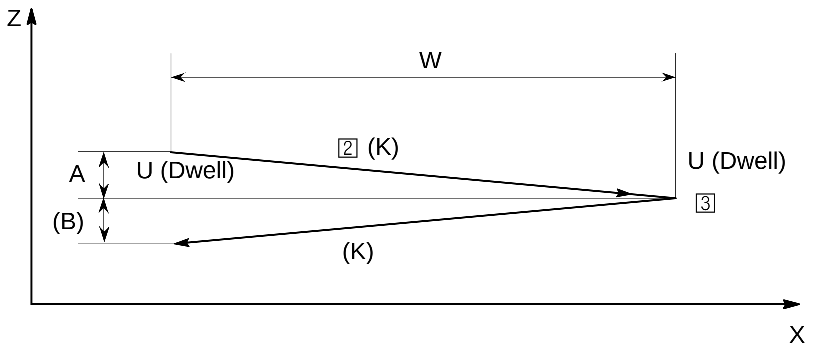


G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

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.

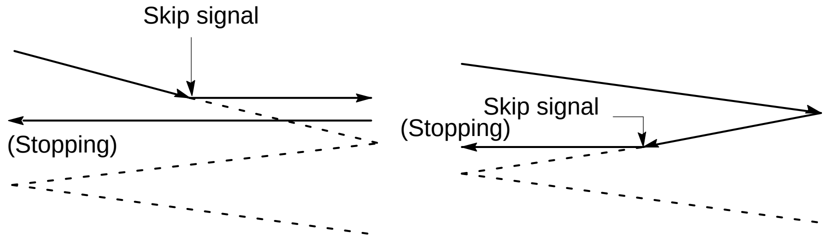
13.13.3 Oscillation Grinding Cycle (G73)



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

13.13.4 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.

13.14 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**

13.14.1
Plunge Grinding Cycle (G75)

The plunge grinding cycle is possible by the following command.

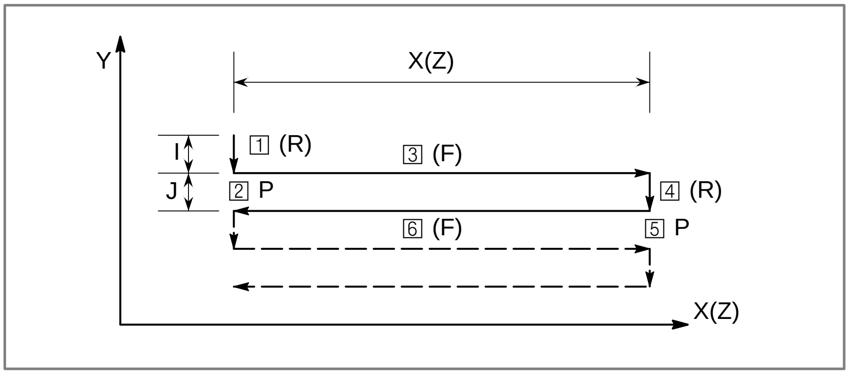
Format

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.

$\boxed{1}$ **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.

$\boxed{2}$ **Dwell :**
Performs dwell for only the time specified by P.

$\boxed{3}$ **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.

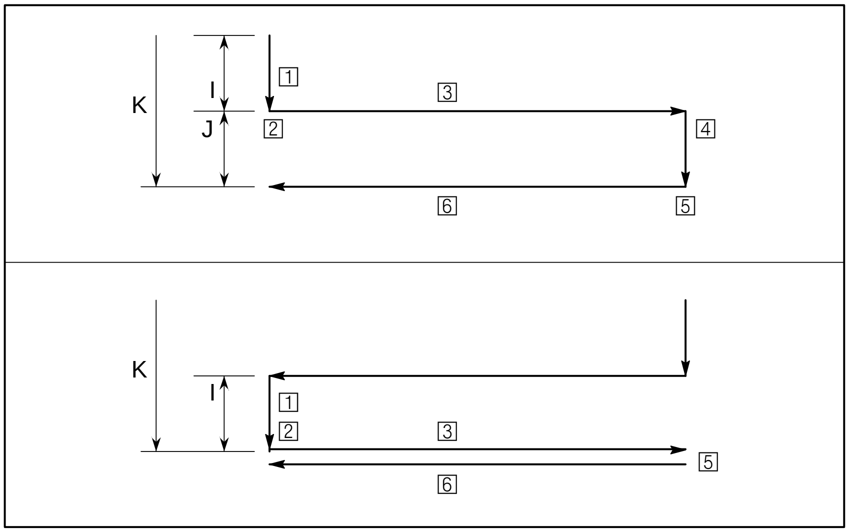
$\boxed{4}$ **Grindstone cutting :**
Cuts in Y axis direction by cutting feed only the amount specified by the second cutting depth J. The feed rate becomes the rate specified by R.

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

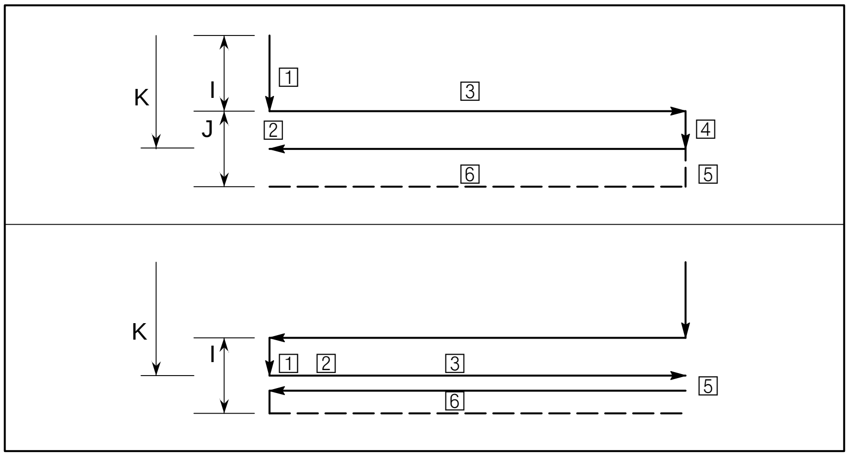
$\boxed{6}$ **Grinding (return direction) :**
Sent at rate specified by F in the reverse direction only the amount specified by X (or Z).

In case of single block, the operations from 1 to 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 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**



13.14.2

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.

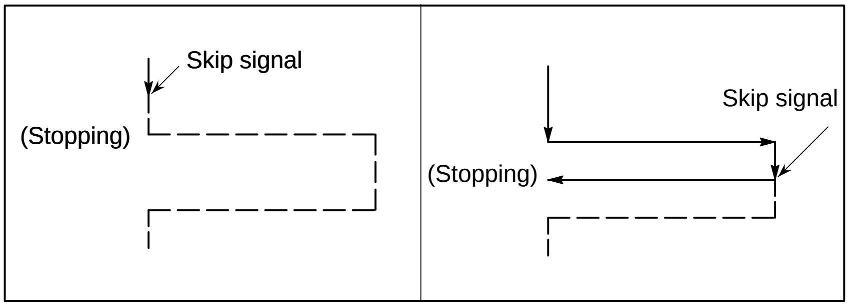
G77 differs from G75 as follows: Inputting a skip signal during a cycle can terminate the cycle after stopping (or terminating) the current operation sequence.

The following shows the operation at skip signal input for each operation sequence.

Explanations

- **Case of during operation sequence $\boxed{1}$ and $\boxed{4}$ (when I and J shift)**

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

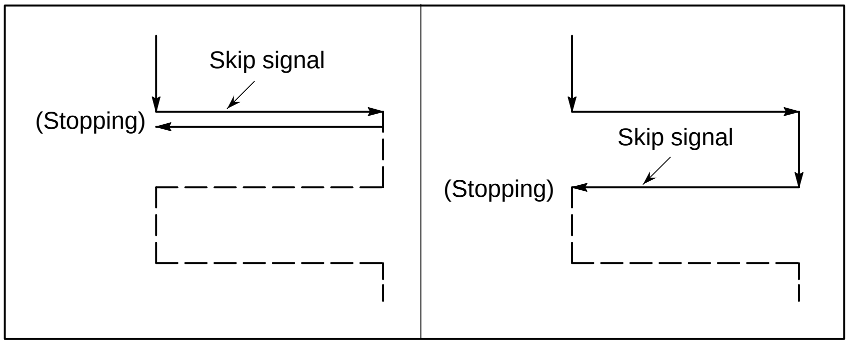


- **Case of during operation sequence $\boxed{2}$ and $\boxed{5}$ (during dwell)**

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

- **Case of during operation sequence $\boxed{3}$ and $\boxed{6}$ (when X(Z) shifts)**

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



13.14.3 Continuous Feed Plane Grinding Cycle (G78)

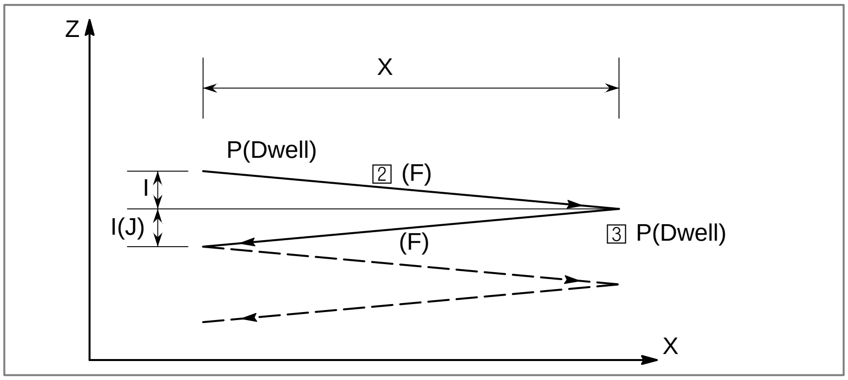
The continuous feed plane grinding cycle is possible by the following command.

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.

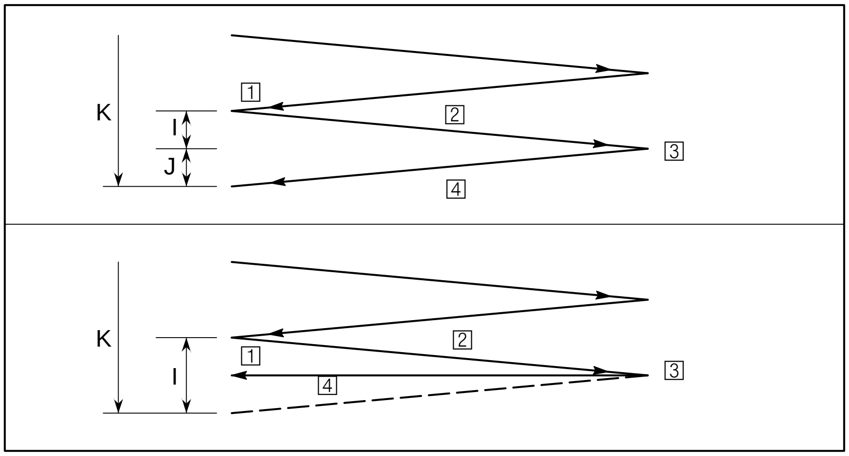
Dwell
$\boxed{2}$ Grinding
$\boxed{3}$ Dwell
$\boxed{4}$ Grinding (return direction)

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

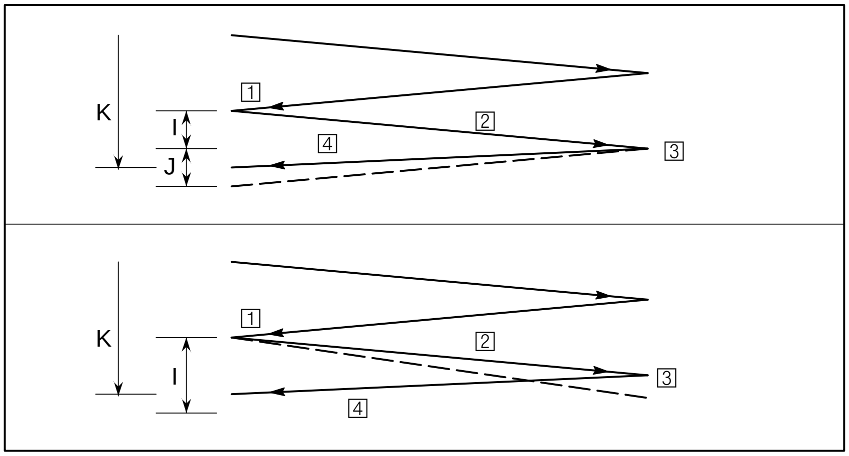
NOTE
When J is not commanded, it is regarded as J=1.

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.**



13.14.4
Intermittent Feed Plane Grinding Cycle (G79)

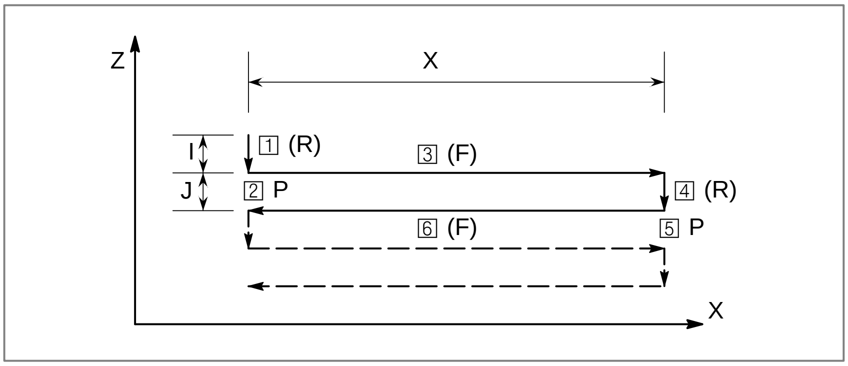
The intermittent feed plane grinding cycle is possible by the following command.

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.

$\boxed{1}$ **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.

$\boxed{2}$ **Dwell :**
Performs dwell for only the time specified by P.

$\boxed{3}$ **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.

$\boxed{4}$ **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.

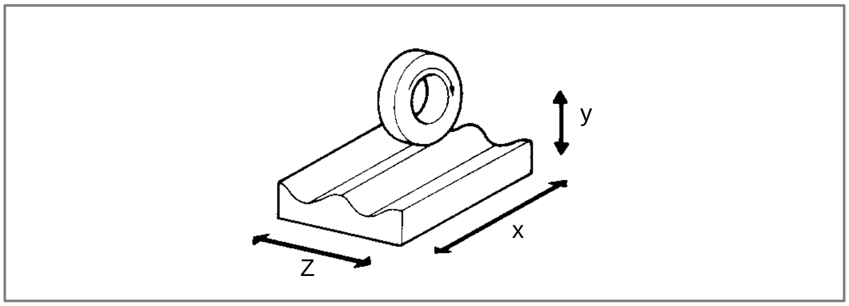
6 **Grinding (return direction) :**
Sent at rate specified by F in the reverse direction only the amount specified by X.

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

13.15 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).

13.16 FIGURE COPYING (G72.1, G72.2)

M series

The repeat cutting can be made by the rotation or translation of a figure commanded with a sub program.

The plane for figure copying is selected by the plane selection commands of G17, G18, and G19.

NOTE

The rotation copy cannot be commanded in the subprogram which commanded a rotation copy. Similarly, the translation copy cannot be further commanded in a subprogram which commanded a translation copy.

However, the translation copy and rotation copy can be commanded in the subprograms which commanded the rotation copy and translation copy, respectively.

13.16.1
Rotation Copy

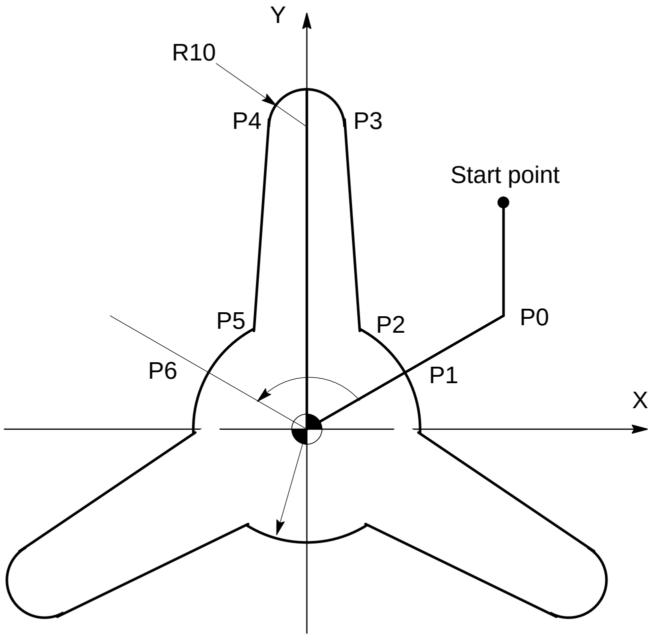
The repeat cutting can be made by the rotation of a figure commanded with a sub program using the following commands :
Select the plane on which rotational copy will be performed, using plane selection commands G17, G18, and G19.

Format

G17 G72.1 P_ L_ Xp_ Yp_ R_ ; Xp–Yp plane
G18 G72.1 P_ L_ Zp_ Xp_ R_ ; Zp–Xp plane
G19 G72.1 P_ L_ Yp_ Zp_ R_ ; Yp–Zp plane

P : Sub program number
L : Number of repetitions
Xp : Xp axis center coordinate of rotation
(Xp : X axis or the axis which is parallel to X axis)
Yp : Yp axis center coordinate of rotation
(Yp : Y axis or the axis which is parallel to Y axis)
Zp : Zp axis center coordinate of rotation
(Zp : Z axis or the axis which is parallel to Z axis)
R : Rotation angle (+ = Counterclockwise direction)

Examples



Main program

```
O1000 ;
N10 G92 X40.0 Y50.0 ;
N20 G00 G90 X_ Y_ ;                    (P0)
N30 G01 G17 G41 X_ Y_ D01 F10 ;        (P1)
N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
N50 G40 G01 X_ Y_ I_ J_ ;              (P0)
N60 G00 X40.0 Y50.0 ;
N70 M30 ;
```

Sub program

```
O2000 G03 X_ Y_ R30.0 ;  (P2)
N100 G01 X_ Y_ ;         (P3)
N200 G03 X_ Y_ R10.0 ;   (P4)
N300 G01 X_ Y_           (P5)
N400 G03 X_ Y_ R30.0 ;   (P6)
N500 M99 ;
```

13.16.2
Linear Copy

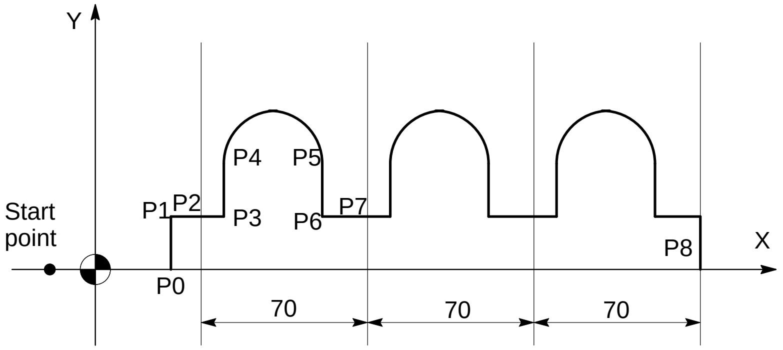
The repeat cutting can be made by the translation of a figure commanded with a sub program using the following commands :
Select the plane of linear copy with the plane selection commands G17, G18, and G19.

Format

G17 G72.2 P_ L_ I_ J_ ; Xp–Yp plane
G18 G72.2 P_ L_ K_ I_ ; Zp–Xp plane
G19 G72.2 P_ L_ J_ K_ ; Yp–Zp plane

P : Sub program number
L : Number of repetitions
I : Shift amount in Xp direction
J : Shift amount in Yp direction
K : Shift amount in Zp direction

Examples



Main program

```
O1000 ;
N10 G92 X–20.0 Y0 ;
N20 G00 G90 X0 Y0
N30 G01 G17 G41 X_ Y_ D01 F10 ;   (P0)
N40 Y_ ;                          (P1)
N50 X_ ;                          (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                       (P8)
N80 X0 ;
N90 G00 G40 X–20.0 Y0 ;
N100 M30 ;
```

Sub program

```
O2000 G90 G01 X_ ;   (P3)
N100 Y_ ;            (P4)
N200 G02 X_ I_ ;     (P5)
N300 G01 Y_          (P6)
N400 X_ ;            (P7)
N500 M99 ;
```

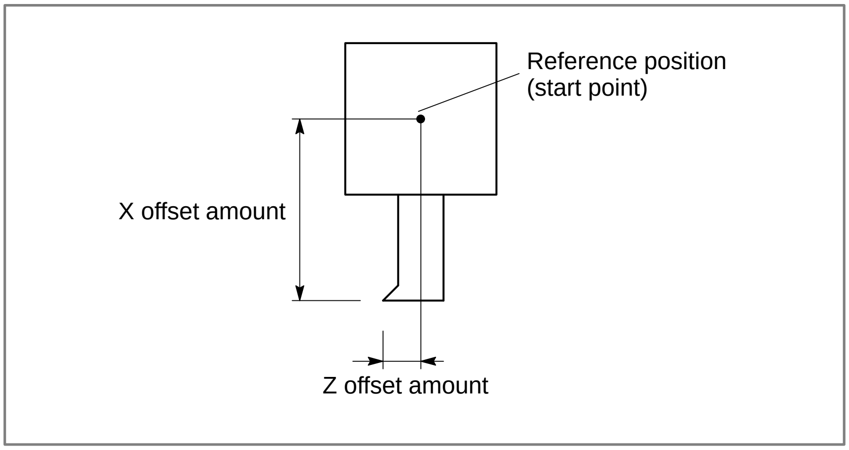
14 TOOL COMPENSATION FUNCTION

14.1 TOOL OFFSET

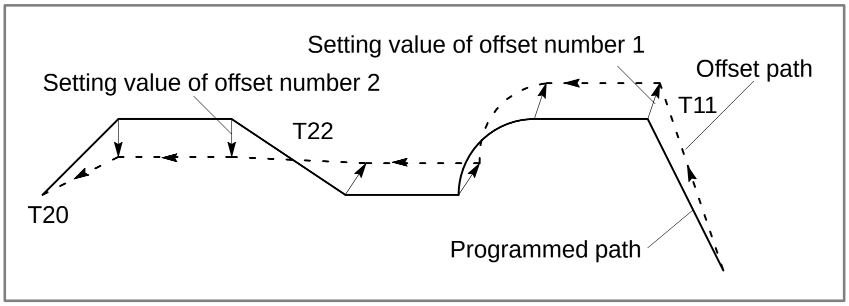
T series

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.

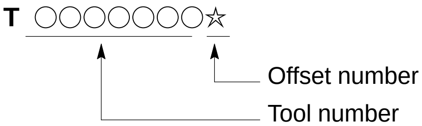


Explanations

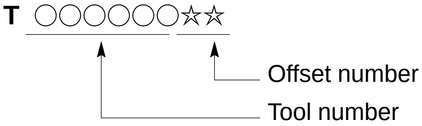
- **Offset number**

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



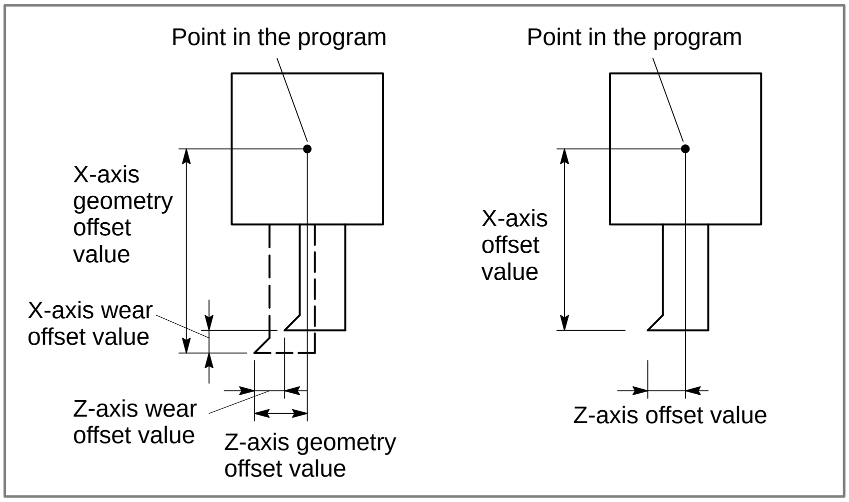
❑ When offset number is specified with two digits



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. 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

14.1.3 Y Axis Offset

In the system in which the Y axis is the fourth axis, the Y axis can be compensated by the tool offset value.When the tool geometry/wear compensation option is specified, tool geometry/ware compensation is valid for the Y-axis offset.

NOTE

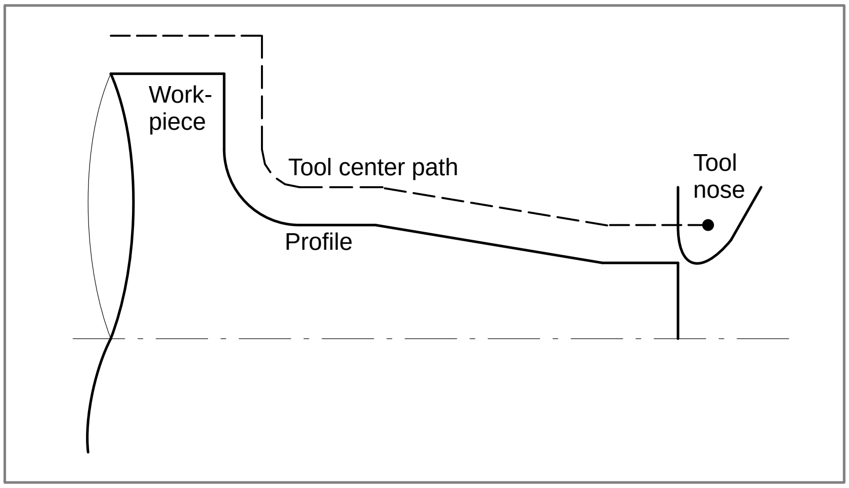
1 To use the Y-axis offset, the Y axis must be a linear axis.

2 The direct input function of tool offset value or direct input B function of tool compensation amount measured value cannot be used for the Y-axis offset.

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.

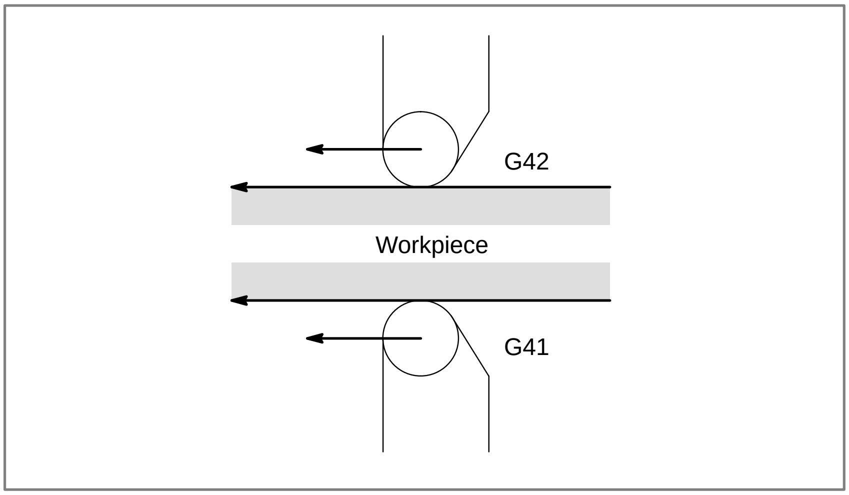


Cross points of line and line, arc and arc, line and arc is automatically calculated in the CNC to obtain offset actual tool path. So, Programming becomes simple, because it is only necessary to program the machining pattern.

Explanations

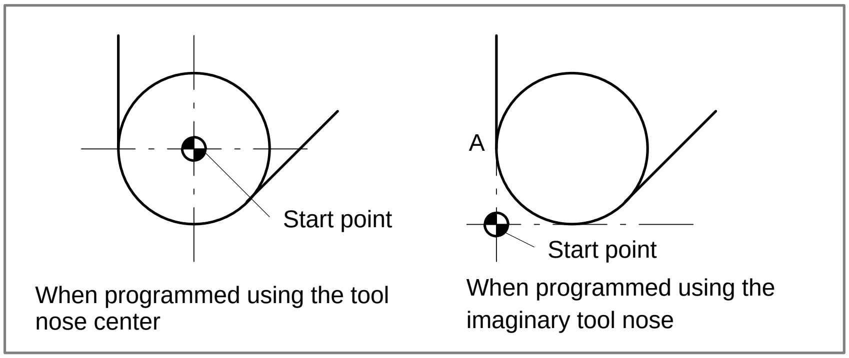
- **Tool nose radius compensation and its cancellation (G40, G41, G42)**

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.

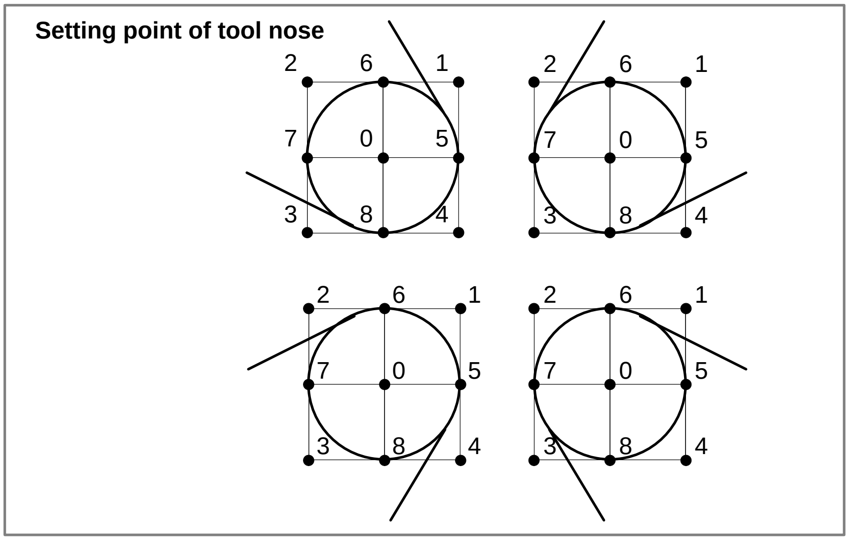


- **Imaginary tool nose**

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)**

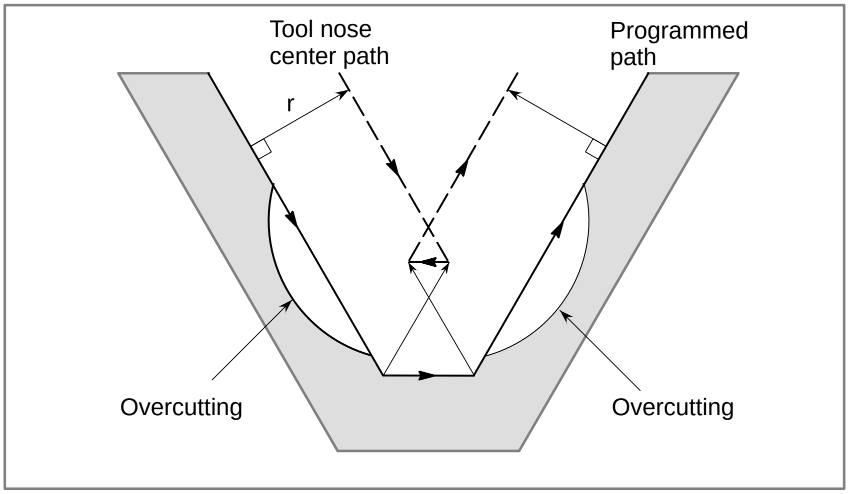
Cutter radius compensation is done on XY, ZX, YZ planes and on parallel axes of X, Y, Z axes.
Plane to perform tool nose radius compensation is selected with G17, G18, G19.

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 T series CORNER CIRCULAR INTERPOLATION FUNCTION (G39)

During radius compensation for the tool tip, corner circular interpolation, with the specified compensation value used as the radius, can be performed by specifying G39 in offset mode.

Format

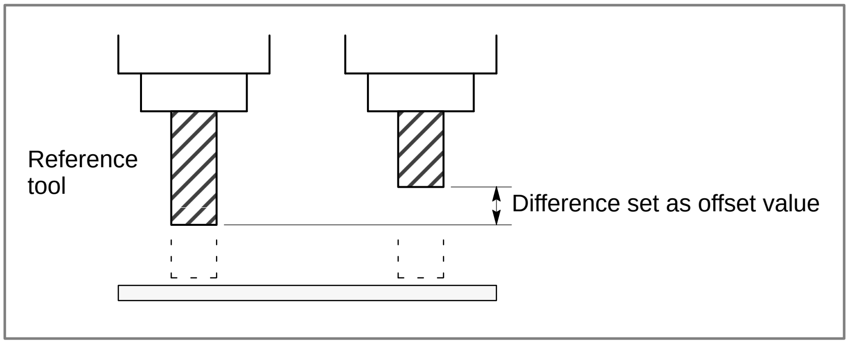
in offset mode, specify

G39 ;
or
G39 $\left\{\begin{array}{l}\text{I\_ J\_}\\ \text{I\_ K\_}\\ \text{J\_ K\_}\end{array}\right\}$

14.4 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

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

14.5 Tool Offset (G45, G46, G47, G48)

M series

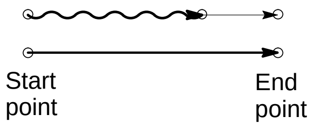
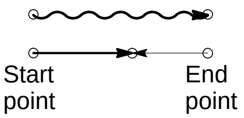
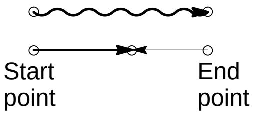
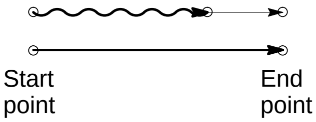
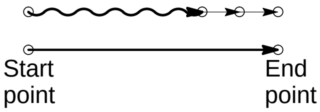
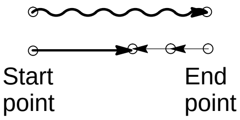
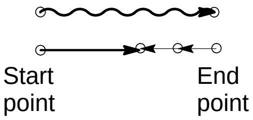
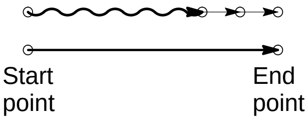
The programmed tool movement can be expanded or reduced for offset amount preset in the tool length compensation memory, by using this function.

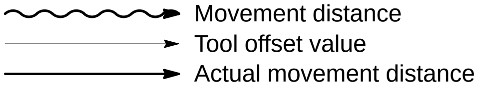
Explanations

- **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		
G46		
G47		
G48		



- **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.

14.6 CUTTER COMPENSATION

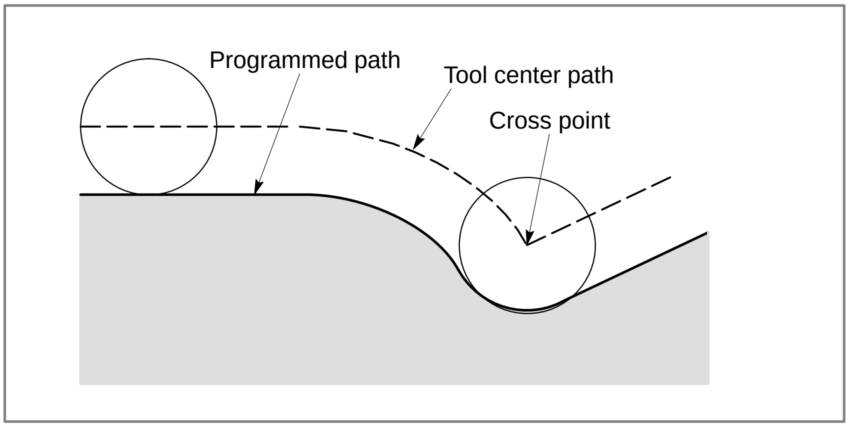
M series

14.6.1 Cutter Compensation B (G40 - 42)

With cutter compensation B, inside of the sharp angle cannot be cut. In this case, an arc larger that the cutter radius can be commanded to the corner by programming. Other functions are same as cutter radius compensation C.

14.6.2 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 NC 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.

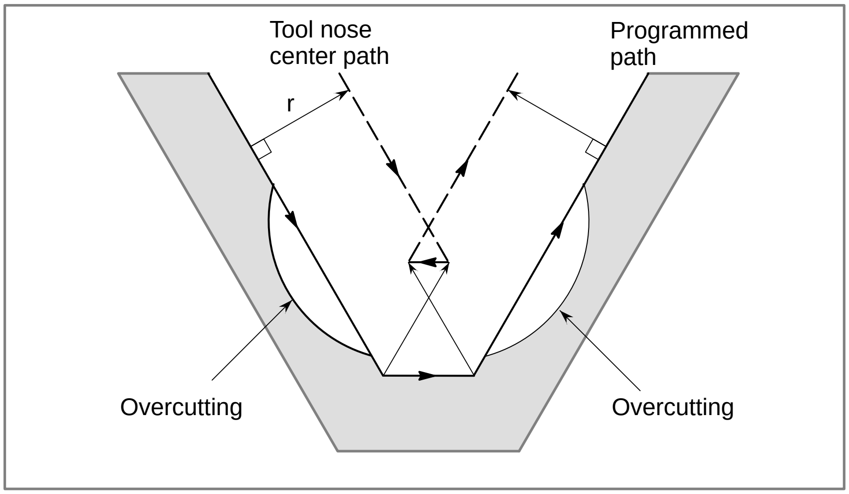
Example)
(U, V, W axes are parallel axes of X, Y, Z axes respectively)

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.



14.7 CORNER CIRCULAR INTERPOLATION FUNCTION (G39)

M series

During cutter compensation B, C, corner circular interpolation, with the specified compensation value used as the radius, can be performed by specifying G39 in offset mode.

Cutter compensation B

in offset mode, specify

$$\mathbf{G39}\left\{\begin{array}{l}\mathbf{X\_\ Y\_}\\ \mathbf{X\_\ Z\_}\\ \mathbf{Y\_\ Z\_}\end{array}\right\}\mathbf{;}$$

or

$$\mathbf{G39}\left\{\begin{array}{l}\mathbf{I\_\ J\_}\\ \mathbf{I\_\ K\_}\\ \mathbf{J\_\ K\_}\end{array}\right\}\mathbf{;}$$

Cutter compensation C

in offset mode, specify

G39 ;

or

$$\mathbf{G39}\left\{\begin{array}{l}\mathbf{I\_\ J\_}\\ \mathbf{I\_\ K\_}\\ \mathbf{J\_\ K\_}\end{array}\right\}\mathbf{;}$$

14.8 TOOL COMPENSATION MEMORY

14.8.1 Tool Compensation Memory

M series

One of the tool compensation memory A/B/C can be selected according to offset amount.
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 A**

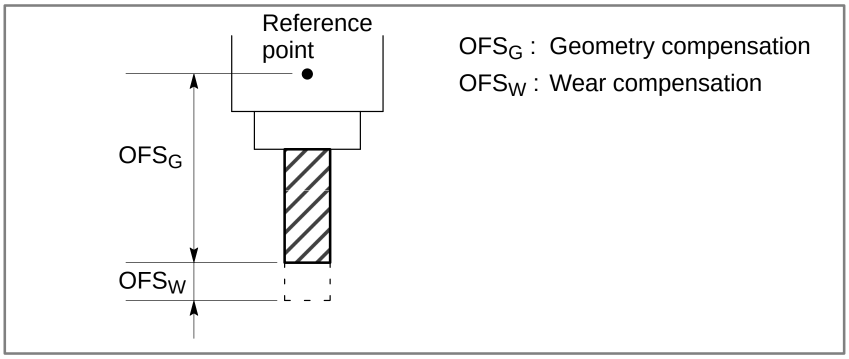
There is no difference between geometry compensation memory and tool wear compensation memory in this tool compensation memory A. Therefore, amount of geometry offset and tool wear offset together is set as the offset memory. There is also no differences between cutter radius compensation (D code) and tool length compensation (H code).

Example

Offset number	Compensation (geometry+wear)	D code/H code common
001	10.0	For D code
002	20.0	For D code
003	100.0	For H code

- **Tool compensation memory B**

Memory for geometry compensation and tool ware compensation is prepared separately in tool compensation memory B. Geometry compensation and tool wear compensation can thus be set separately. There is no difference between cutter radius compensation (D code) and tool length compensation (H code).



Example

Offset number	Geometry compensation	Wear compensation	D code/H code common
001	10.1	0.1	For D code
002	20.2	0.2	For D code
003	100.0	0.1	For H code

- **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

14.8.2 **T series**

Tool Offset Amount Memory

There are two types of tool offset amount memory, which can be selected according to offset amount.
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

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

2 The option enabling seven–digit tool offset specification cannot be used for B–axis offsets for B–axis control.

- **Tool geometry/wear compensation option not specified**

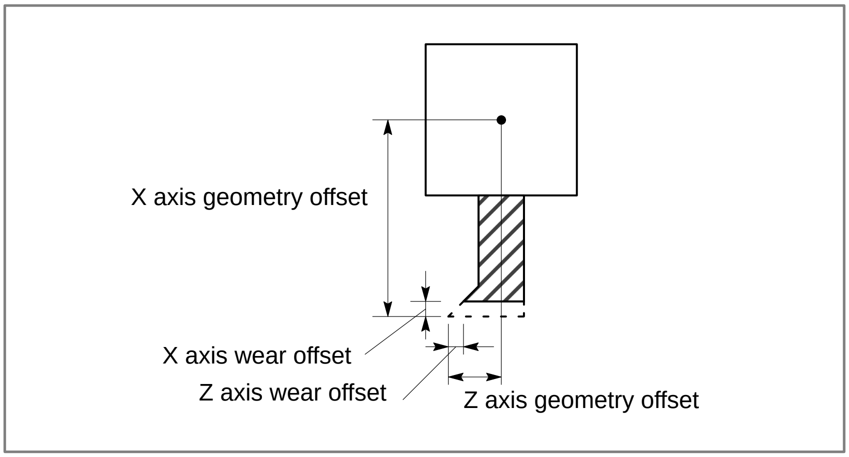
No distinction is made between the memory for geometry compensation values and that for wear compensation values. The total of the geometry compensation value and wear compensation value for a tool is stored in compensation memory.

Example

Offset number	X axis offset amount	Z axis offset amount	Y axis offset amount	Amount of tool nose compensation	Imaginary tool nose number
01	10.0	100.1	0	0.5	3
02	20.2	150.3	0	0.7	2
03	30.4	200.5	0	1.0	8

- **Tool geometry/wear compensation option specified**

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		Y axis offset value		Tool nose radius compensation value		Imaginary tool nose number
	Geometry offset	Wear offset	Geometry offset	Wear offset	Geometry offset	Wear offset	Geometry offset	Wear offset	
01	10.0	0.0	100.0	0.1	0	0	0.4	0.1	3
02	20.2	0.2	150.0	0.3	0	0	0.5	0.2	2
03	30.4	0.4	200.0	0.5	0	0	1.2	–0.2	8

14.9
NUMBER OF TOOL OFFSETS

14.9.1 M series
Number of Tool Offsets

- **32 tool offsets (standard)**
  Offset numbers (D code/H code) 0 - 32 can be used.
  D00 - D32, or H00 - H32
- **64 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 64 can be used.
  D00 - D64, or H00 - H64
- **99 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 99 can be used.
  D00 - D99, or H00 - H99
- **200 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 200 can be used.
  D00 - D200, or H00 - H200
- **400 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 400 can be used.
  D00 - D400 or H00 - H400
- **499 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 200 can be used.
  D00 - D499, or H00 - H499
- **999 tool offsets (optional)**
  Offset numbers (D code/H code) 0 - 400 can be used.
  D00 - D999 or H00 - H999

14.9.2 T series
Number of Tool Offsets

- **16 tool offsets (standard)**
  Offset numbers 0 - 16 can be used.
- **32 tool offsets (optional)**
  Offset numbers 0 - 32 can be used.
- **64 tool offsets (optional)**
  Offset numbers 0 - 64 can be used.
- **99 tool offsets (optional)**
  Offset numbers 0 - 99 can be used.

14.10
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 A**

G10 L11 P_ R_ ;

P_ : Offset number
R_ : Tool offset amount

- **Tool compensation memory B**

Setting/changing of geometry offset amount

G10 L10 P_ R_ ;

Setting/changing of tool wear offset amount

G10 L11 P_ R_ ;

- **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_ Y_ Z_ R_ Q_ ;

or

G10 P_ U_ V_ 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)
Y : Offset value on Y axis (absolute)
Z : Offset value on Z axis (absolute)
U : Offset value on X axis (incremental)
V : Offset value on Y 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, Y , 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, V, W, and C is added to the current offset value corresponding to the offset number.

NOTE

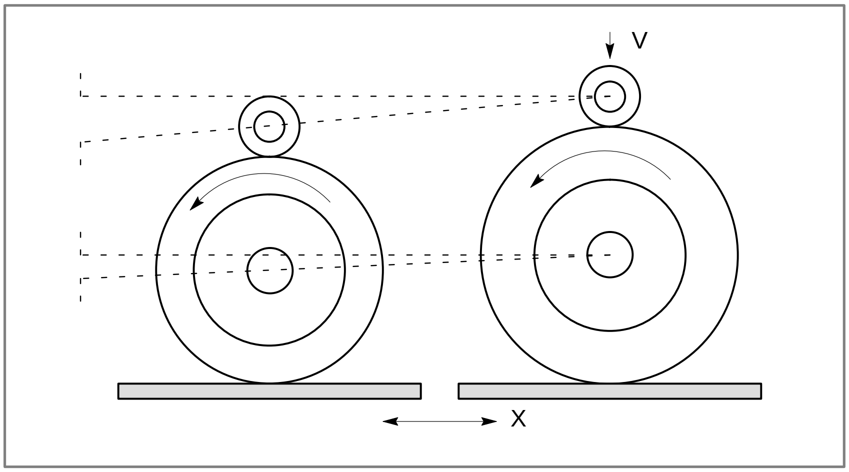
1 Addresses X, Y, Z, U, V, and W can be specified in the same block.

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.

14.11 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.



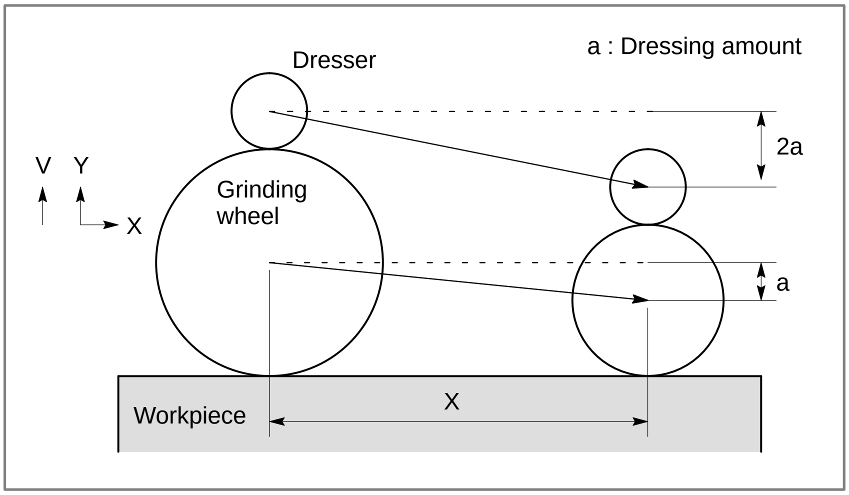
Explanations

- **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).



14.12 THREE–DIMENSIONAL TOOL COMPENSATION (G40, G41)

M series

In cutter compensation C, two–dimensional offsetting is performed for a selected plane. In three–dimensional tool compensation, the tool can be shifted three–dimensionally when a three–dimensional offset direction is programmed.

Format

- **Start–up (Starting three–dimensional tool compensation)**

When the following command is executed in the cutter compensation cancel mode, the three–dimensional tool compensation mode is set :

G41 Xp_ Yp_ Zp_ I_ J_ K_ D_ ;

Xp : X–axis or a parallel axis
Yp : X–axis or a parallel axis
Zp : Z–axis or a parallel axis

- **Canceling three–dimensional tool compensation**

When the following command is executed in the three–dimensional tool compensation mode, the cutter compensation cancel mode is set :

When canceling the three–dimensional tool compensation mode and tool movement at the same time

G40 Xp_ Yp_ Zp_ ;
or
Xp_ Yp_ Zp_ D00 ;

When only canceling the vector

G40 ;
or
D00 ;

- **Selecting offset space**

The three–dimensional space where three–dimensional tool compensation is to be executed is determined by the axis addresses specified in the start–up block containing the G41 command. If Xp, Yp, or Zp is omitted, the corresponding axis, X–, Y–, or Z– axis (the basic three axis), is assumed.

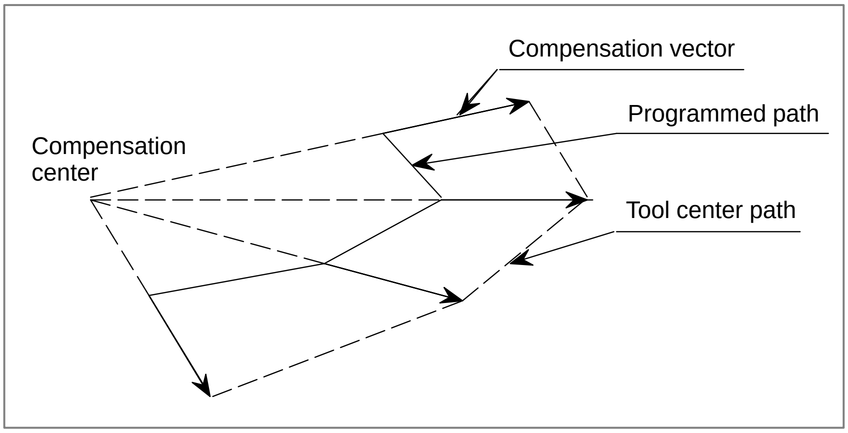
(Example)
When the U–axis is parallel to the X–axis, the V–axis is parallel to the Y–axis, and the W–axis is parallel to the Z–axis

G41 X_ I_ J_ K_ D ;	XYZ space
G41 U_ V_ Z_ I_ J_ K_ D_ ;	UVZ space
G41 W_ I_ J_ K_ D ;	XYW space

14.13 GRINDING WHEEL WEAR COMPENSATION (G40, G41)

T series

The grinding wheel compensation function creates a compensation vector by extending the line between the specified compensation center and the specified end point, on the specified compensation plane.



Format

Selecting the compensation center	**G41 Pn (n=1, 2, 3);** **G41 P1;** Select the first compensation center **G41 P2;** Select the second compensation center **G41 P3;** Select the third compensation center
Start–up	**D_;** D code other than 0
Canceling compensation mode	**D0;**
Holding the compensation vector	**G40;**

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). Total offset points are:

Total offset points = 128 × controlled axes.

Optional distribution to each axis can be done by parameter setting. As each position:

Offset pulse = (-7 to +7) x (magnification)

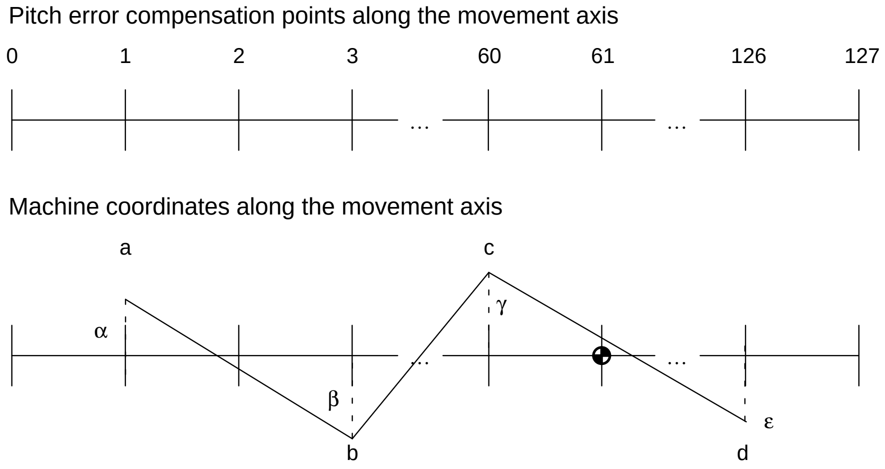
Where Offset pulse unit is the same as detection unit

Magnification: 0 to 100 times, set by parameter (per axis)

15.2 STRAIGHTNESS COMPENSATION

On a machine having a long stroke, machining precision may be reduced if the straightness of the axes is poor. The straightness compensation function compensates an axis in detection units while the tool is moving along another axis, thus improving machining precision.
Moving the tool along an axis (movement axis, specified with a parameter) compensates another axis (compensation axis, specified with a parameter), at each pitch error compensation point along the movement axis.

Example:



a, b, c, d : Compensation point numbers for the movement axis (same as the numbers of the pitch error compensation points)
α, β, γ, ε : Compensation value for each compensation point number

The compensation value applied to the compensation axis is (α – β)/(b – a) for the range between points a and b.

NOTE

1 Straightness compensation is enabled once reference position return has been performed along the movement and compensation axes.
2 When the optional straightness compensation function is used, the optional storage pitch error compensation function is required.
3 Straightness compensation data is added to the storage pitch error compensation data when output.

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.

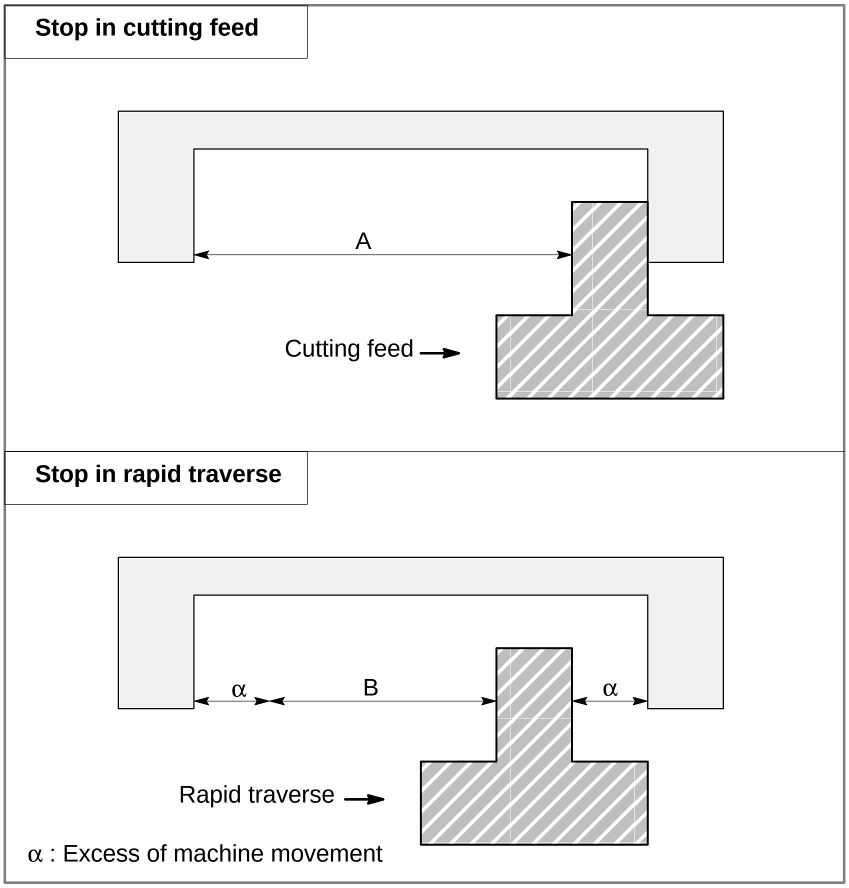
15.4 BACKLASH COMPENSATION SPECIFIC TO RAPID TRAVERSE AND CUTTING FEED

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	±α	±(–α)
Movement in opposite directions	±A	±B	±(B+α)	±(B+α)

*1) α=(A–B) / 2

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



Stop in cutting feed

Stop in rapid traverse

15.5 PROGRAMMABLE PARAMETER ENTRY (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)

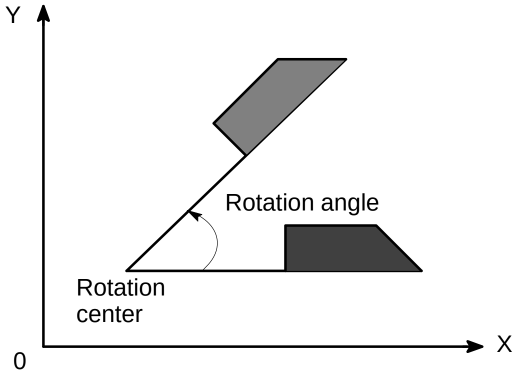
Note) Some parameters cannot be set.

16 COORDINATE SYSTEM CONVERSION

16.1

COORDINATE SYSTEM ROTATION (G68, G69) – (M SERIES) (G68.1, G69.1) – (T 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

$$\left\{\begin{matrix}\mathbf{G17}\\\mathbf{G18}\\\mathbf{G19}\end{matrix}\right\}\ \mathbf{G68}\ \alpha\_\ \beta\_\ \mathbf{R\_\ ;}$$

T series

$$\left\{\begin{matrix}\mathbf{G17}\\\mathbf{G18}\\\mathbf{G19}\end{matrix}\right\}\ \mathbf{G68.1}\ \alpha\_\ \beta\_\ \mathbf{R\_\ ;}$$

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 alpha, beta as the rotation center. Rotation angle is commanded in 0.001 x units in a range of:

–360000 ≦ R ≦ 360000

The rotation plane is the plane selected (G17, G18, G19) when G68 (G68.1) was commanded.
G17, G18 and G19 may not be commanded in the same block as G68.
When alpha, beta is omitted, the point where G68 (G68.1) was commanded becomes the rotation center.

G69; (M series), G69.1 ; (T series) Cancels the coordinate system rotation.

16.2 SCALING (G50, G51)

M series

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

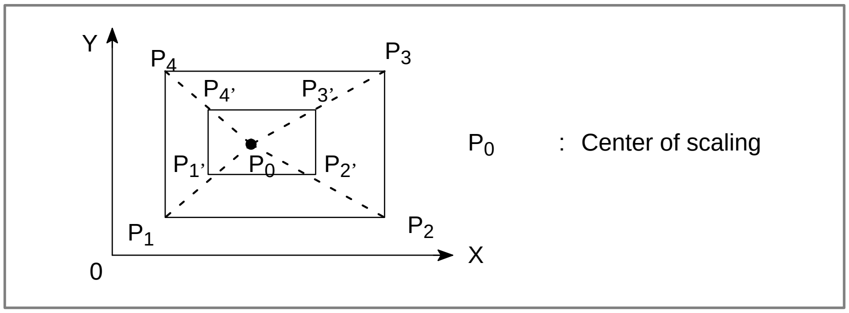
Format

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.

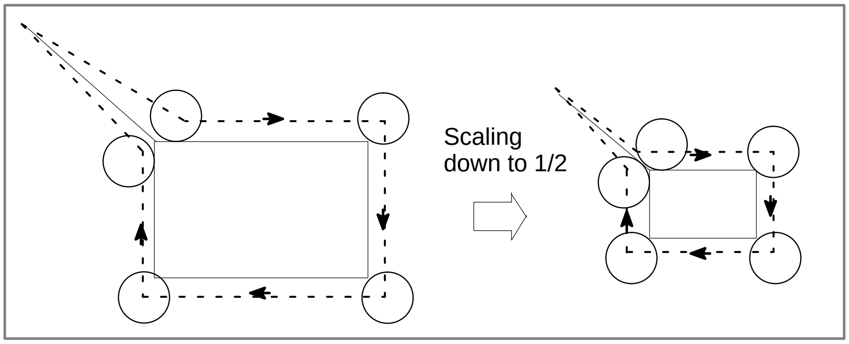
G50 : Scaling mode cancel
G51 : Scaling mode command

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.

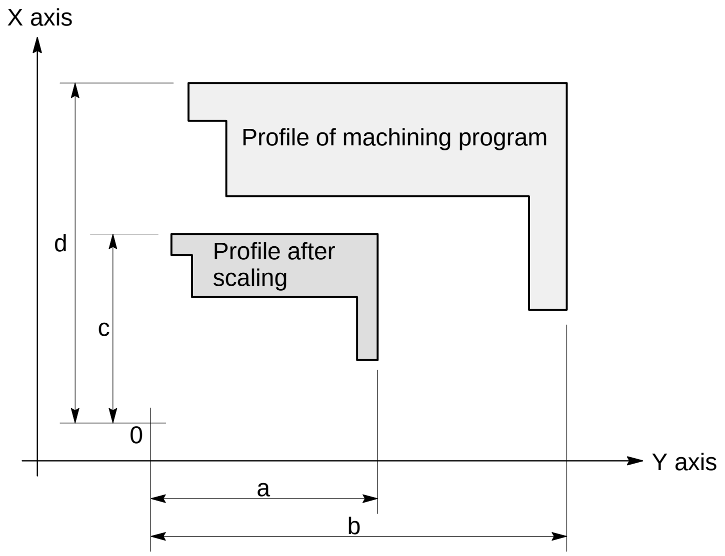


A scaling magnification can be set for each axis or for all axes in common. A parameter can specify whether it should be set for each axis or for all axes.

Format

<table>
<tr><th colspan="2">Scaling of each axis (Mirror image)</th></tr>
<tr><th>Format</th><th>Sign explanation</th></tr>
<tr><td>G51 X_ Y_ Z_ I_ J_ K_ ; Scaling start
⋮ } Scaling is effective. (Scaling mode)
G50 ; Scaling cancellation</td><td>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</td></tr>
</table>

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

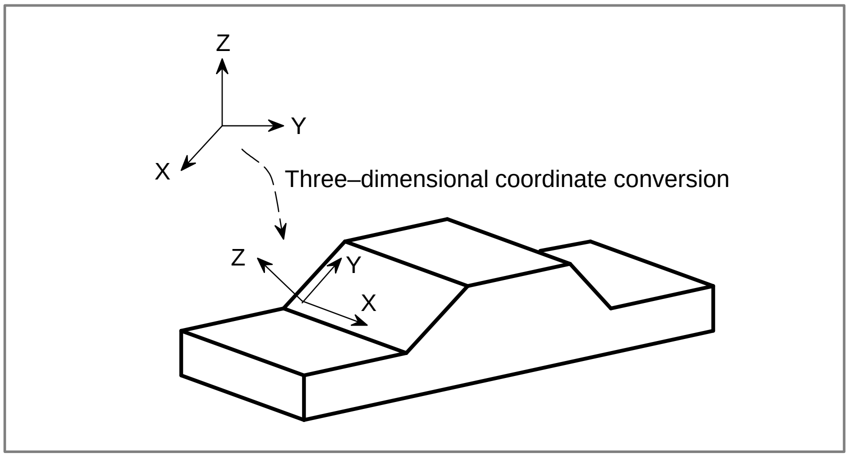


a/b : X-axis scaling magnification
c/d : Y-axis scaling magnification
o : Center of scaling

16.3 THREE–DIMENSIONAL COORDINATE CONVERSION (G68, G69)

M series

Coordinate conversion about an axis can be carried out if the center of rotation, direction of the axis of rotation, and angular displacement are specified. This function is very useful in three–dimensional machining by a die–sinking machine or similar machine. For example, if a program specifying machining on the XY plane is converted by the three–dimensional coordinate conversion function, the identical machining can be executed on a desired plane in three–dimensional space.



Format

G68 Xp$\underline{x_1}$ Yp$\underline{y_1}$ Zp$\underline{z_1}$ I$\underline{i_1}$ J$\underline{j_1}$ K$\underline{k_1}$ R$\underline{\alpha}$; Starting

Three–dimensional coordinate conversion mode

G69 ; Canceling three–dimensional coordinate

Xp, Yp, Zp : Center of rotation (absolute coordinates) on the X, Y, and Z axis or parallel axes
I, J, K : Direction of the axis of rotation
R : Angular displacement

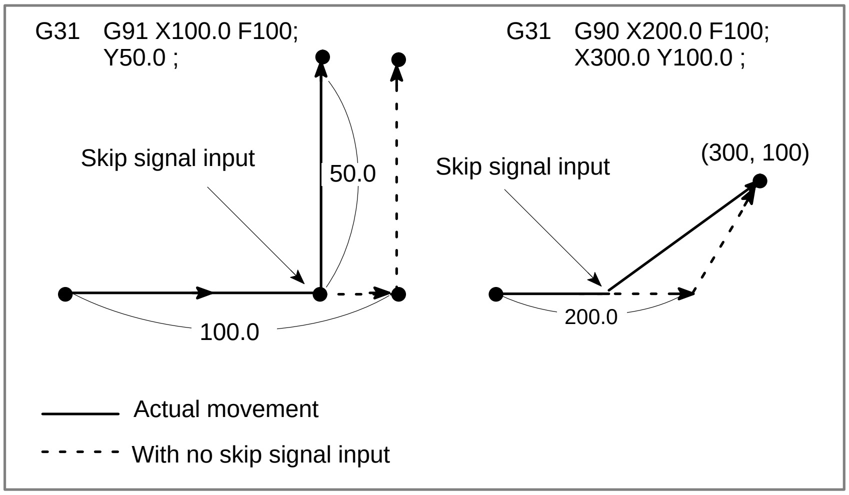
Three–dimensional coordinate conversion can be applied up to two times.

N1 G68 Xp$\underline{x_1}$ Yp$\underline{y_1}$ Zp$\underline{z_1}$ I$\underline{i_1}$ J$\underline{j_1}$ K$\underline{k_1}$ R$\underline{\alpha}$;
N2 G68 Xp$\underline{x_2}$ Yp$\underline{y_2}$ Zp$\underline{z_2}$ I$\underline{i_2}$ J$\underline{j_2}$ K$\underline{k_2}$ R$\underline{\beta}$;

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 T series
TORQUE LIMIT SKIP (G31 P99, G31 P98)

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 M series
CONTINUOUS HIGH–SPEED SKIP FUNCTION (G31, P90)

The continuous high–speed skip function enables reading of absolute coordinates by using the high–speed skip signal. Once a high–speed skip signal has been input in a G31P90 block, absolute coordinates are read into custom macro variables #5061 to #5068. The input of a skip signal does not stops axial movement, thus enabling reading of the coordinates of two or more points.

The rising and falling edges of the high–speed skip signal can be used as a trigger, depending on the parameter setting.

Format

G31 P90 α__ F__

α__: Skip axis address and amount of travel
Only one axis can be specified. G31 is a one–shot G code.

17.6 TOOL LENGTH AUTOMATIC MEASUREMENT (G37)

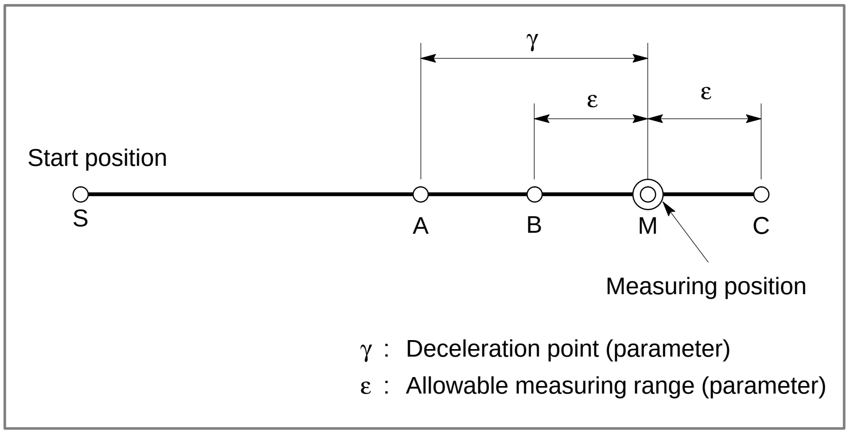
M series

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)

17.7

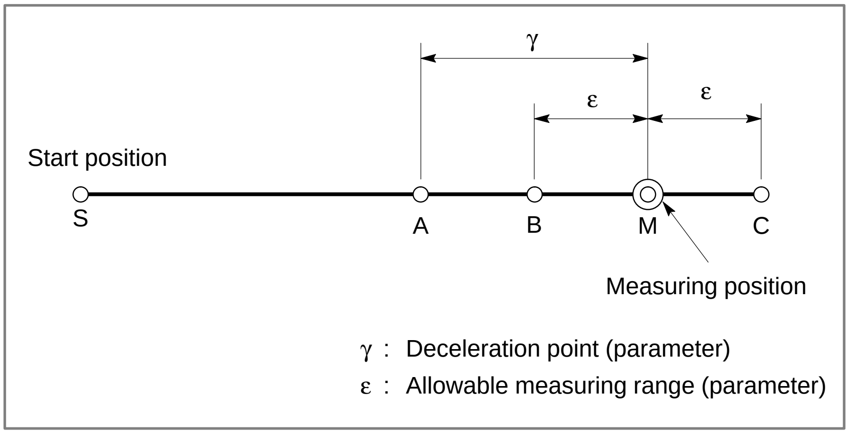
T series

AUTOMATIC TOOL OFFSET (G37, G36)

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_ ;



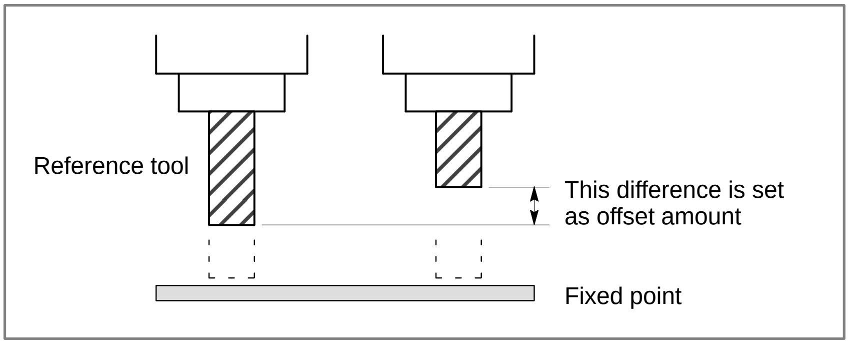
The tool is moved from the start position to the deceleration point A in rapid traverse, tool speed is decelerated to the measurement speed preset by parameter, and moved on till the measuring position reach signal is output. In case measuring position reach signal is not output in the allowable measuring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

17.8 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.



17.9 DIRECT INPUT OF TOOL COMPENSATION MEASURED VALUE/ DIRECT INPUT OF WORKPIECE COORDINATE SYSTEM SHIFT AMOUNT

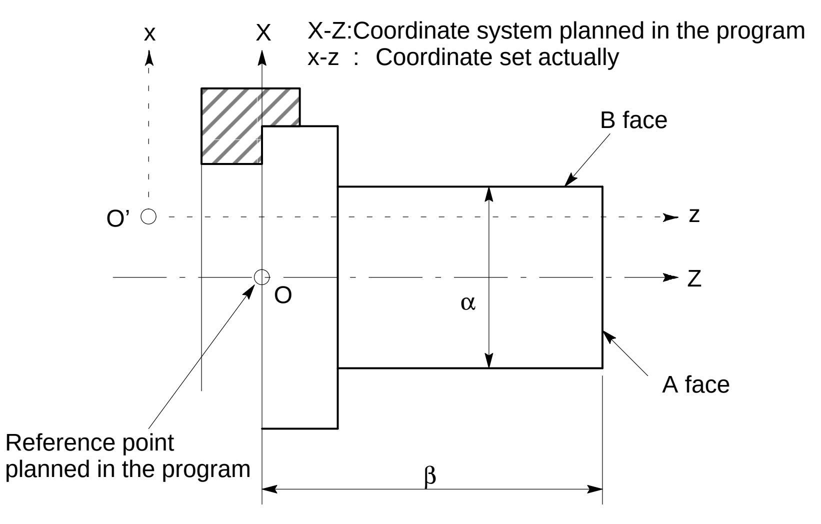
T series

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 Beta or Alpha. Direct input the measured value.

17.10 TOOL COMPENSATION VALUE MEASURED VALUE DIRECT INPUT 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.

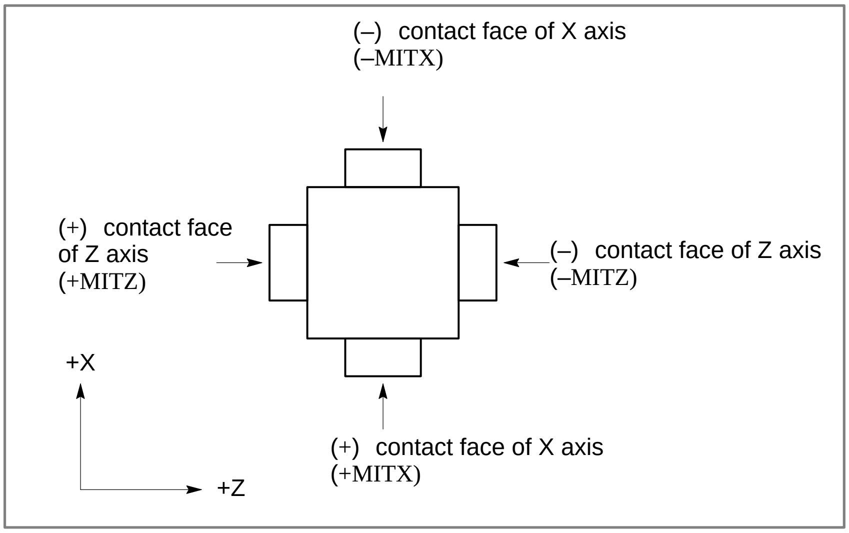
Explanations

Touch sensor

The touch sensor has contact faces in two directions along each axis, and outputs the following four signals by contact detection. These signals are input to CNC as tool compensation value writing signals.

The contact faces are to be selected according to the shape of the nose of the tool to be measured.

- **+MIT1 (+MITX) :**
  Contact the (+) contact face of the X axis
  (Contact in the X+ direction)
- **–MIT1 (–MITX) :**
  Contact the (–) contact face of the X axis
  (Contact in the X- direction)
- **+MIT2 (+MITZ) :**
  Contact the (+) contact face of the Z axis
  (Contact in the Z+ direction)
- **–MIT2 (–MITZ) :**
  Contact the (–) contact face of the Z axis
  (Contact in the Z- direction)



• 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]

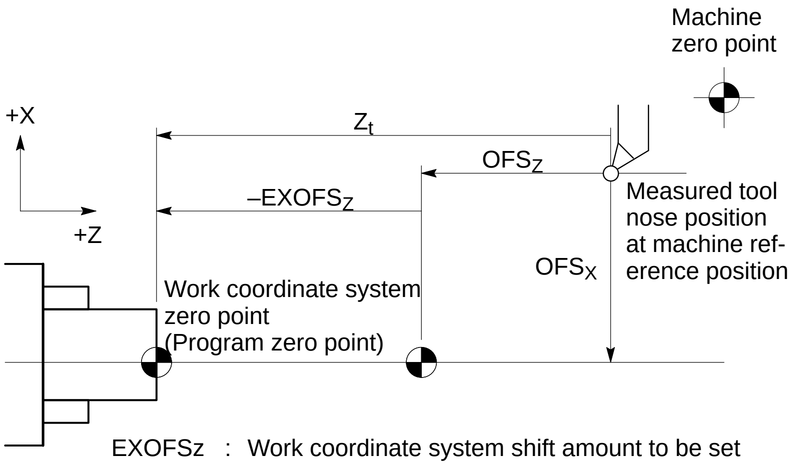
The tool offset amount to be set depends on how to determine the measurement reference position.

❑ Setting of work coordinate system shift amount

The work 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.

[Work 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 work coordinate system with the work edge (sensor contact point) being taken as the work 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).



EXOFSz : Work coordinate system shift amount to be set

OFSz : Tool geometry offset amount

Zt : Machine coordinate value (Distance to work edge)

17.11 T series
COUNT INPUT OF TOOL OFFSET VALUES

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.

17.12
DIRECT INPUT OF WORKPIECE ZERO POINT 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.

17.13 M series
TOOL LENGTH/ WORKPIECE ORIGIN MEASUREMENT B

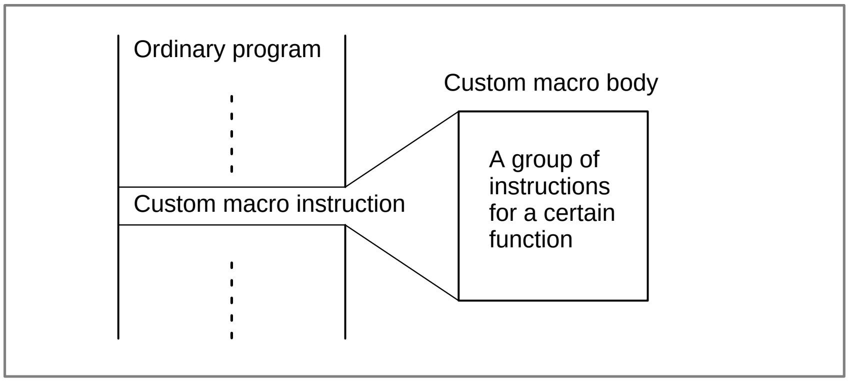
To enable measurement of the tool length, the following functions are supported: automatic measurement of the tool length by using a program command (G37) (automatic tool length measurement, described in Section 17.6) and measurement of the tool length by manually moving the tool until it touches a reference position, such as the workpiece top surface (tool length measurement, described in Section 17.8). In addition to these functions, tool length/workpiece origin measurement B is supported to simplify the tool length measurement procedure, thus facilitating and reducing the time required for machining setup. This function also facilitates the measurement of the workpiece origin offsets.

This function allows the operator to specify T/M code commands or reference position return, by means of a manual numeric command, while the tool length offset measurement screen is displayed.

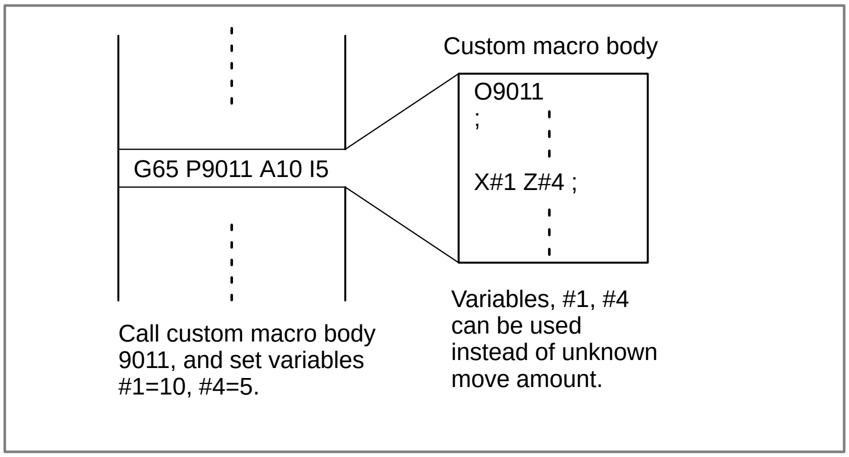
18 CUSTOM MACRO

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.

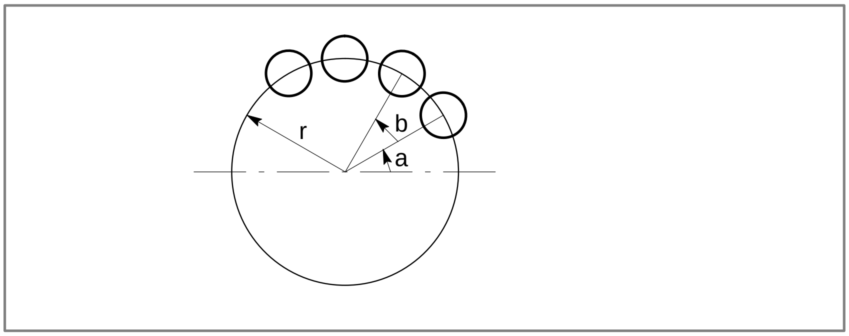


The programmer need not remember all the instructions in the custom macro body. He needs only to remember the representative, custom macro instruction.
The greatest feature in custom macro is that variables can be used in the custom macro body. Operation between the variables can be done, and actual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



Call custom macro body 9011, and set variables #1=10, #4=5.

Variables, #1, #4 can be used instead of unknown move amount.



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

❑ **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.

```
0 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_ G <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 – #149, #500 – #531

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 #149 are cleared when power is turned off, but common variables #500 to #531 are not cleared after power is turned off.

NOTE

The range of common variables can be enlarged to #100 to #199, and #500 to #999 by the option.

❑ 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, work zero point 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 6) above.

❑ **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

INCREASED CUSTOM MACRO COMMON VARIABLES

The range of common variables can be enlarged to #100 to #199, and #500 to #999 by the option.

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.

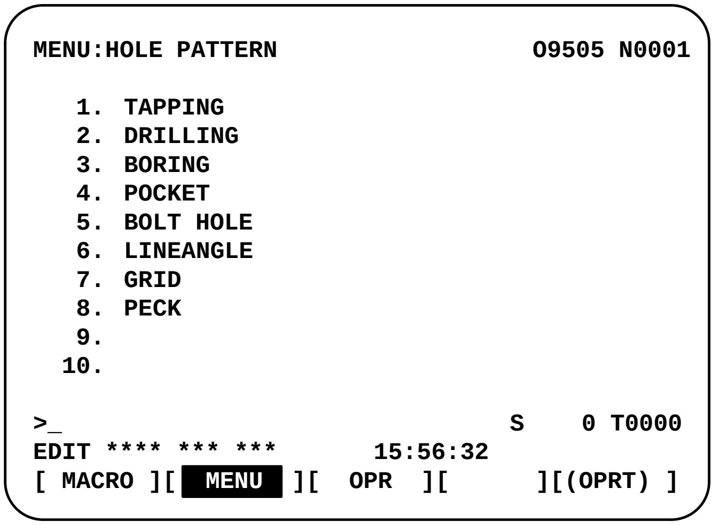
```
  :
M96P_ ;
  :  When custom macro interruption signal is input between M96 block
     and M97 block, custom macro specified by P is activated.
M97;
  :
```

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.

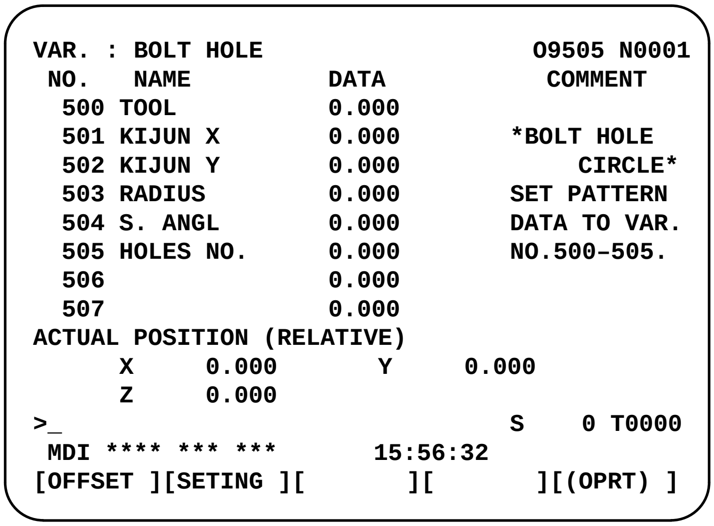
This function is basically executed by the custom macro created by a machine tool builder. What menus and machining data to prepare totally depends on a machine tool builder. Therefore, a machine tool builder can incorporate their own know-how into this function.



```
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 EXECUTER 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

1 When the macro executor is attached, the order-made macro cannot be specified.

2 To use the macro executor function for graphics display, the option for the graphics function is required.

18.6
C LANGUAGE EXECUTER FUNCTION

As with the conversational macro function of macro executors/compilers, the C language executor function is used to customize screens and include unique operations. Application programs for display and operation can be created in standard C language, in the same way as programs are made for normal personal computers. A program compiled on a personal computer is transferred and stored in flash ROM in the CNC via a memory card. The program is read into memory upon activation of the CNC, and executed by the C language executor.

Features

- **Low–cost customization**

No special additional hardware is needed to run the C language executor and application programs (*). All available display units are supported. User applications can be included in the current CNC system.

NOTE
(*) : The flash ROM/DRAM capacity may have to be increased.

- **Application development on a personal computer**

Application programs can be developed using an ordinary personal computer. Program development, from program creation and editing to compilation/linkage, can also be performed on a personal computer. And, to a certain extent, debugging is also possible on the personal computer.

- **High compatibility with C language application programs for personal computers**

Microsoft Corporation’s C compiler (MS–C) is employed as the C language compiler. It is the de–facto standard C compiler for personal computers. The function library provided by the C language executor has excellent compatibility with the ANSI standards and MS–C. Therefore, application programs for ordinary personal computers can be transported to the CNC, except when they are dependent on particular hardware.

- **Integration of CNC software and applications**

An application program created by the machine tool builder is executed as one task of the CNC software. The application program can display its own screens in place of existing CNC screens. In addition, the application program can read and write CNC system data via libraries provided by the C language executor. This enables operation of the application program to be integrated with CNC software.

- **Using the C language executor with the macro executor**

The C language executor can be used with the macro executor. Not only executable macros, but also conversational macros can be used together. The screen display portion of a macro program already created by the machine tool builder can be replaced with a program coded in C. This can prevent existing software resources from becoming useless.

19

SERIES 15 TAPE FORMAT/SERIES 10/11 TAPE FORMAT

19.1 SERIES 15 TAPE FORMAT

The programs for the following functions can be created in the Series 10/11 tape format, and be executed by the setting parameter, using the memory.

- Equal lead threading (G33) (T series): (G32 for G-code system A)
- Sub program calling (M98)
- Canned cycles (G77, G78, G79) (T series):
  (G90, G92, G94 for G-code system A)
- Multiple repetitive canned cycles (G71 to G76) (T series)
- Canned cycles for drilling (G80 to G85) (T series)
- Canned cycles (G73, G74, G76, G80 to G89) (M series)

NOTE

Addresses and range of values to be specified the Series 16/18 format restrictions are placed on the range of specifiable values of the basic address. Specifying a value outside the Series 16/18 format range causes a P/S alarm. The restrictions are placed also on some addresses.

19.2 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 21 format are also imposed on the value specification range for the basic addresses. When a specified value exceeds the range of the Series 21 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 HIGH-SPEED CYCLE MACHINING (ONLY AT 1–PATH CONTROL)

This function converts the profile to be machined into data for high-speed pulse distribution, using the macro compiler or macro executor. It then calls and executes the data with the CNC command (G05) as a machining cycle.
Up to six axes can be commanded. (Up to six axes can be controlled simultaneously.)
The following command calls and executes the high-speed cycle machining data specified by the macro compiler or micro executor.

Format

G05 P10○○○ L○○○ ;

P10001 to P10999 : Starting number of the machining cycle to be called

L1 to L999 : Number of machining cycle repeats (The default value is L1.)

Up to 999-cycle data can be created. Address P specifies the cycle for machining. Two or more cycles can be called and executed successively according to the connection information (in the header). Address L specifies the number of times that the machining cycle is repeated. The number of repeats (in the header) can be specified for each cycle.
The following example explains the cycle connection and number of repeats.

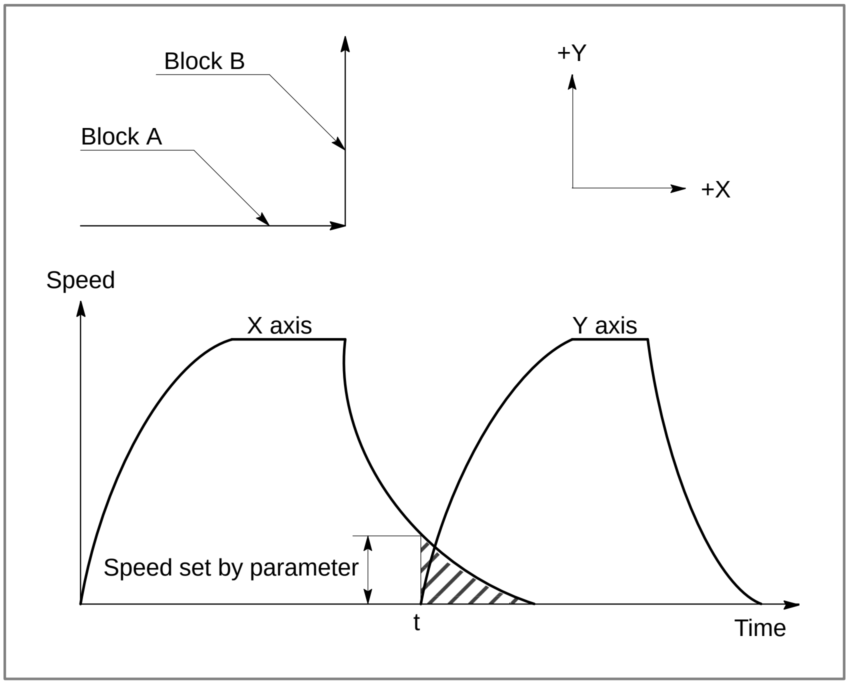
Example)
G05 P10001 L2 is specified for the following cycles:
Cycle 1, connection 2, number of repeats 1
Cycle 2, connection 3, number of repeats 3
Cycle 3, connection 0, number of repeats 1
The cycles of 1,2,2,2,3,1,2,2,2,3 are executed successively.

20.2 AUTOMATIC CORNER DECELERATION

M series

This function automatically decelerates the tool at a corner according to the corner angle. It can prevent a large sag caused by acceleration/ deceleration and servo delay on the junction of two blocks.

If the angle made by two consecutive blocks is less than the angle set by the parameter in the cutting mode (G64), the speed is automatically reduced at the end of the block. When the speed is reduced to the value set by the parameter or lower, movement of the next block starts.



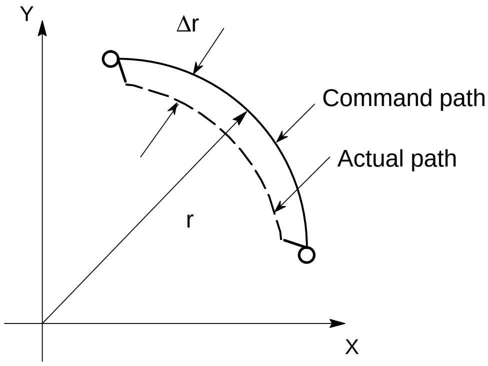
20.3 FEEDRATE CLAMP BY CIRCULAR RADIUS

M series

The machine is accelerated/decelerated automatically when the movement is started/stopped, so that the machine system should not be applied with any shock. When programming, therefore, no consideration needs to be made for acceleration/deceleration.

Especially when performing the high-speed arc cutting, however, the actual tool passage may bring about some error against the designated arc during circular interpolation due to this automatic acceleration/deceleration.

This error can approximately be given by the following formula;



$$\Delta r = \frac{1}{2}(T_1{}^2 + T_2{}^2)\frac{V^2}{r} \qquad \ldots\ldots\ (1)$$

Δr : Maximum value of radius error (mm)
v : Feedrate (mm/sec)
r : Circular radius (mm)
T_1 : Exponential Acceleration/deceleration time constant at cutting (sec)
T_2 : Time constant of servo motor (sec)

When performing the actual machining, the actual arc machining radius (r) and tolerance (delta r) are given, therefore, the maximum permissible speed v (mm/min.) can be given by the formula-(1).

”Feedrate clamp by circular radius” is such function that the circular cutting feed is automatically clamped when the feedrate designated may exceed the permissible tolerance to radial direction against the circular arc having optional radius designated by the program.

20.4 LOOK–AHEAD CONTROL (G08)

M 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 look–ahead control mode is entered.

Format

G08 P_ ;

P1 : Turn on look–ahead control mode.
P0 : Turn off look–ahead control mode.

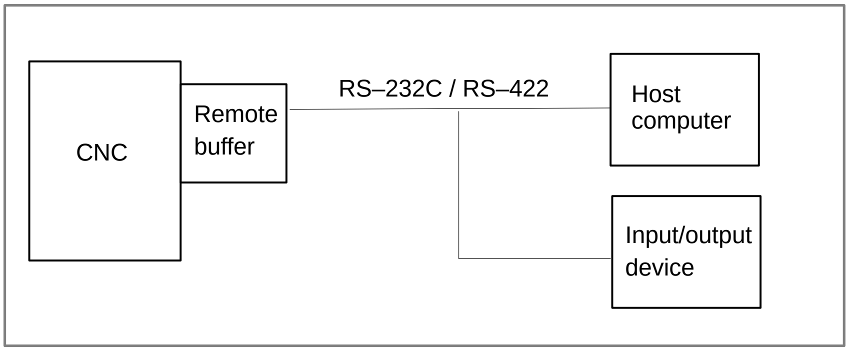
In look–ahead control mode, the following functions are available :

- Linear acceleration/deceleration before interpolation
- Automatic corner deceleration function

20.5 REMOTE BUFFER

20.5.1 Remote Buffer (Only at 1–path Control)

When the remote buffer is connected to the host computer or input/output device via serial interface, a great amount of data can be sent to CNC consecutively at a high speed.



The remote buffer enables the following operations:

- When connected to the host computer online, it performs DNC operation with high reliability and at a high speed.
- The CNC program and parameters can be down-loaded from the host computer.
- When connected to an input/output device, it enables DNC operation, and various data can be down-loaded. The following input/output devices can be connected.
  - ☐ FANUC PPR
  - ☐ FANUC FA Card
  - ☐ FANUC FLOPPY CASSETTE
  - ☐ FANUC PROGRAM FILE Mate
  - ☐ FANUC Handy File

  Hereafter, the device to which the remote buffer is connected is called the host computer.

Explanations

- **Interface between the remote buffer and host computer**

☐ Electrical interface

The following two types of interface are prepared as standard specifications.

- RS-232C Interface
- RS-422 Interface

	RS-233C	**RS-422**
Interface	Serial voltage interface (start-stop)	Balanced transmission serial interface (start-stop)
Baud rate	50 to 19,200 BPS	50 to 86,400 BPS (*1)
Cable length	4800 BPS or less 9600 BPS Varies according to I/O device.	Approx. 800 m (9600 BPS or less) 19,200 BPS or more

- **Software interface**

The following three protocols are prepared as the communication protocols between the remote buffer and host computer. The protocol can be selected by a parameter according to the specifications of the device to be connected.

Protocol	Features	Interface	Maximum transfer rate
Protocol A	Handshake method. Sending and receiving are repeated between two stations.	RS-232C	19200 BPS
		RS-422	86400 BPS
Extended protocol A	Similar to protocol A. Enables high-speed transfer of the NC program to meet high-speed DNC operation.	RS-422	86400 BPS
Protocol B	Controls communication with control codes output from the remote buffer.	RS-232C	19200 BPS
		RS-422	86400 BPS

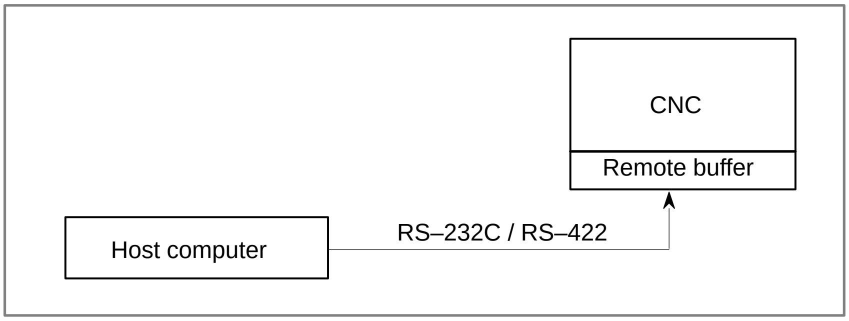
NOTE

The average data transfer rate is lower than the maximum transfer rate.

20.5.2

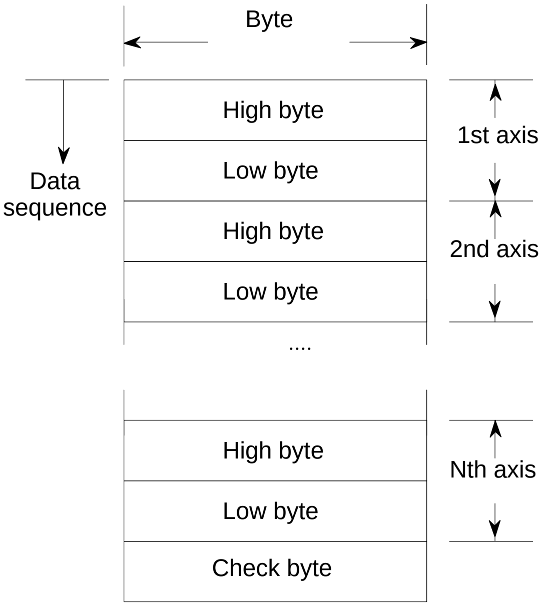
High–speed Remote Buffer A (G05) (Only at 1–path Control)

Specify G05 only in a block using normal CNC command format. Then specify move data in the special format explained below. When zero is specified as the travel distance along all axes, normal CNC command format can be used again for subsequent command specification.



Format

- Binary input operation enabled : G05;
  Binary input operation disabled : The travel distance along all axes are set to zero.
  Data format for binary input operation



In the data format for binary input operation, the travel distance along each axis (2 bytes) per unit time is specified. The travel distances along all axes are placed sequentially from the first axis, then a check byte is added. (The data length for one block is [2 x N + 1] bytes).
All data must be specified in binary.

20.5.3 **M series**

High–speed Remote Buffer B (G05) (At 1–path Control)

High–speed remote buffer A uses binary data. On the other hand, high–speed remote buffer B can directly use NC language coded with equipment such as an automatic programming unit to perform high–speed machining.

Format

G05 P01 ; Start high–speed machining

G05 P00 ; End high–speed machining

```
Example) O1234 ;
            :
         G05P01 ;  ⇐ Start high–speed machining
         X_ Y_ Z_ ;
            :
         G05P00 ;  ⇐ End high–speed machining
            :
         M02 ;
```

20.6 HIGH–PRECISION CONTOUR CONTROL (ONLY FOR ONE SYSTEM)

M series

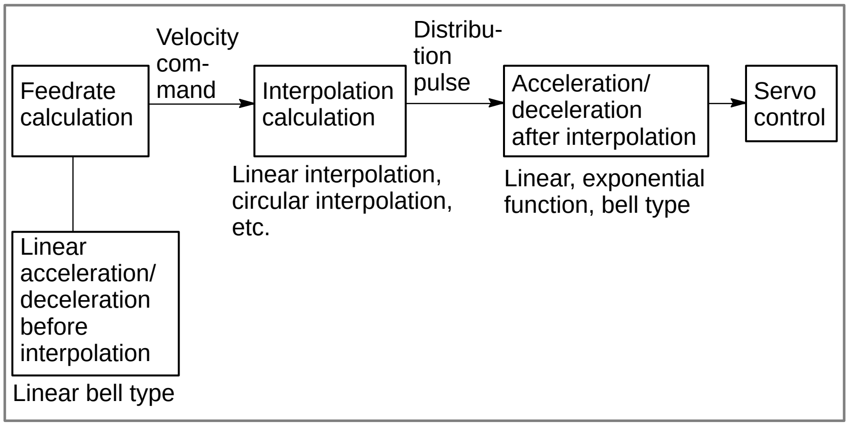
Machining errors by CNC include those caused by acceleration/deceleration after interpolation. To prevent such errors, the RISC processor provides the following functions:

- Acceleration/deceleration before interpolation by pre-reading multiple blocks. Because executed before interpolation, acceleration/deceleration does not cause a machining error.
- Automatic velocity control by smooth acceleration/deceleration. By pre-reading multiple blocks, changes in the profile and speed, and the allowable acceleration of the machine can be taken into consideration to execute smooth acceleration/deceleration.

Smooth acceleration/deceleration increases the feed-forward coefficient. As a result, the tracking error of the servo system can be reduced.

20.6.1 Acceleration/Decelera–tion Before Interpolation by Pre-reading Multiple Blocks

When cutting feed per minute is specified, tens of blocks are pre-read. The linear acceleration/deceleration is executed for the command speed before interpolation.
If acceleration/deceleration is executed after interpolation, the interpolation data is changed.
If it is executed for the feedrate before interpolation, the interpolation data is not affected.
Since the interpolation data can always be placed on the specified line or curve, there will be no machining profile error caused by acceleration/deceleration.



A change in feedrates for each axis on the junction of two blocks (corner section) may be greater than the value set in the parameter. In such a case, the appropriate feedrate (reduced speed) is calculated so that the change is within the set value at the corner. The feedrate in the former block automatically reduces to the calculated value.

20.6.2
Automatic Velocity Control Function

This function pre-reads 15 blocks, and automatically controls the feedrate.

The feedrate is determined on the basis of the following items. If the command speed exceeds the feedrate, acceleration/deceleration before interpolation is executed to reduce the speed.

- Change in speed for each axis at the corner, and allowable speed change specified
- Acceleration expected for each axis, and allowable acceleration specified
- Change in cutting load estimated by direction of Z-axis movement

In the automatic velocity control mode, acceleration/deceleration is executed before interpolation. The speed is then reduced automatically so that the machine is not shocked much.

Therefore, the time constant for automatic velocity control should be reduced, and the feed-forward coefficient should be increased. The machining error caused by delay of acceleration/deceleration or the servo system is then reduced.

20.7 SIMPLE HIGH–PRECISION 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 simple high–precision contour control mode
Q0 : End simple high–precision contour control mode
G05.1 A block for specifying G05.1 must not contain any other command.
Simple high–precision contour control mode can also be canceled by a reset.

20.8 HIGH–SPEED LINEAR INTERPOLATION (G05)

The high–speed linear interpolation function processes a move command related to a controlled axis not by ordinary linear interpolation but by high–speed linear interpolation. The function enables the high–speed execution of an NC program including a series of minute amounts of travel.

Format

G05 P2 ; Start high–speed linear interpolation

G05 P0 ; End high–speed linear interpolation

G05 A block for specifying G05 must not contain any other command.

21 AXES 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 CRT/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

The traveling command of master axis is given to two motors of master and slave axes in a simple synchronous control. However, no synchronous error compensation or synchronous error alarm is detected for constantly detecting the position deviation of the master and slave axes to compensate the deviation.

Simple synchronous operation for M series is allowed in the automatic operation and in the manual operation such as manual continuous feed, manual handle feed, incremental feed, or manual reference point return.

In simplified synchronous control in the T series, only automatic operation is allowed; manual operation cannot be performed.

In the manual reference point return, the master and slave axes similarly move until the deceleration operation is performed. After that, the detection of grid is performed independently.

The pitch error and backlash compensation are independently performed for the master and slave axes.

An input signal from PMC can be select whether the slave axis traveling is carried out based on the traveling command for that axis as in normal case or whether the slave axis traveling is carried out while synchronizing with the traveling of the master axis.

21.7 SYNCHRONIZATION CONTROL (ONLY AT 1–PATH CONTROL)

T series

The synchronization control function enables the synchronization of movements on two axes. If a move command is programmed for one of those two axes (master axis), the function automatically issues the same command to the other axis (slave axis), thus establishing synchronization between the two axes. The parking state can be selected to suppress movement of the slave axis, even if a move command is specified for the master axis. If the parking state is used with the synchronization control function, the operation can be controlled as follows:

(1) Synchronizes the movement on the slave axis with that of the master axis.

(2) Performs slave axis movement according to the move command programmed for the master axis. However, the movement specified by the command is not made for the master axis itself (master parking).

(3) Updates the slave axis coordinates according to the distance travelled along the master axis. However, no movement is made for the slave axis (slave parking).

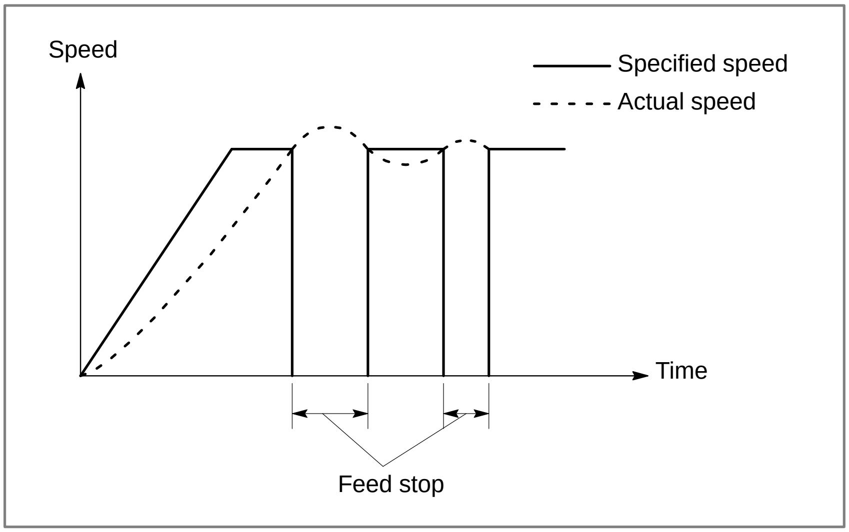
CAUTION

In the synchronization control described above, an identical move command is simultaneously output for two servo processing systems. Positional error between the two servo motors is not monitored nor is either servo motor adjusted to minimize the error. That is, synchronization error compensation is not carried out.

21.8 FEED STOP

This function usually checks position deviation amount during motion. If the amount exceeds the parameter set "feed stop position deviation amount", pulse distribution and acceleration/deceleration control is stopped for the while exceeding, and move command to the positioning control circuit is stopped.

The overshoot at rapid feed acceleration is thus kept to a minimum.



21.9 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.

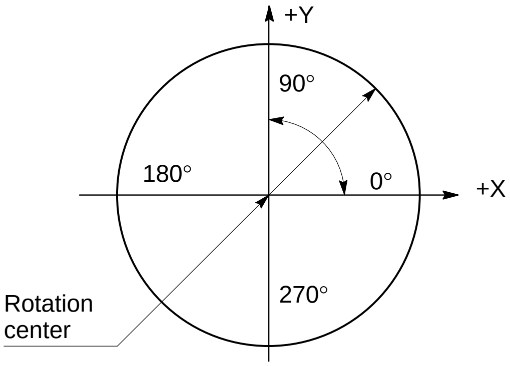
Format

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 (GOO), 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.

21.10 POLYGONAL 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

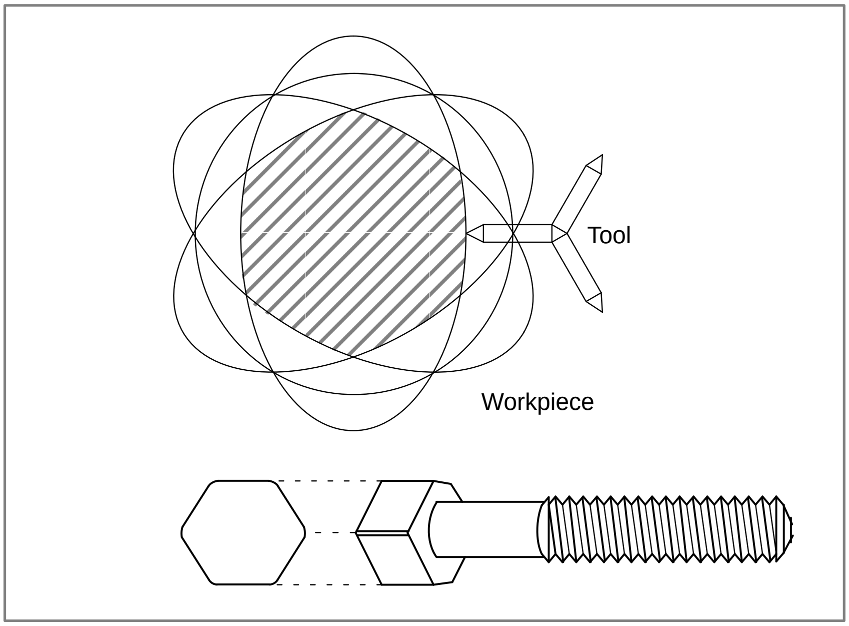
The polygon can be a quadrilateral or hexagon according to the above machining conditions.

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 polygonal 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.

21.11 POLYGONAL TURNING WITH TWO SPINDLES

T series

In the polygonal turning with two spindles, the first spindle is used as a workpiece rotation axis (master axis). The second spindle is used as a tool rotation axis (polygon synchronization axis). Spindle rotation control is applied to both spindles with a constant ratio.

The polygonal turning with two spindles can use different spindle speeds for the same workpiece, because it performs automatic phase compensation when a polygon synchronization mode command is issued or the S command is changed during polygon synchronization mode. With this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.

Moreover, polygon turning works with the first and second spindles on each tool post in a two–path lathe application. However, polygonal turning does not work with spindles on different tool posts.

21.12 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 :

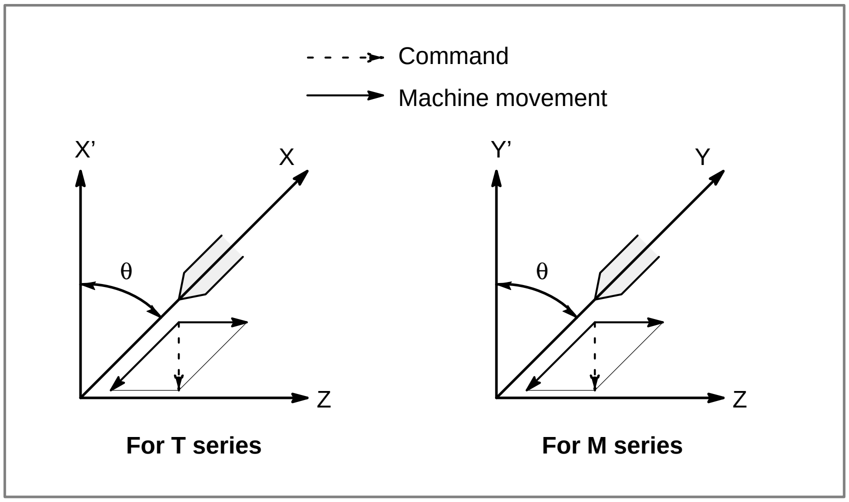
(1) Rapid traverse with moving distance specified
(2) Cutting feed–feed per minute, with moving distance specified
(3) Cutting feed–feed per revolution, with moving distance specified
(4) Skip–feed per minute, with moving distance specified
(5) Dwell
(6) Continuons feed
(7) Reference position return
(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.

21.13 SLANTED 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.

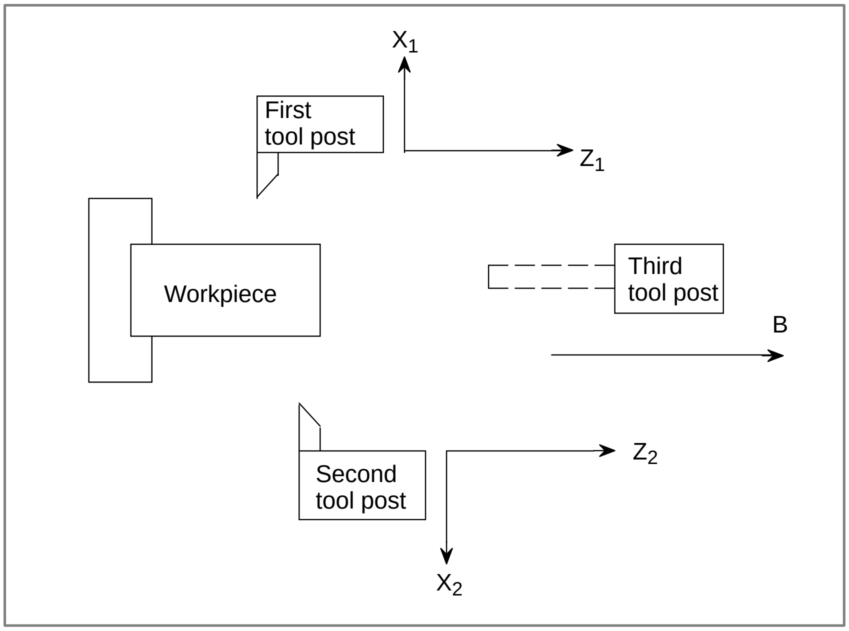


21.14 ARBITRARY AXIS ANGULAR AXIS CONTROL

For the ordinary angular axis control function of the T series, the X–axis is always used as the angular axis, while the Z–axis is always used as the perpendicular axis. (For the M series, the Y–axis is always used as the angular axis, while the Z–axis is always used as the perpendicular axis.) With arbitrary axis angular axis control, however, any axes can be specified as the angular and perpendicular axes, by specifying parameters accordingly.

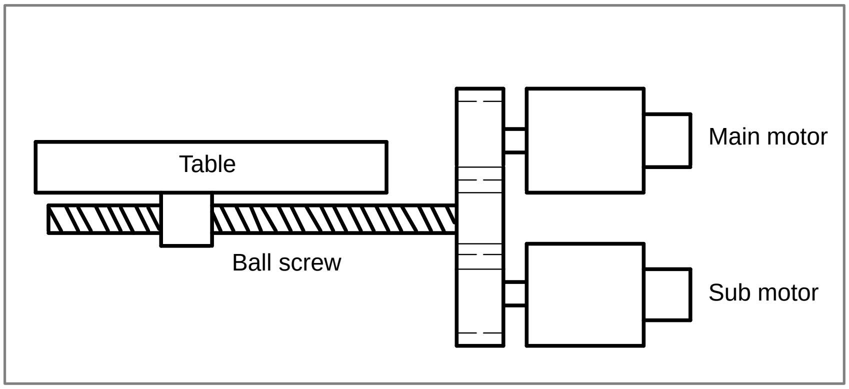
21.15 T series B–AXIS CONTROL

This function sets an axis (B–axis) independent of the basic controlled axes X_1, Z_1, X_2, and Z_2 and allows drilling, boring, or other machining along the B–axis, in parallel with the operations for the basic controlled axes. The X_2 and Z_2 axes can be used in two–path control mode.



21.16 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 NC 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.

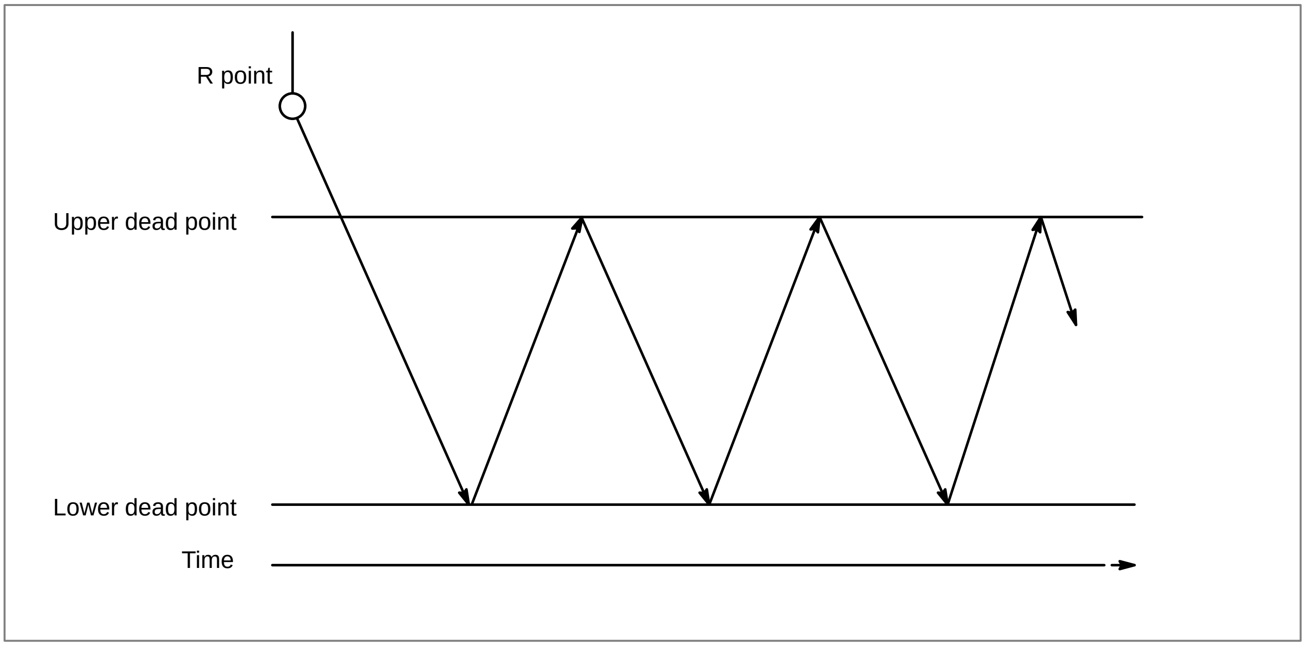
21.17 CHOPPING FUNCTION (G80, G81.1)

When contour grinding is performed, the chopping function can be used to grind the side face of a workpiece. By means of this function, while the grinding axis (the axis with the grinding wheel) is being moved vertically, a contour program can be executed to instigate movement along other axes.

In addition, a servo delay compensation function is supported for chopping operations. When the grinding axis is moved vertically at high speed, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The servo delay compensation function compensates for any displacement by increasing the feedrate. Thus, grinding can be performed almost up to the specified position.

There are two types of chopping functions: that specified by programming, and that activated by signal input.

Explanations



Format

G81.1 Z__ Q__ R__ F__ ;

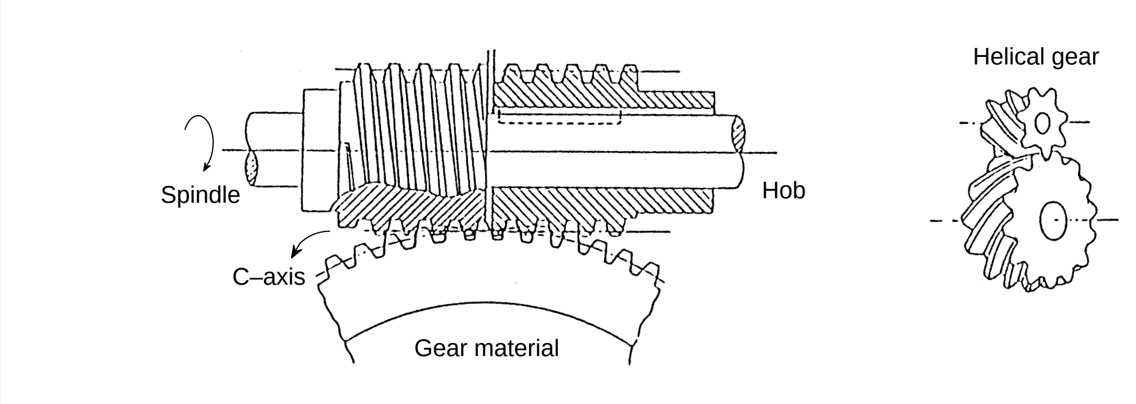
Z : Upper dead point
(For an axis other than the Z–axis, specify the axis address.)
Q : Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value, relative to the upper dead point.)
R : Distance from the upper dead point to point R
(Specify the distance as an incremental value, relative to the upper dead point.)
F : Feedrate during chopping

G80; Cancels chopping

21.18 HOBBING MACHINE FUNCTION (G80, G81)

M series

Gears can be cut by turning the workpiece (C–axis) in sync with the rotation of the spindle (hob axis) connected to a hob.
Also, a helical gear can be cut by turning the workpiece (C–axis) in sync with the motion of the Z–axis (axial feed axis).



Format

G81 T _ L _ Q _ P _ ;

T : Number of teeth (Specifiable range: 1 to 5000)

L : Number of hob threads (Specifiable range: 1 to 20 with a sign)

- The sign of L specifies the direction of rotation of the C–axis.
- If L is positive, the C–axis rotates in the positive direction (+).
- If L is negative, the C–axis rotates in the negative direction (–).

Q : Module or diametral pitch
For metric input, specify a module.
(Units: 0.00001 mm, Specifiable range: 0.01 to 25.0 mm)
For inch input, specify a diametral pitch.
(Units: 0.00001 inch^{-1}, Specifiable range: 0.01 to 250.0 inch^{-1})

P : Gear helix angle
(Units: 0.0001 deg, Specifiable range: –90.0 to +90.0 deg)

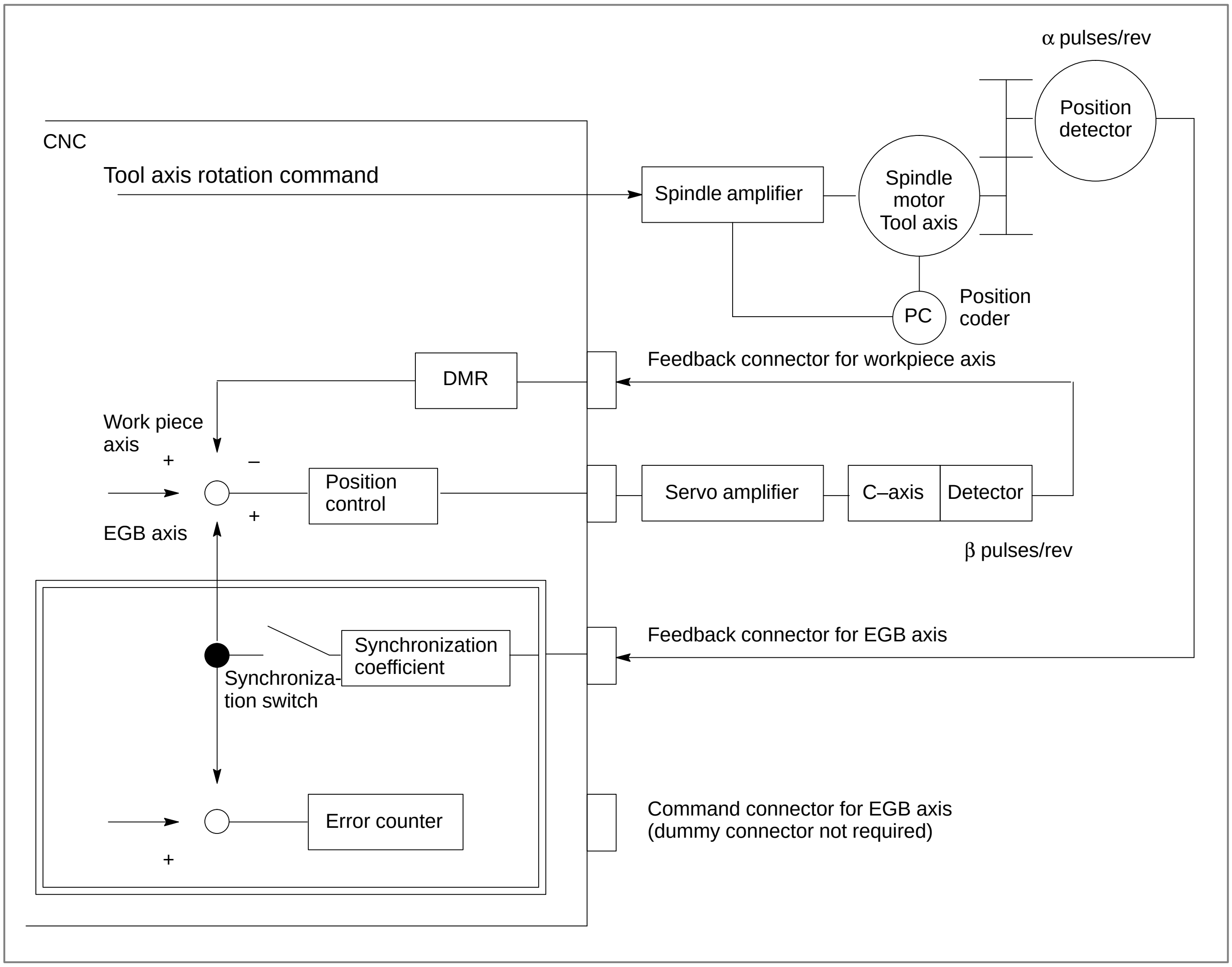
P and **Q** must be specified when a helical gear is to be cut.

G80 ; Cancels synchronization between the hob axis and C–axis.

21.19 SIMPLE ELECTRIC GEAR BOX (G80, G81)

M series

To machine (grind/cut) a gear, the rotation of the workpiece axis connected to a servo motor is synchronized with the rotation of the tool axis (grinding wheel/hob) connected to the spindle motor. To synchronize the tool axis with the workpiece axis, an electric gear box (EGB) function is used for direct control using a digital servo system. With the EGB function, the workpiece axis can trace tool axis speed variations without causing an error, thus machining gears with great precision.
A dedicated servo axis is also used for the tool axis, connected to the spindle motor and for which the rotational position must be read directly by the digital servo system (this axis is called the EGB axis).



Format

G81 T _ L _ Q _ P _ ; Starts synchronization.

T : Number of teeth

L : Number of hob threads

Q : Module or diametral pitch
Specify a module in the case of metric input.
Specify a diametral pitch in the case of inch input.

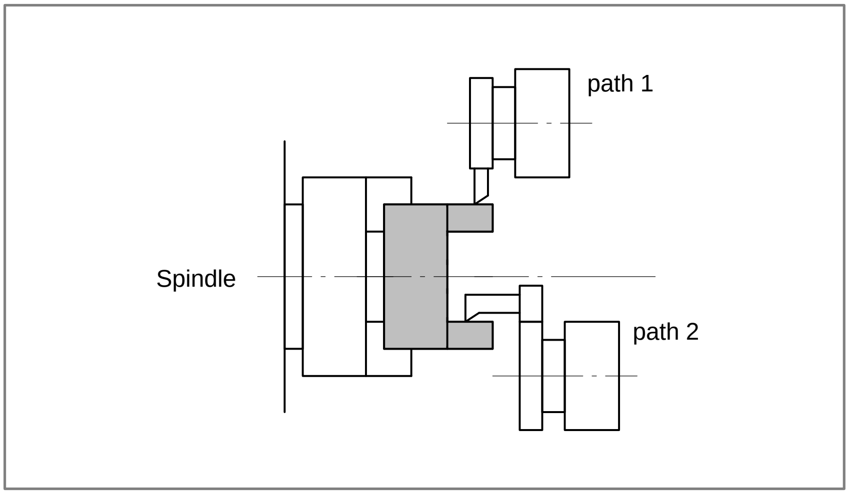
P : Gear helix angle

G80: Cancels synchronization

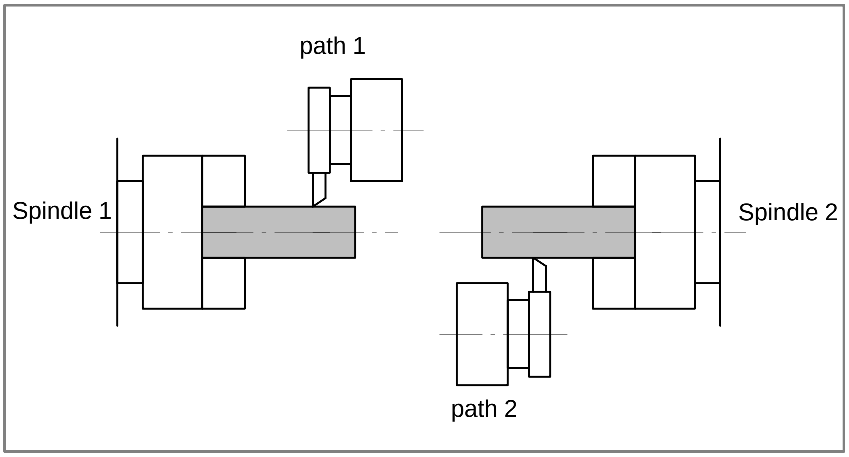
22 FUNCTIONS SPECIFIC TO 2–PATH CONTROL

Two paths can be independently controlled to cut the workpiece simultaneously.

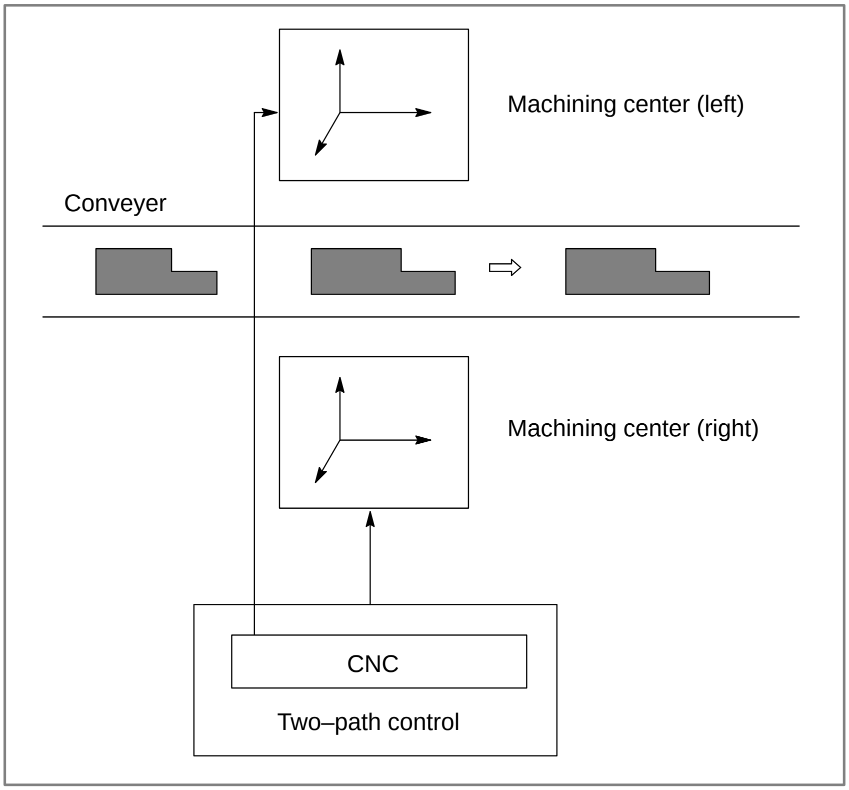
- **Application to a lathe with one spindle and two paths (T series)**
  Two paths can operate simultaneously to machine one workpiece attached to the spindle.
  For example, while one path performs external machining, the other path can perform internal machining. The machining time is then reduced greatly.



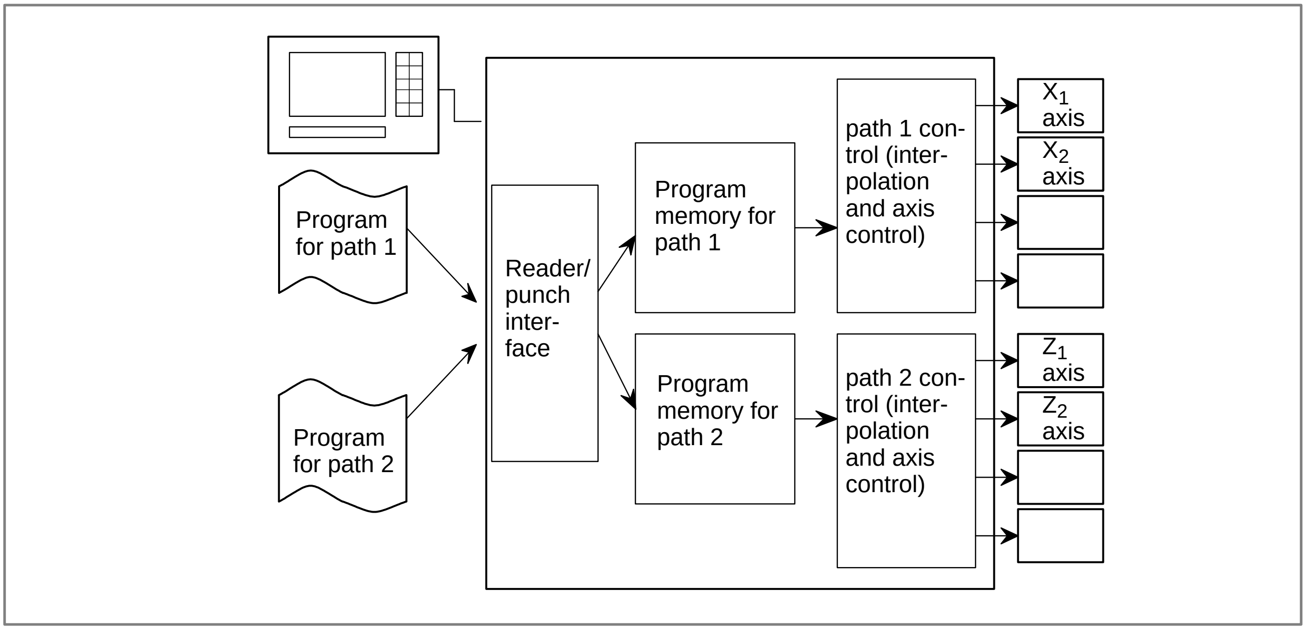
- **Application to a lathe with two spindles and two paths (T series)**
  Two paths can operate simultaneously to machine two workpieces attached to two spindles. Since each path operates independently, the productivity is improved as if two lathes were used simultaneously.



- **Application to transfer line (M series)**
  A single CNC can independently control two machining centers mounted on both sides of the transfer line.
  Application to transfer line (M series)



- **Controlling two paths simultaneously and independently**
  The movement of each path is separately programmed and stored in the program memory for path. In automatic operation, this function selects the program for path 1 and that for path 2 from the program memory. When the paths are activated, the selected programs are executed simultaneously and independently. To make paths 1 and 2 synchronous during machining, the synchronization function (Section 22.2) can be used.

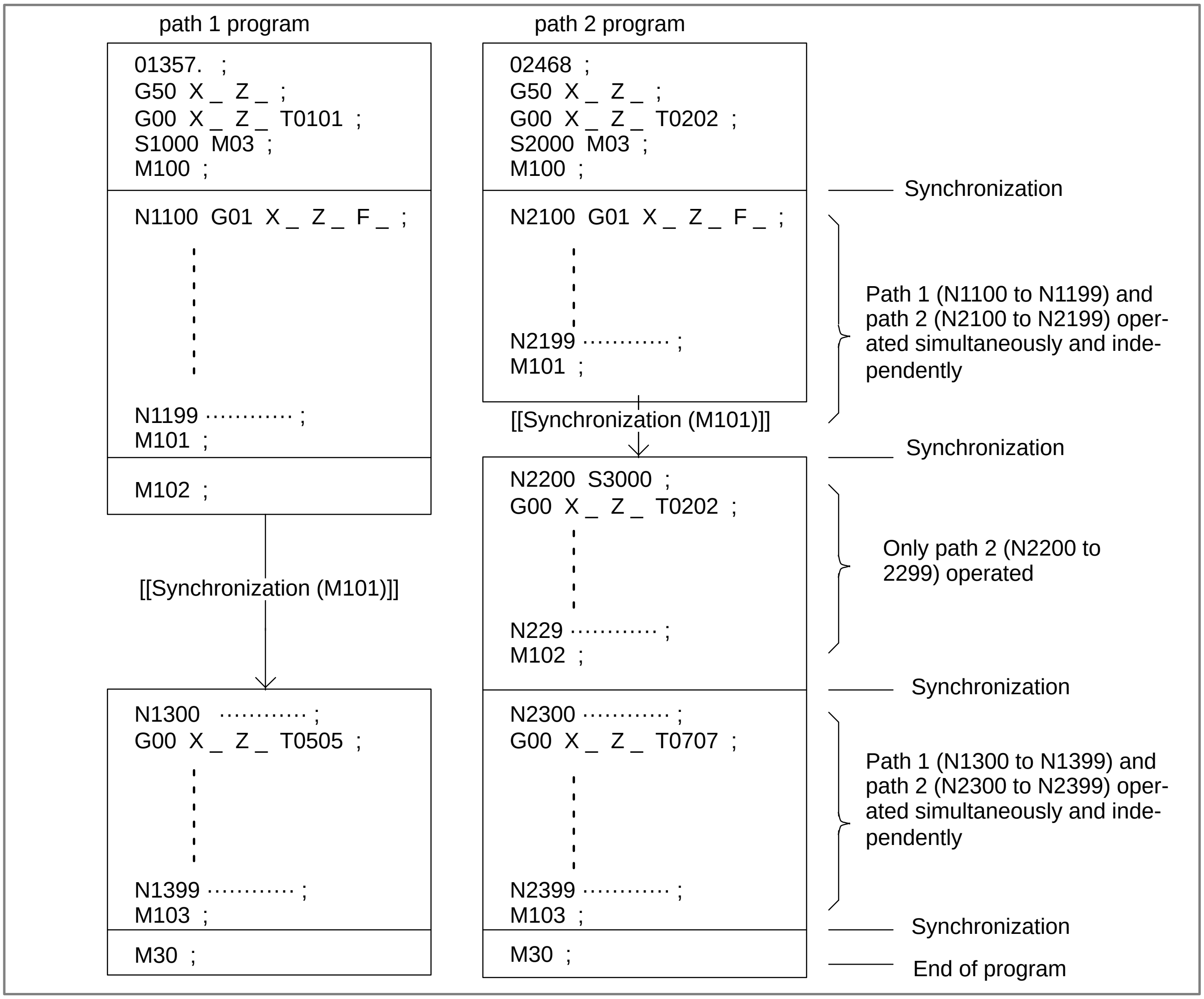


Only one MDI panel is provided for two paths. The path selection signal is used to switch the panel operation or display between paths 1 and 2.

22.1 WAITING FUNCTION

The M code controls the timing of paths 1 and 2 during machining. When the synchronization M code is specified in the machining program of each path, the paths are synchronized at the specified block. During automatic operation, if the synchronization M code is specified at one path, the path waits until the same M code is specified at the other path. After that, the next block is executed.
The range of the synchronization M codes to be used is set in a parameter beforehand.

Example) The synchronization M codes are M100 to M300.



path 1 program

```
01357.   ;
G50  X _  Z _  ;
G00  X _  Z _  T0101  ;
S1000  M03  ;
M100  ;

N1100  G01  X _  Z _  F _  ;
   ⋮
N1199 ………… ;
M101  ;

M102  ;

N1300   ………… ;
G00  X _  Z _  T0505  ;
   ⋮
N1399 ………… ;
M103  ;

M30  ;
```

path 2 program

```
02468  ;
G50  X _  Z _  ;
G00  X _  Z _  T0202  ;
S2000  M03  ;
M100  ;

N2100  G01  X _  Z _  F _  ;
   ⋮
N2199 ………… ;
M101  ;

N2200  S3000  ;
G00  X _  Z _  T0202  ;
   ⋮
N229 ………… ;
M102  ;

N2300 ………… ;
G00  X _  Z _  T0707  ;
   ⋮
N2399 ………… ;
M103  ;

M30  ;
```

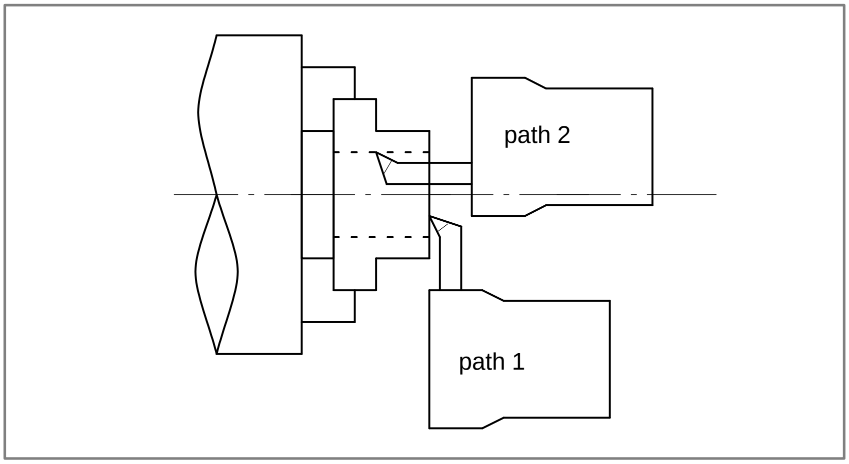
NOTE

1 While a path is waiting because of a synchronization M code, if a different synchronization M code is specified from the other path, an alarm occurs. Both paths, then stop operating.
2 Unlike other M codes, the code signal and strobe signal are not output for the synchronization M code.
3 The synchronization signal is output from the path which is waiting.
4 The synchronization-ignore signal can be used to ignore the synchronization M code specified in the machining program. This signal is used when only one path is used for machining.

22.2 PATH INTERFERENCE CHECK

T series

When one workpiece is machined by two paths operating simultaneously, paths may come close to each other. If these paths touch each other because of a program error or setting error, the tool or even the machine may be damaged.
If such an accident is expected, the path interference check function decelerates and stops the paths.

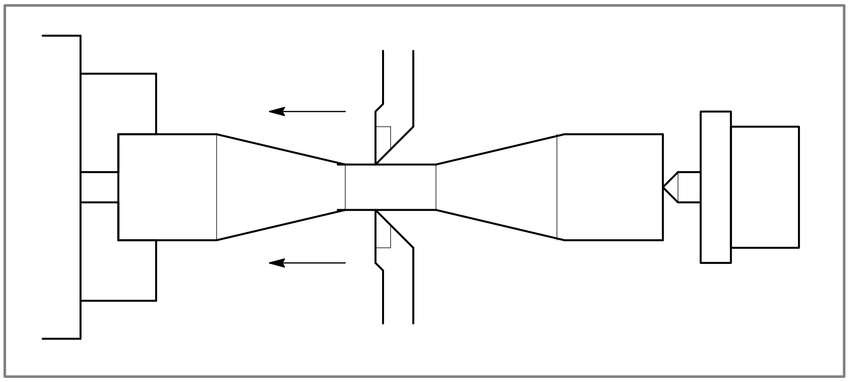


To execute the path interference check, the contour of each path (contour including the tool mounted on the path) must be set as a contact-inhibited area for each tool beforehand.
This function checks if the contact-inhibited areas of these paths overlap. If they overlap each other, it determines that the paths have interfered with each other, and decelerates them till they stop as an alarm.

22.3 BALANCE CUT (G68, G69)

T series

To machine a fine workpiece, two cutting tools should be applied on both sides of a workpiece as shown below. When only one tool is applied, the other side of the workpiece may be deflected. Using two cutting tools can make machining precision higher. If, however, these tools are not moved synchronously, the workpiece will shake and machining will not be done precisely. The balance cut function makes paths move synchronously.



NOTE
The balance cut function cannot be used if the option of mirror-image operation of facing paths is specified.

22.4 MEMORY COMMON TO PATHS

A machine with two paths has different custom macro common variables and tool compensation memory areas for paths 1 and 2. paths 1 and 2 can share the custom macro common variables and tool compensation memory areas provided certain parameters are specified accordingly.

- Custom macro common variables

Paths 1 and 2 can share all or part of custom macro common variables #100 to #149 and #500 to #531, provided parameters 6036 and 6037 are specified accordingly. (The data for the shared variables can be written or read from either path.)

- Tool compensation memory

Path 2 can reference or specify the data in the tool compensation memory area of path 1, provided the CMF bit (bit 5 of parameter 8100) is specified accordingly. This can be executed only when paths 1 and 2 have identical data for tool compensation (number of groups, number of columns, unit system, etc.).

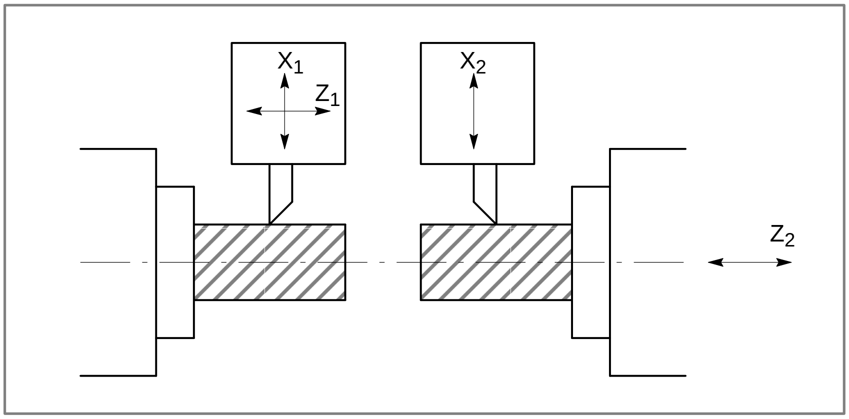
22.5 SYNCHRONIZATION/ MIX CONTROL

T series

In 16-TB(2–path control), usually the axes belonging to path 1 (X1, Z1,...) are moved by the move command of path 1. The axes belonging to path 2 (X2, Z2, ...) are moved by that of path 2 (individual path control). The synchronization/mix control function can move an optional axis of one path and that of the other path synchronously (synchronization control). This control function can exchange the move commands for optional axes between two paths (mix control).

- **Individual path control**

The axes belonging to path 1 (X1, Z1,...) are moved by the move command of path 1. The axes belonging to path 2 (X2, Z2, ...) are moved by that of path 2.



- **Synchronization control**

The move command for an axis (master axis) is given also to another optional axis (slave axis). These two axes are then moved synchronously. The slave axis can be moved also by its own move command. Which command to use can be selected by the synchronization control selection signal from PMC.

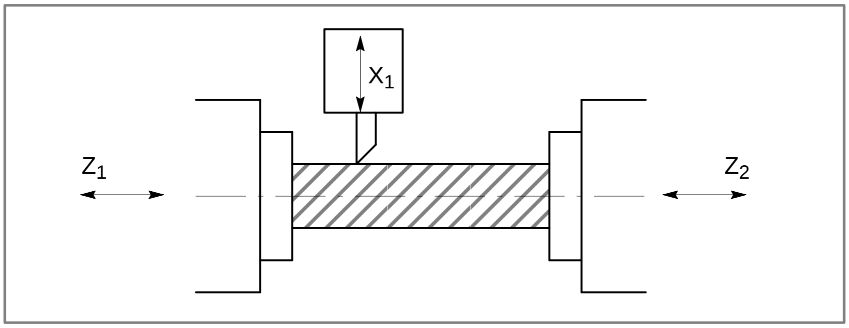
NOTE

1 Synchronization here means issuing the master axis move command to the master axis and also to the slave axis simultaneously. The position deviation of master and slave axes is always detected. It, however, is not compensated because synchronization compensation is not performed. If a deviation exceeding the limit set in the parameter is detected, an alarm occurs and the movements of both axes are stopped.

2 The master axis and slave axis do not need to belong to the same path. Two or more slave axes can be specified for one master axis.

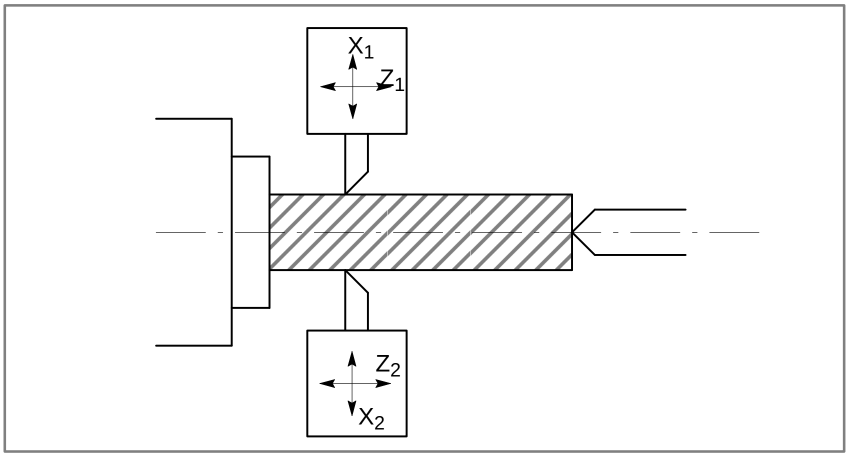
Example 1)

The Z2 axis is synchronized with the Z1 axis (machining with both ends of a workpiece being held).



Example 2)

The X2 and Z2 axes are synchronized with the X1 and Z1 axes (balance cut).



- **Mix control**

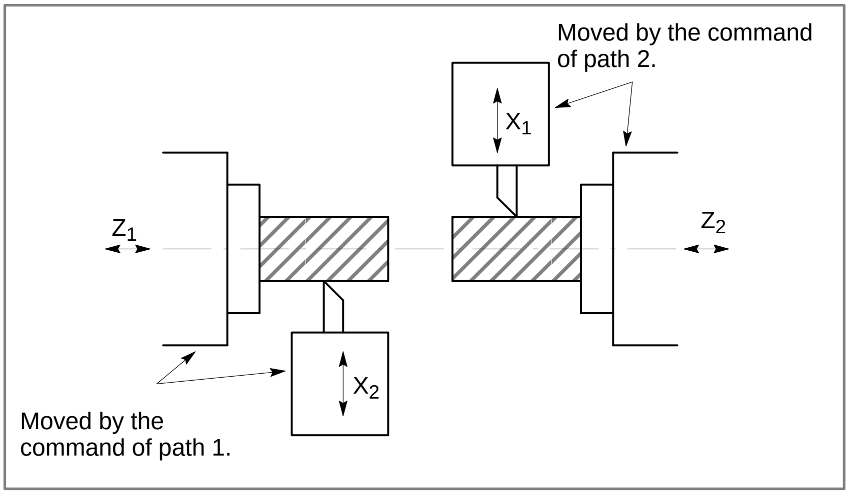
When moving axes, the move commands for optional axes can be exchanged between two paths.

Example 1)

The move commands for X_1 and X_2 axes are exchanged.

The program command of path 1 moves X_2 and Z_1 axes.

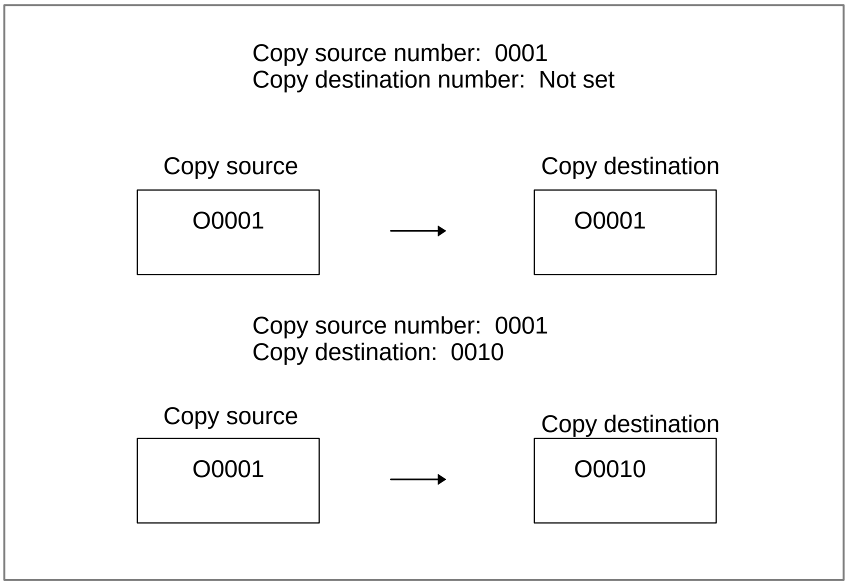
The program command of path 2 moves X_1 and Z_2 axes.



22.6 COPYING A PROGRAM BETWEEN TWO PATHS

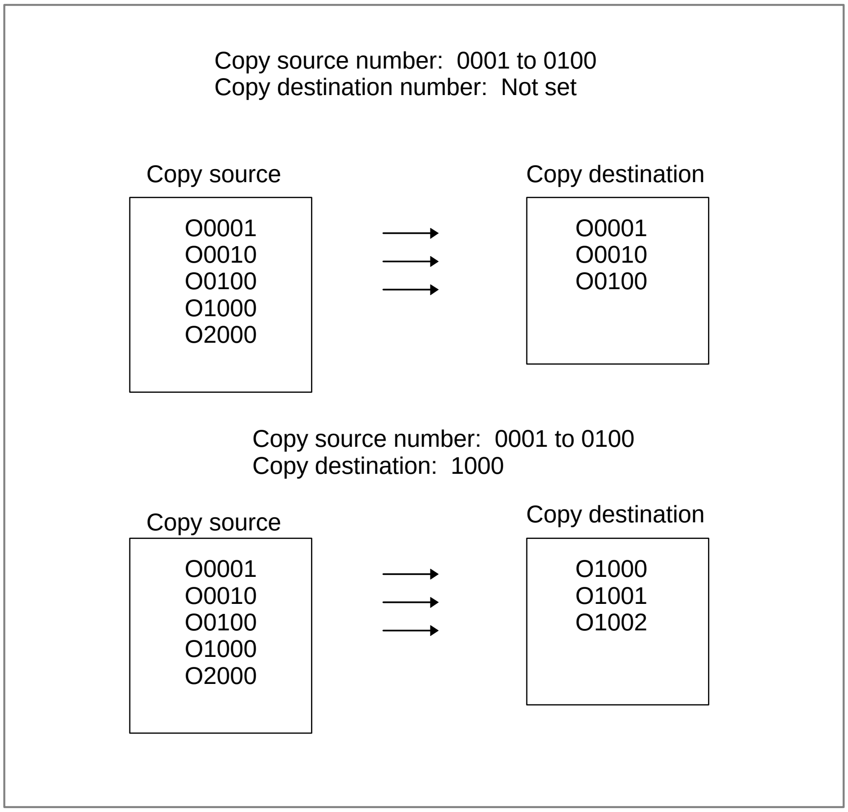
In a CNC supporting two–path control, specified machining programs can be copied between the two paths by setting a parameter accordingly. A copy operation can be performed by specifying either a single program or a range.

- **Single–program copy**


Copy source number: 0001
Copy destination number: Not set

Copy source number: 0001
Copy destination: 0010

- **Specified–range copy**


Copy source number: 0001 to 0100
Copy destination number: Not set

Copy source number: 0001 to 0100
Copy destination: 1000

23 MANUAL OPERATION

23.1 MANUAL FEED

- **Jogging**
  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.

23.2 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

23.3 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

23.4 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 system, 2 axes) can be done at a time. Multiplier is common to 1st, 2nd and 3rd manual pulse generators.

23.5

HANDLE FEED IN THE SAME MODE AS FOR JOGGING

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).

23.6

T series

MANUAL PER-ROTATION FEED

The feedrates in manual continuous feed (jogging) 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.

23.7

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.

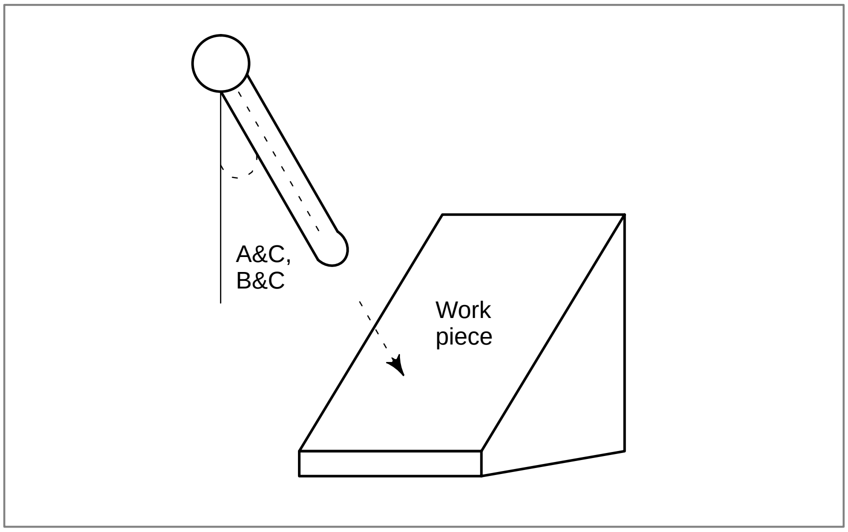
23.8

TOOL AXIS DIRECTION HANDLE FEED AND TOOL AXIS DIRECTION HANDLE FEED B (FOR M SERIES)

The tool axis direction handle feed function allows the tool to be moved a specified distance by handle feed, along the axis of the tool, tilted by rotating the rotation axes.
Tool axis direction handle feed function B provides two functions: handle feed along the tool axis and that perpendicular to the tool axis.
These functions are used for applications such as 5–axis diesinking machining.

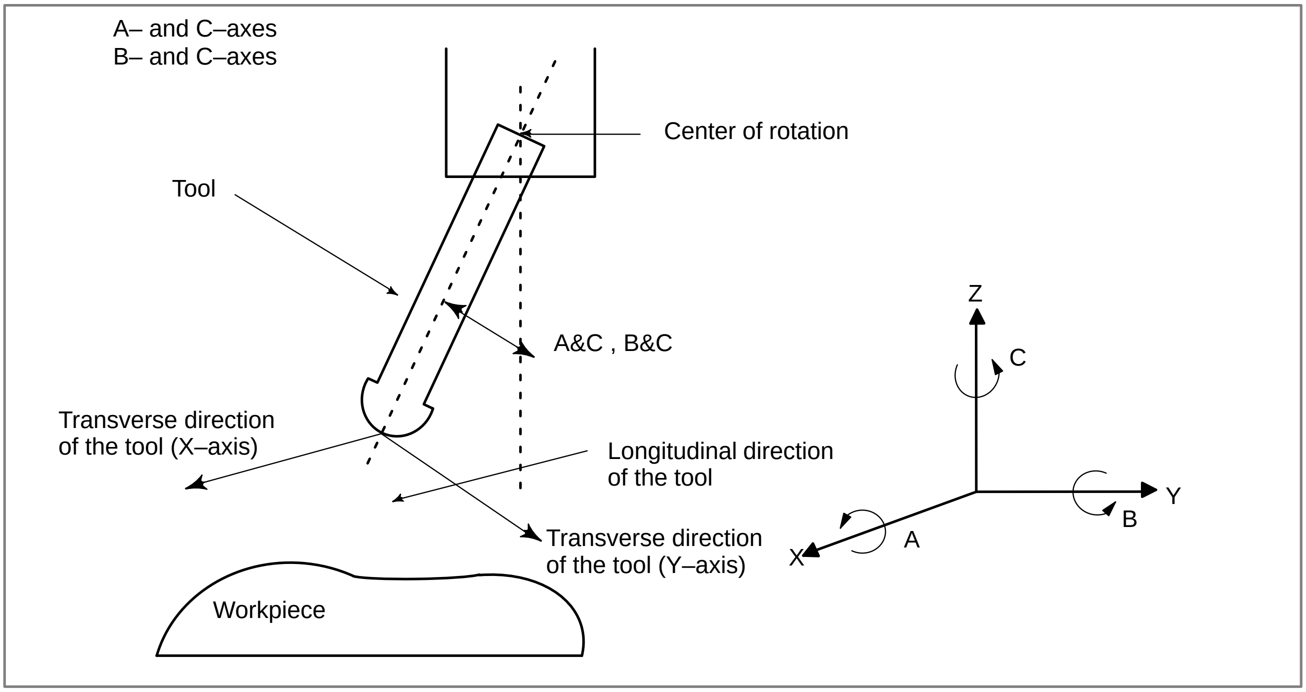
23.8.1
Tool Axis Direction Handle Feed

When the tool axis direction handle mode is selected and the manual pulse generator is rotated, the tool is moved by the specified travel distance in the direction of the tool axis tilted by the rotation of the rotary axis.



23.8.2
Tool Axis Normal Direction Handle Feed

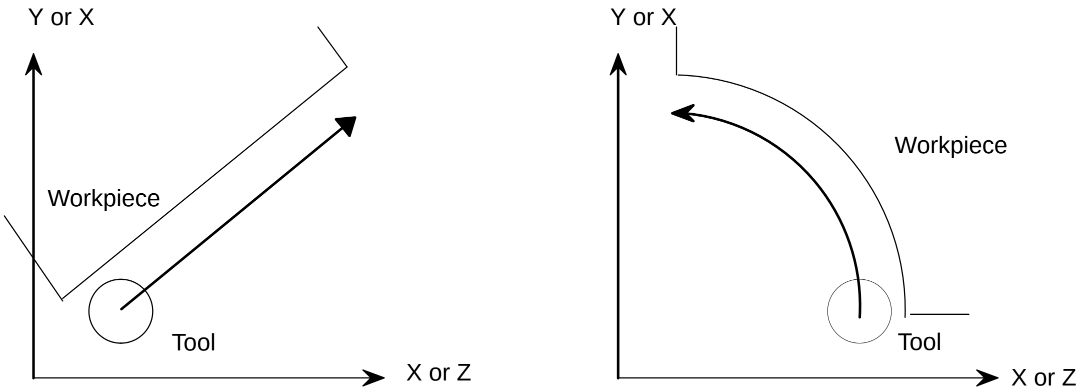
When the tool axis normal direction handle mode is selected and the manual pulse generator is rotated, the tool is moved by the specified travel distance in the direction normal to the tool axis tilted by the rotation of the rotary axis.



23.9
MANUAL LINEAR/CIRCULAR INTERPOLATION (ONLY FOR ONE PATH)

In manual handle feed or jog feed, the following types of feed operations are enabled in addition to the feed operation along a specified single axis (X–axis, Y–axis, Z–axis, and so forth) based on simultaneous 1–axis control:

- Feed along a tilted straight line in the XY plane (M series) (linear feed) or in the ZX plane (T series) (linear feed) based on simultaneous 2–axis control
- Feed along a circle in the XY plane (M series) (circular feed) or in the ZX plane (T series) (circular feed) based on simultaneous 2–axis control



<<Linear feed>> <<Circular feed>>

NOTE

The X–axis and Y–axis (M series) or Z–axis and Xaxis (T series) must be the first controlled axis and second controlled axis, respectively.

23.10

M series

MANUAL RIGID TAPPING

For execution of rigid tapping, set rigid mode with MDI mode, then switch to handle mode and select the tapping axis and move a manual handle. Manual rigid tapping is available when parameter is set.

The rotation direction of the spindle in manual rigid tapping is determined by a specified tapping cycle G code and the setting parameter.

23.11 MANUAL NUMERIC COMMAND

The manual numeric command function allows data programmed through the MDI to be executed in jog mode. Whenever the system is ready for jog feed, a manual numeric command can be executed. The following eight functions are supported:

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)
(5) M codes (miscellaneous functions)
(6) S codes (spindle functions)
(7) T codes (tool functions) (M series)
(8) B codes (second auxiliary functions)

By setting the corresponding parameters, the following commands for axial motion and the M, S, T, and B functions can be disabled:

(1) Positioning (G00)
(2) Linear interpolation (G01)
(3) Automatic reference position return (G28)
(4) 2nd/3rd/4th reference position return (G30)
(5) M codes (miscellaneous functions)
(6) S codes (spindle functions)
(7) T codes (tool functions) (M series)
(8) B codes (second auxiliary functions)

24 AUTOMATIC OPERATION

24.1
OPERATION MODE

24.1.1
DNC Operation

The part program can be read and executed block by block from the input device connected to the reader/puncher interface.

24.1.2
Memory Operation

Program registered in the memory can be executed.

24.1.3
MDI Operation

Multiple blocks can be input and executed by the MDI unit.

24.2 SELECTION OF EXECUTION PROGRAMS

24.2.1 Program Number Search

Program number currently in need can be searched from the programs registered in memory operating the MDI.

24.2.2 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.

24.2.3 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)

24.2.4 External Workpiece Number Search

By specifying work numbers of 01 - 15 externally (from the machine side, etc.), program corresponding to the work number can be selected.
The work number equals the program number. For example when work number 12 is specified, program, O0012 is selected.

24.3

ACTIVATION OF AUTOMATIC OPERATION

24.3.1

Cycle Start

Set operation mode to memory operation, MDI operation, or DNC operation, press the cycle start button, and automatic operation starts.

24.4

EXECUTION OF AUTOMATIC OPERATION

24.4.1

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.

24.5 AUTOMATIC OPERATION STOP

24.5.1 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.

24.5.2 Program End (M02, M30)

The CNC is reset after executing the M02 (end of program) or M30 (end of tape) commanded block.

24.5.3 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.

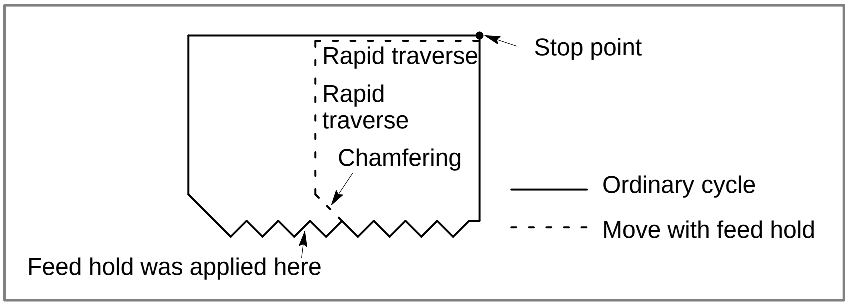
24.5.4 Feed Hold

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.

24.5.5 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.

24.5.6 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.

24.6
RESTART OF AUTOMATIC OPERATION

24.6.1
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.

24.6.2
Tool Retract & Recover

These functions are used for replacing tools damaged retraction of tools for confirming the cutting conditions, and recovering the tools efficiently to restart the cutting.
Also, the escape operation can be performed with the tool retract signal by previously setting the escape amount (position) with a program. This can be used for retraction for detecting tool damage.

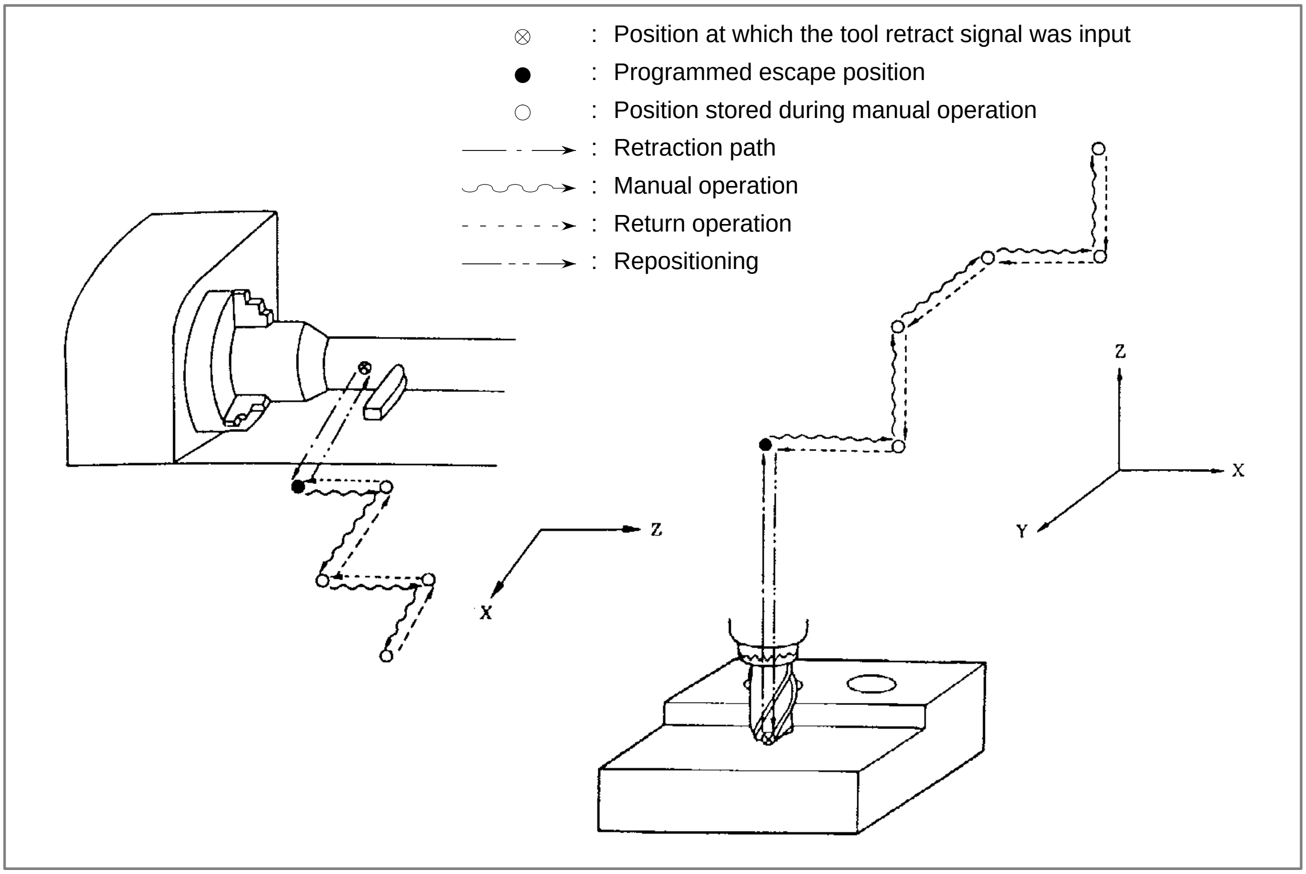
1 Input the tool retract signal during executing the automatic operation. Then, the automatic operation is halted and the escape operation (retraction) is performed to the escape position commanded by the program.

2 Input the tool retract signal to initiate the retract mode.

3 After that, switch the automatic mode to the manual mode to move tools with manual operation such as the jog feed and handle feed. A maximum of 10 points can be automatically memorized as travel path.

4 Input the tool recovery signal to return the tool to the retraction position in the opposite direction along the path moved by manual operation automatically (recovery operation).

5 Perform the cycle start to return the tool to the position where the tool retract signal was entered (repositioning). When the recovery operation completes, the halted automatic operation resumes.



Command the escape amount using the G10.6.

G10.6 _ ;

The escape data sorted by G10.6 is valid until the next G10.6 is commanded. Command the following to cancel the escape amount:

G10.6 ; (Single command)

where

The G10.6 is the one-shot G code.

The tool can be retracted to a special location of work coordinate system when the escape amount is command by the ABSOLUTE (G90). When the escape amount is commanded by the INCREMENTAL (G91), the tool can retract by only the commanded escape amount.

24.6.3 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.

24.6.4 **M series**
Machining Return and Restart Functions

The machining return and restart functions are based on the PMC and custom macros. If machining is stopped by the issue of a reset or emergency stop, the machining return function returns the tool from the point at which machining was stopped and the machining restart function restarts machining from the start block.

The following operations constitute the functions:

(1) Controlling the machining cycle according to specified sequence numbers

The following sequence numbers are used to control the machining cycle:

N7000 to N7998 : Machining start point
N7999 : Clears the data for machining return and restart.

(The data for machining return and restart is not cleared until N7999 is specified.)

N8000 to N8999 : Machining cycle start point
N9000 to N9999 : Machining cycle end point

(2) Saving the position and modal information at the machining start point and machining cycle start point to custom macro variables

(3) Rigid tapping return function

(4) Restarting machining from the machining start point or machining cycle start point

24.7
MANUAL INTERRUPTION DURING AUTOMATIC OPERATION

24.7.1
Handle Interruption

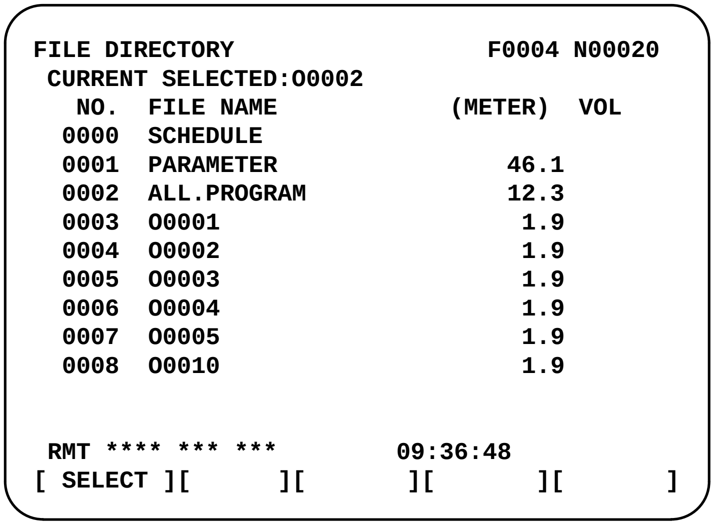
During automatic operation, tool can be adjusted by the manual pulse generator without changing the mode. The pulse from the manual pulse generator is added to the automatic operation command and the tool is moved for the recommended pulses.
The work coordinate system thereafter is shifted for the pulse commanded value. Movement commanded by handle interruption can be displayed.

24.8 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.

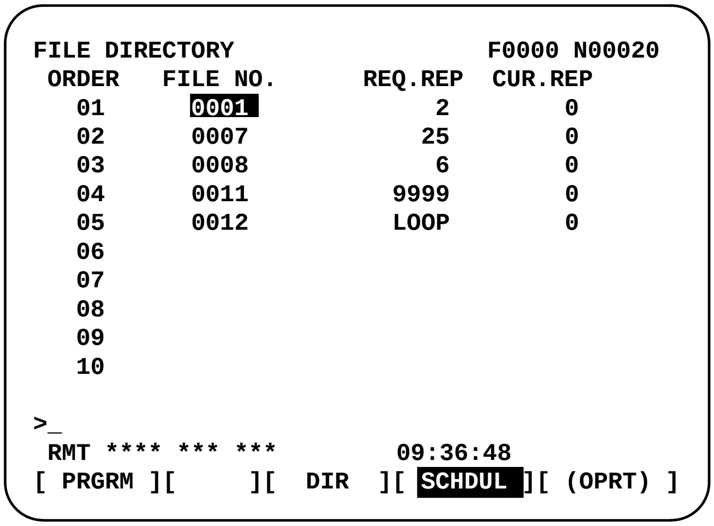
- A list of the files stored on the Floppy Cassette can be displayed.
- Files can be executed in an arbitrary order and executed an arbitrary number of times by specifying file numbers in a desired order along with their repeat counts.

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) ]
```

24.9 SIMULTANEOUS INPUT AND OUTPUT OPERATIONS (AT 1–PATH CONTROL)

M series

While a tape is running, a program input from an I/O device connected to the reader/punch interface can be executed and stored in memory. Similarly, a program stored in memory can be executed and output through the reader/punch interface at the same time.

24.10 RETRACE FUNCTION

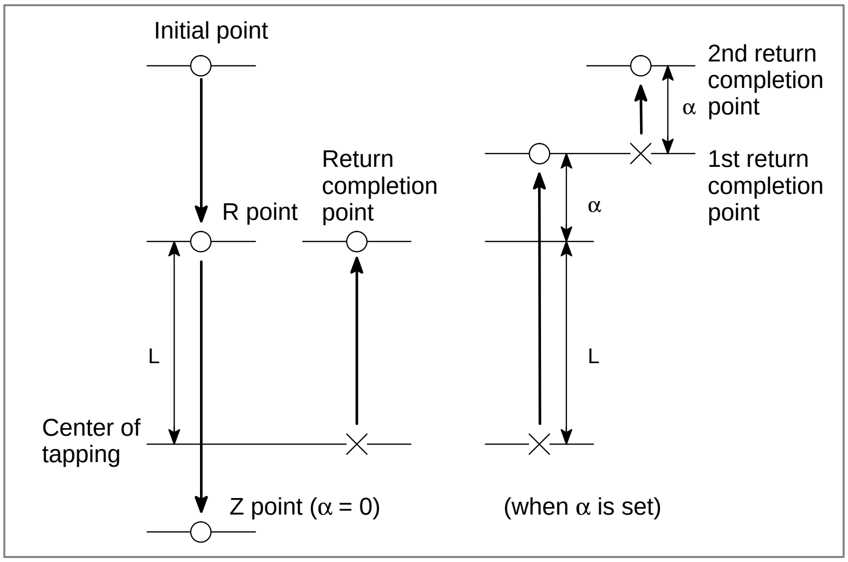
M series

With the retrace function, the tool can be moved in the reverse direction (reverse movement) by using the REVERSE switch during automatic operation to trace the programmed path. The retrace function also enables the user to move the tool in the forward direction again (forward return movement) along the retraced path until the retrace start position is reached. When the tool reaches the retrace start position, the tool resumes movement according to the program.

24.11 RIGID TAPPING RETURN

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.



25

PROGRAM TEST FUNCTIONS

25.1
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.

25.2
MACHINE LOCK ON EACH AXIS

Machine lock can be commanded per axis.

25.3
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.

25.4
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.

25.5
SINGLE BLOCK

The program can be executed block by block under automatic operation.

26 SETTING AND DISPLAY UNIT

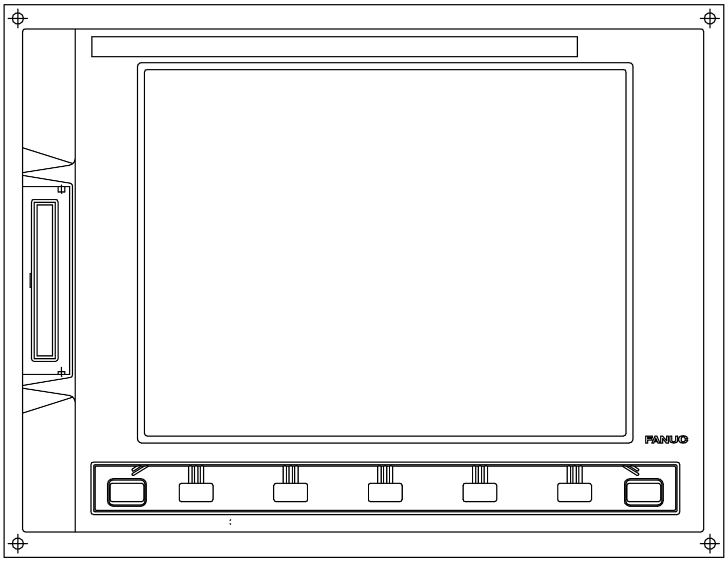
The available operational devices include the setting and display unit attached to the CNC, the machine operator's panel, and external input/output devices such as a tape reader, PPR, Handy File, Floppy Cassette, and FA Card.

26.1
SETTING AND DISPLAY UNIT

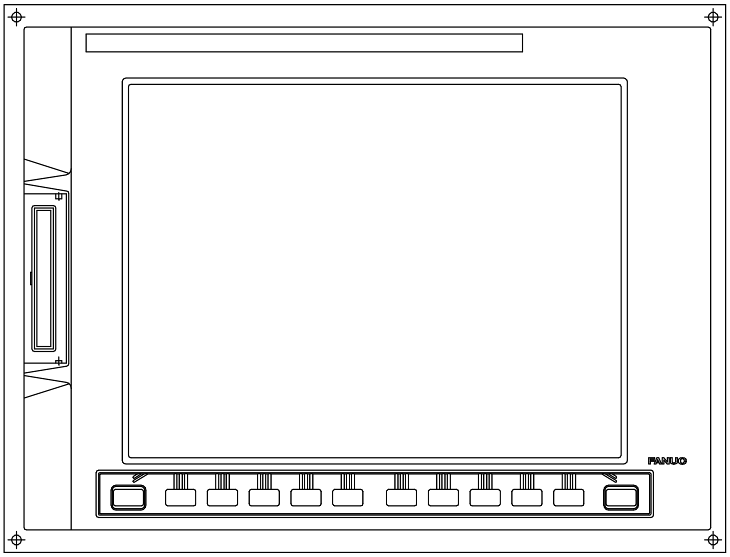
The setting and display units are shown in Subsections II–26.1.1 to II–26.1.6.

CNC control unit with 7.2″/8.4″ LCD: II–26.1.1
CNC control unit with 9.5″/10.4″ LCD: II–26.1.2
Separate–type small MDI unit: II–26.1.3
Separate–type standard MDI unit (horizontal type): II–26.1.4
Separate–type standard MDI unit (vertical type): II–26.1.5
Separate–type standard MDI unit (vertical type) (for 160*i*/180*i*): II–26.1.6

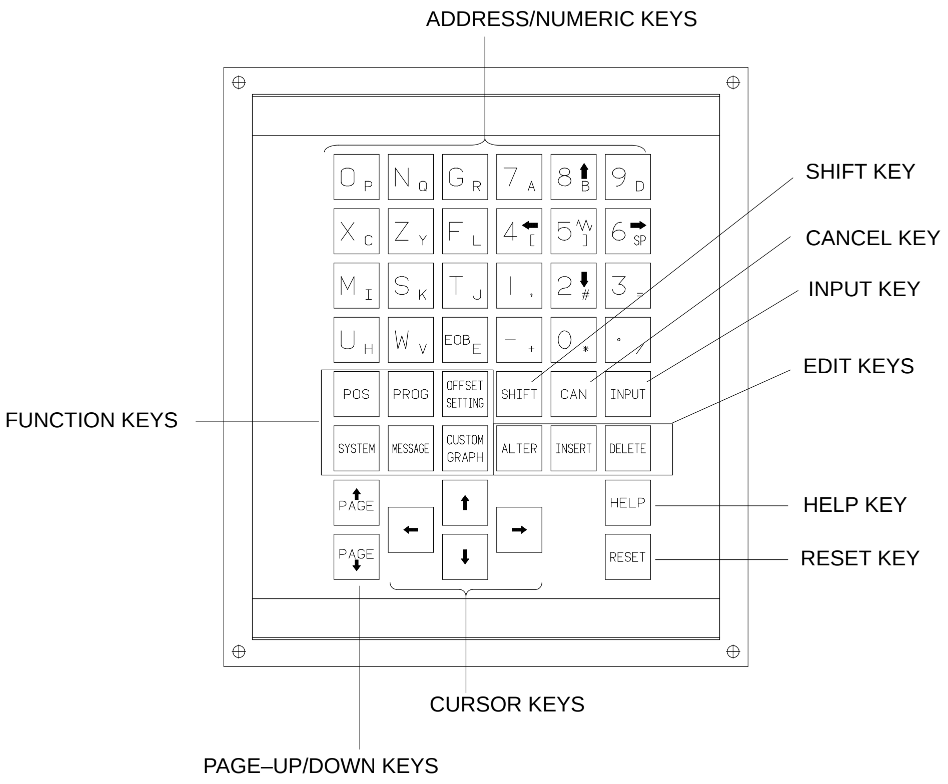
26.1.1
CNC Control Unit with 7.2″/8.4″ LCD



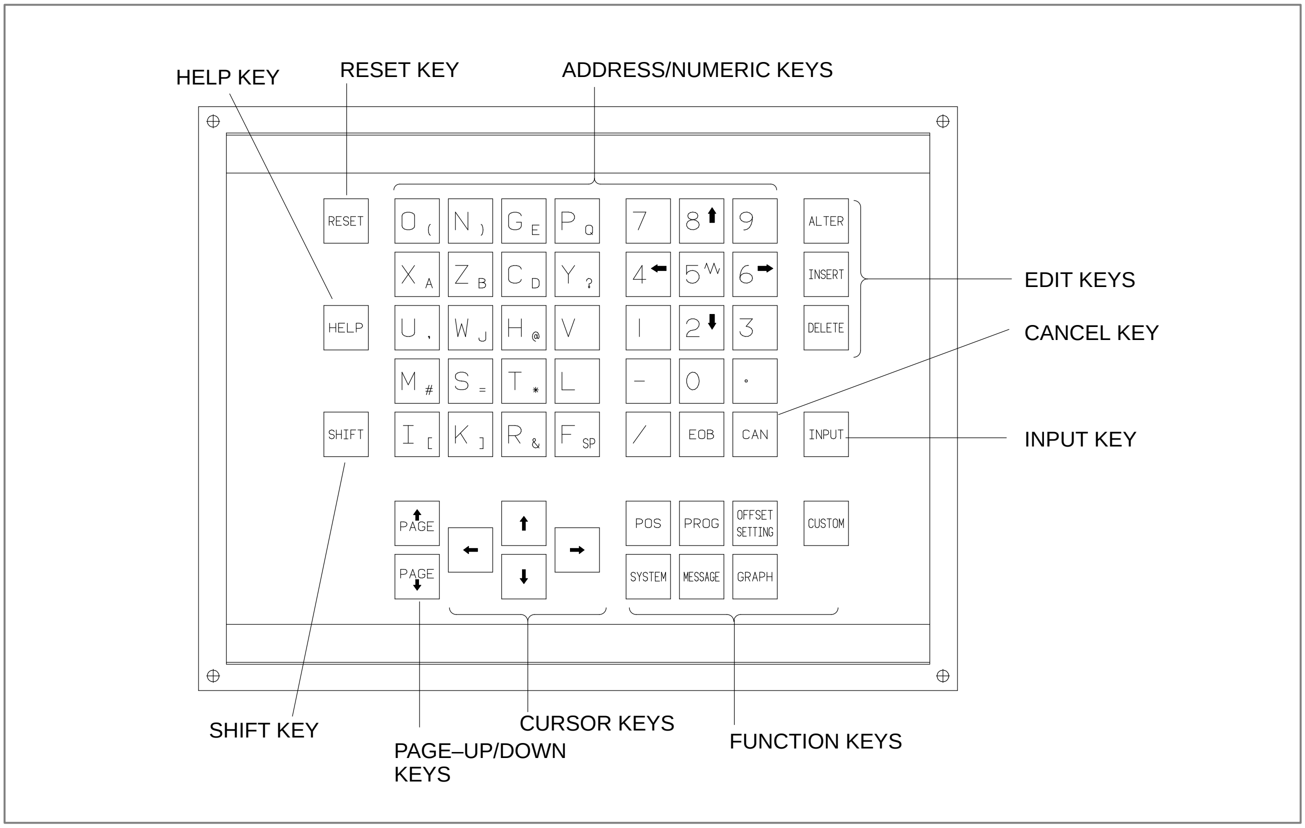
26.1.2
CNC Control Unit with 9.5″/10.4″ LCD



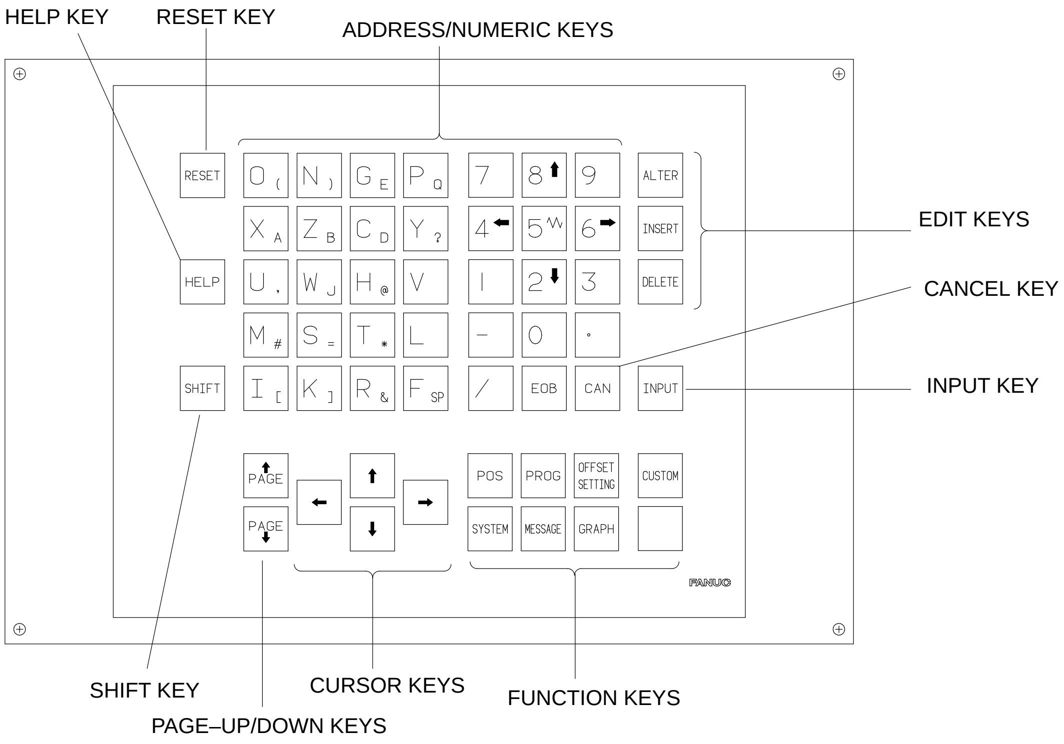
26.1.3
Separate–Type Small MDI Unit



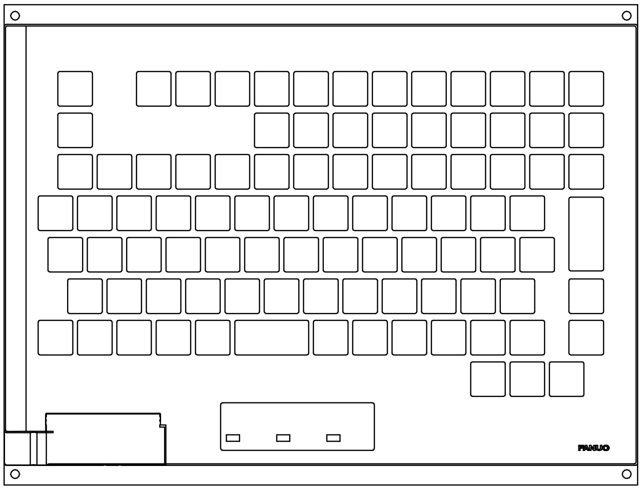
26.1.4 Separate–Type Standard MDI Unit (Horizontal Type)



26.1.5 Separate–Type Standard MDI Unit (Vertical Type)



26.1.6
Separate–Type FA Full Keyboard (Vertical Type) (for 160*i*/180*i*/210*i*)



The key legends are the same as those of a personal computer keyboard.

26.2 EXPLANATION OF THE KEYBOARD

No.	Key	Function
(1)	Reset key RESET	Used to reset the CNC to release an alarm or other similar state.
(2)	Help key HELP	Used to get help with operations such as for the MDI keys, when the operator does not know what to do next.
(3)	Soft keys	The soft keys are assigned different functions depending on the application. The functions currently assigned to the soft keys are displayed on the lowermost line of the screen.
(4)	Address/numeric keys N) 4 ← …	Used to enter letters and numbers.
(5)	Shift key SHIFT	Some of the address keys have two different letters. When the shift key is pressed first before pressing one of these address keys, the lower-right letter is input. When the shift key is pressed, ^ is displayed in the key input buffer indicating that the lower-right letter will be input.
(6)	Input key INPUT	Data input by pressing an address or numeric key is stored in the key input buffer, then displayed. When data input to the key input buffer needs to be written to the offset register, press the <INPUT> key. This key is equivalent to soft key [INPUT]. Either key may be used.
(7)	Cancel key CAN	Used to delete letters or numbers input to the key input buffer. Example) When N001X100Z is displayed on the key input buffer, pressing the cancel key deletes the letter Z, and N001X100 is displayed.
(8)	Edit keys ALTER INSERT DELETE	Used to edit programs. ALTER : Alter INSERT : Insert DELETE : Delete
(9)	Function keys POS PROG …	Used to switch screens for each function.

No.	Key	Function
(10)	Cursor keys ← ↑ ↓ →	Four cursor keys are provided. → : Moves the cursor to the right or forwards in small units. ← : Moves the cursor to left or backwards in small units. ↓ : Moves the cursor downward or forwards in large units. ↑ : Moves the cursor upward or backwards in large units.
(11)	Page-up/down keys ↑ PAGE PAGE ↓	Page-up and page-down keys are provided. PAGE ↓ : Used to display the next page. ↑ PAGE : Used to display the previous page.

26.2.1 Explanation of the Function Keys

The function keys select what is displayed. Each function is divided into sub-functions, and the sub-functions are selected by soft keys.

There are six function keys : POS , PROG , OFFSET SETTING , SYSTEM , MESSAGE , and GRAPH .

POS : Displays the current position.

PROG : Displays and edits a program stored in memory.

OFFSET SETTING : Displays an offset value, offset from the workpiece zero point, custom macro variable, and tool life management data. Allows data to be input into these items.

SYSTEM : Displays and sets a parameter and pitch error compensation value, and displays self diagnostic data.

MESSAGE : Displays an alarm message, external operator message, external alarm message, and alarm history.

GRAPH : Displays graphic data.

26.2.2
Explanation of the Soft Keys

The MDI panel has 10 soft keys (or 5 soft keys), a next-menu key on the right, and a previous-menu key on the left. The next menu key and previous menu key are used to select the functions of the soft keys.
These soft keys can be assigned with various functions, according to the needs.
The following functions are mainly available via the MDI panel:

- Actual position display
- Contents of program display, program directory display (display of program number, program name, part program storage length left, number of programs left)
- Program editing
- Offset amount display and setting
- Commanded value display, MDI input
- Parameter setting and display
- Alarm message/operator message display
- Custom macro variables display and setting
- Tool life management data display and setting
- Diagnosis
- Others

This manual may refer to a display device with 10 + 2 soft keys as a 12 soft key type, and a display device with 5 + 2 soft keys as a 7 soft key type.

27 DISPLAYING AND SETTING DATA

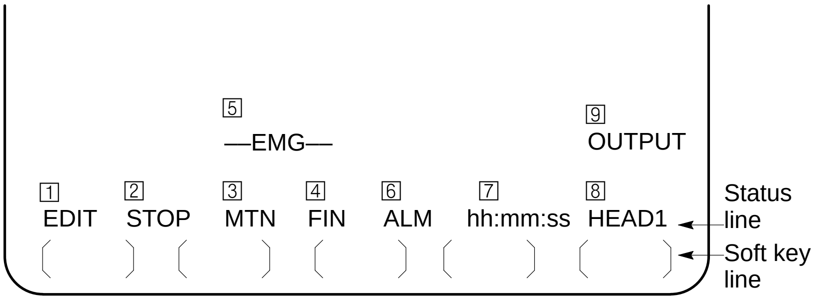
27.1 DISPLAY

The following data are displayed. 7 soft keys can display maximum 640 characters (40 × 16 lines) and 12 soft keys can display maximum 2080 (80 × 26 lines).

Explanations

- **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.



$\boxed{1}$ Operation mode (MDI, MEM, RMT, EDIT, HND, TJOG, THND, INC, or REF)

$\boxed{2}$ 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.

$\boxed{3}$ Axis movement/dwell (MTN, DWL, or ***)

$\boxed{4}$ FIN wait state (FIN or ***)

$\boxed{5}$ Emergency stop (--EMG--) (displayed above in 3 and 4)

$\boxed{6}$ Alarm status (ALM, WNG, or ***)

$\boxed{7}$ Clock (hh:mm:ss)

$\boxed{8}$ 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.

$\boxed{9}$ Status display such as program editing (INPUT, OUTPUT, SRCH, EDIT, LSK, or RSTR)
($\boxed{8}$ and $\boxed{9}$ are displayed in the same column. When a program is being edited, $\boxed{9}$ 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)
  ☐ Actual spindle speed (rpm)

- **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**

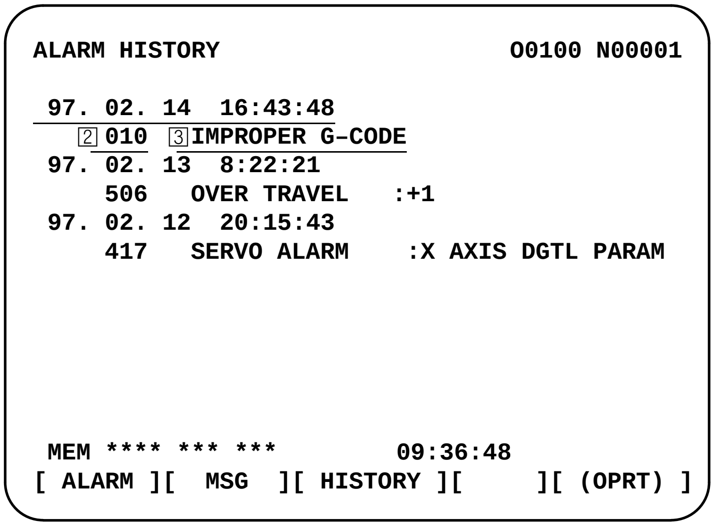
The load values (torque values) of spindle motor and servo motor are displayed in bar chart.
The most recent sampling values are displayed in bar chart display. Set the rated load value of motor corresponding to each load meter to parameters. The load meter displays 100% when the load value is the rated 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

 97. 02. 14  16:43:48
   2 010  3 IMPROPER G–CODE
 97. 02. 13  8:22:21
     506   OVER TRAVEL   :+1
 97. 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.

27.2 LANGUAGE SELECTION

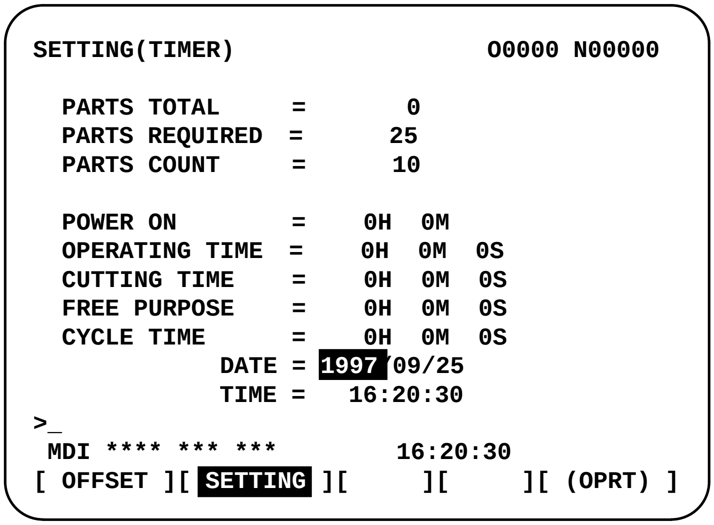
The Japanese, English, German, French, Italian, Spanish, Chinese, and Korean are prepared as display languages. Select the language to be displayed by parameters.

27.3 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.

27.4 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 = 1997/09/25
             TIME =   16:20:30
>_
 MDI **** *** ***         16:20:30
[ OFFSET ][ SETTING ][     ][     ][ (OPRT) ]
```

27.5 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

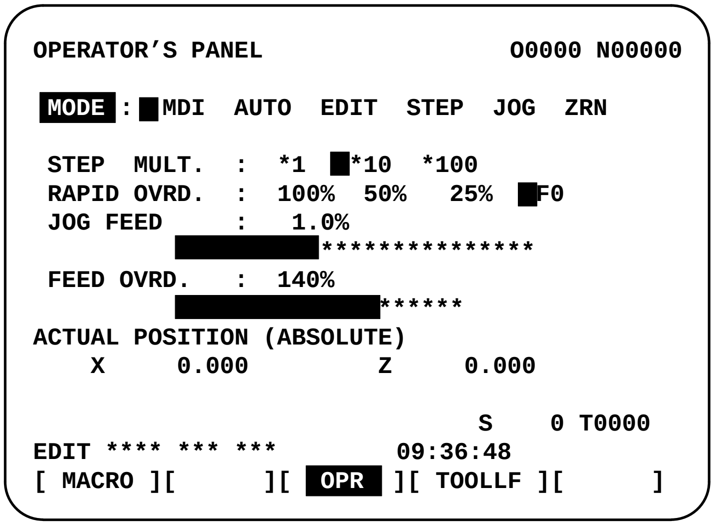
E Memory protect

F Feed hole

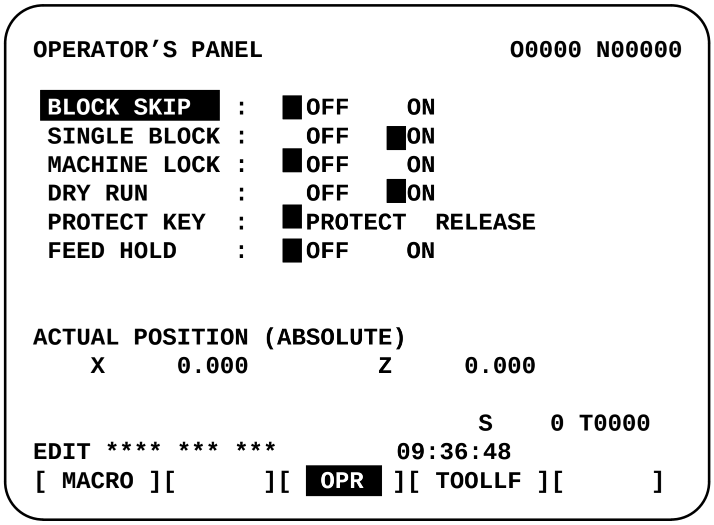
G Jogging/incremental feed axis direction selection
Manual rapid traverse selection

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%  █F0
  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 ][      ]
```

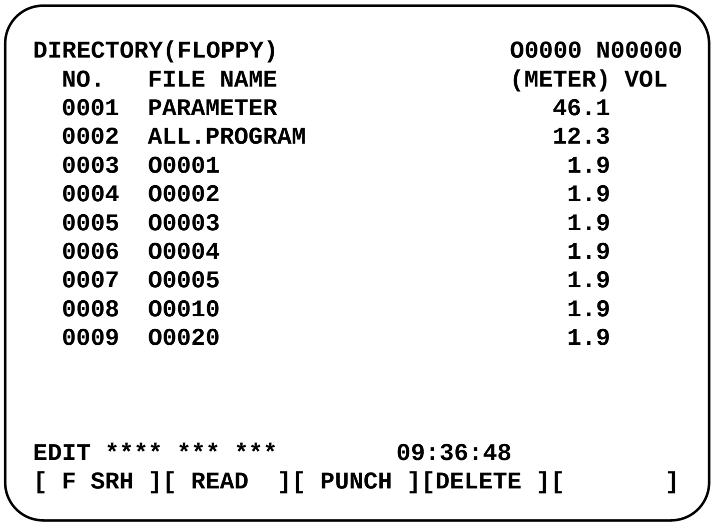
27.6 DIRECTORY DISPLAY OF FLOPPY CASSETTE

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 ][       ]
```

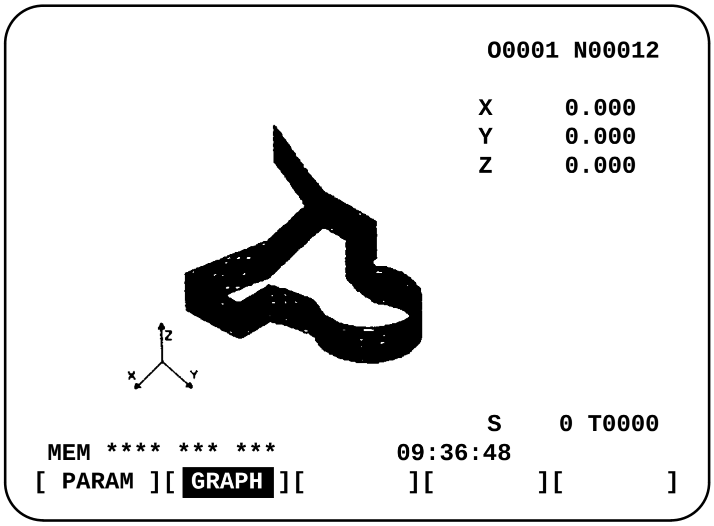
27.7 GRAPHIC DISPLAY FUNCTION

27.7.1 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 system, display is possible with the XY plane, YZ plane, ZX plane, or isometric; for T system, with the XZ plane.
- Scaling of the screen is possible.

Tool path drawing (M series)



```
                              O0001 N00012

                             X     0.000
                             Y     0.000
                             Z     0.000

                              S    0 T0000
 MEM **** *** ***        09:36:48
[ PARAM ][ GRAPH ][       ][       ][       ]
```

Graphic display of tool path (M series)

27.7.2 Dynamic Graphic Display

Created programs can be checked visually by displaying them using graphic data.

Dynamic graphic display function (for M series)

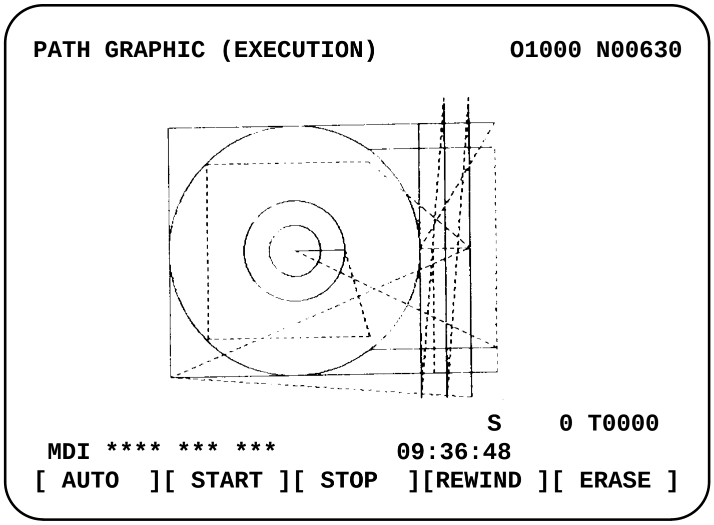
Graphic data can be displayed in the following two drawing modes:

- **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.

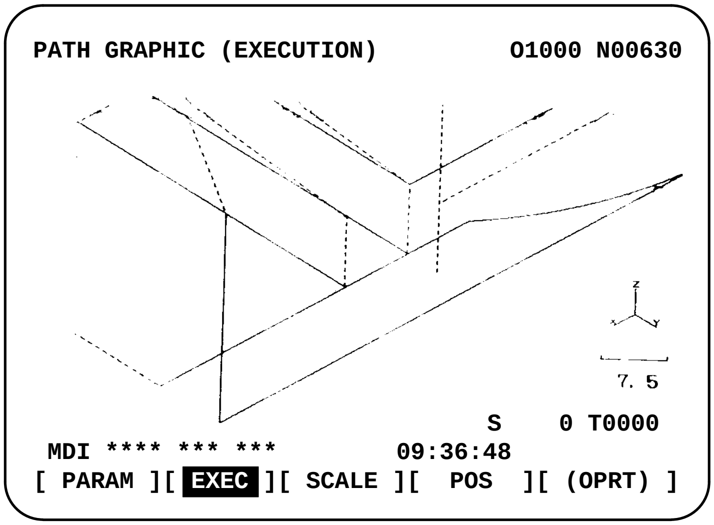
Two-dimensional drawing



```
PATH GRAPHIC (EXECUTION)        O1000 N00630

                                S    0 T0000
 MDI **** *** ***          09:36:48
```

- 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.

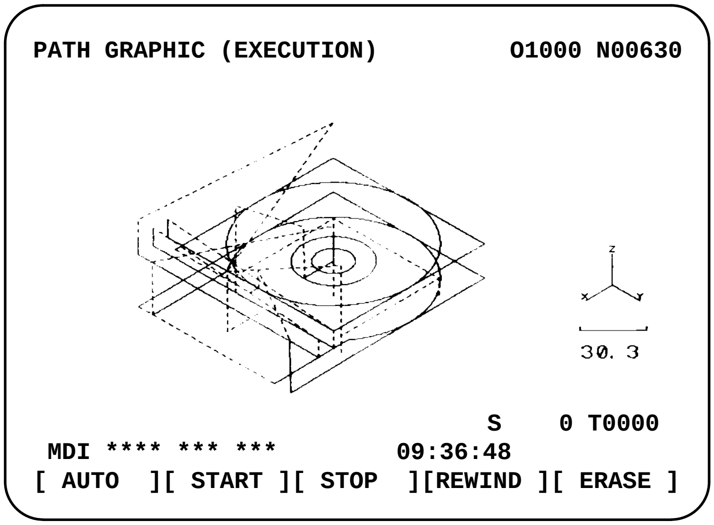
Enlarged-view drawing



```
PATH GRAPHIC (EXECUTION)        O1000 N00630

                                S    0 T0000
 MDI **** *** ***          09:36:48
```

- In addition to two-dimensional drawings, isometric projection drawings and biplanar drawings can be created.

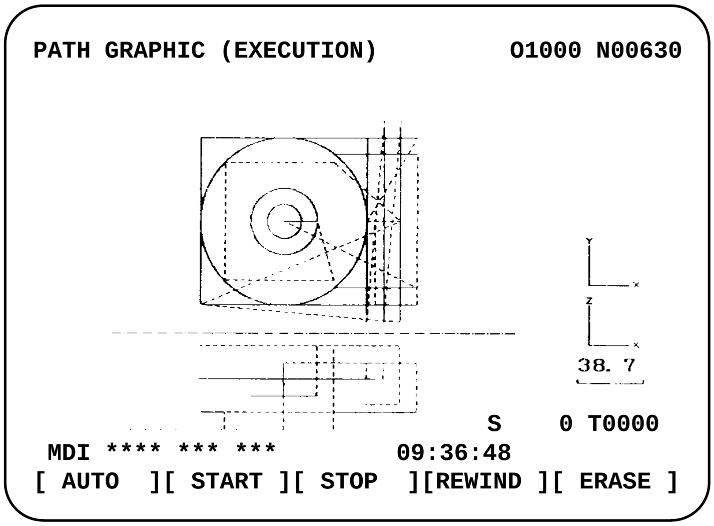
Isometric projection drawing



```
PATH GRAPHIC (EXECUTION)        O1000 N00630

                                 S    0 T0000
 MDI **** *** ***        09:36:48
[ AUTO  ][ START ][ STOP  ][REWIND ][ ERASE ]
```

- Because the current position of a tool is marked on the drawn tool path, the progress of machining can be monitored accurately.

Biplanar drawing

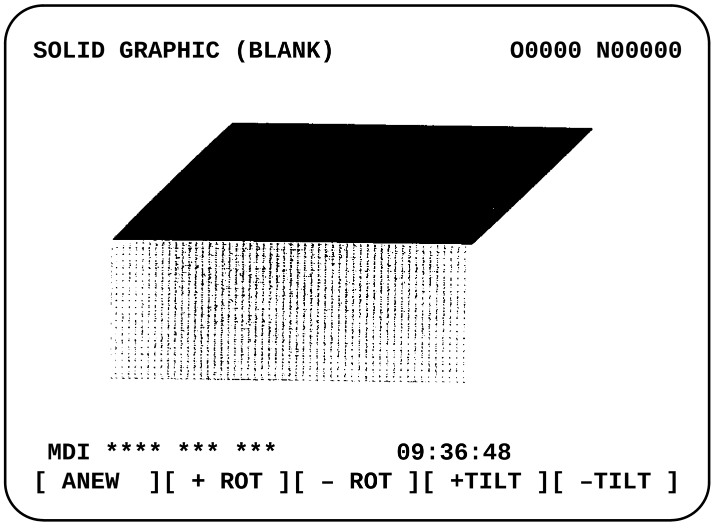


```
PATH GRAPHIC (EXECUTION)        O1000 N00630

                                 S    0 T0000
 MDI **** *** ***        09:36:48
[ AUTO  ][ START ][ STOP  ][REWIND ][ ERASE ]
```

- **Machining profile drawing mode**

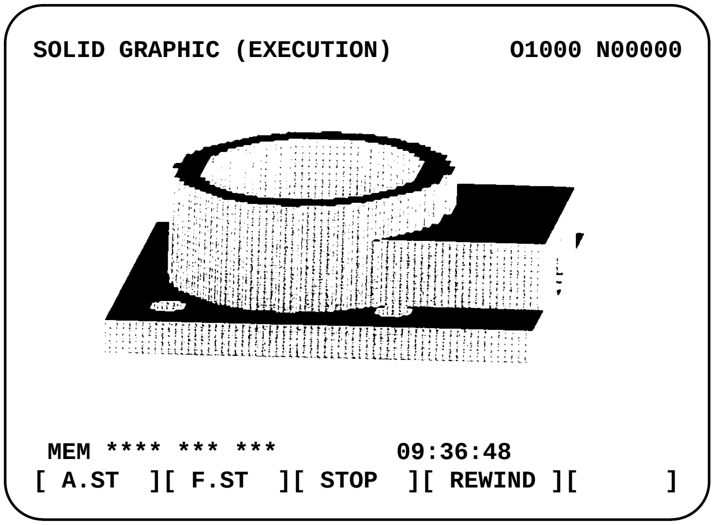
- The profile of a workpiece that changes as the tool moves can be simulated and drawn three-dimensionally, making it easier to check programs visually.

Blank figure



```
SOLID GRAPHIC (BLANK)            O0000 N00000

 MDI **** *** ***        09:36:48
[ ANEW  ][ + ROT ][ – ROT ][ +TILT ][ –TILT ]
```

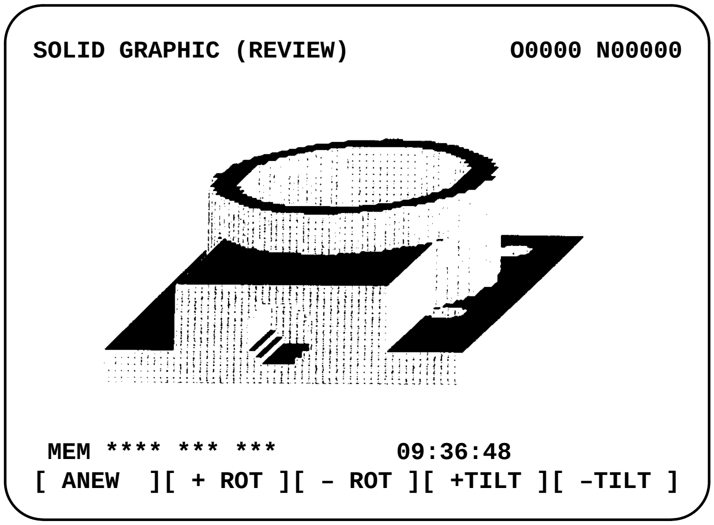
Final figure



```
SOLID GRAPHIC (EXECUTION)        O1000 N00000

 MEM **** *** ***        09:36:48
[ A.ST  ][ F.ST  ][ STOP  ][ REWIND ][      ]
```

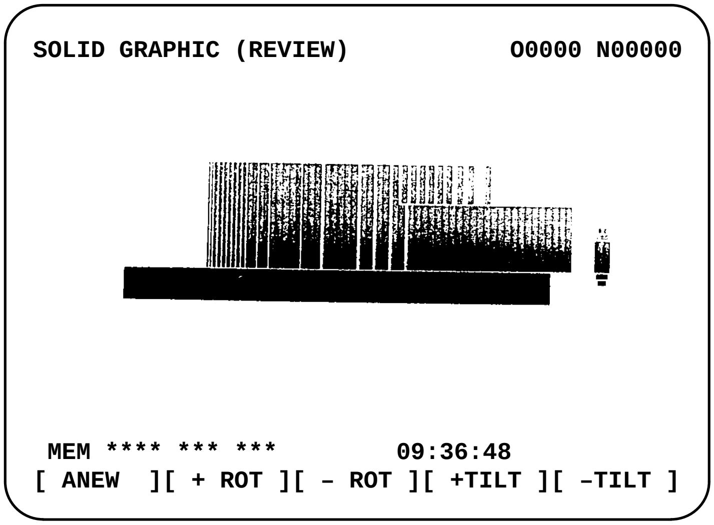
- The coordinate axes and projection angles can be changed at the operator’s option.

Modification of a coordinate axis (inclination)



```
SOLID GRAPHIC (REVIEW)          O0000 N00000

 MEM **** *** ***        09:36:48
[ ANEW  ][ + ROT ][ – ROT ][ +TILT ][ –TILT ]
```

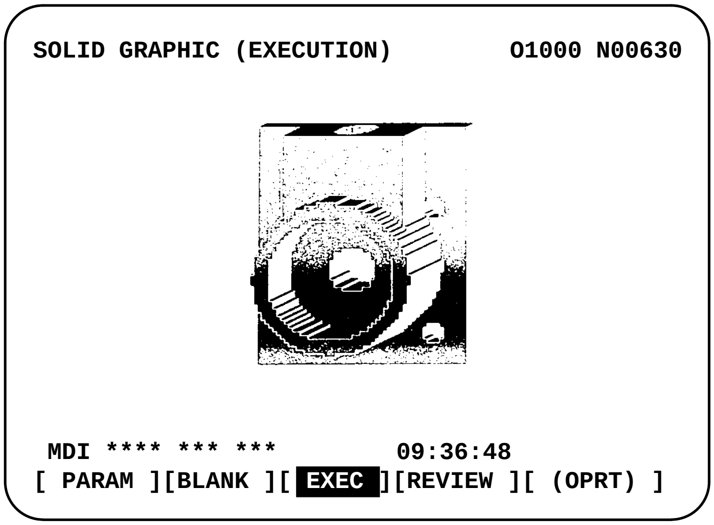
Modification of a coordinate axis (inclination)



```
SOLID GRAPHIC (REVIEW)          O0000 N00000

 MEM **** *** ***        09:36:48
[ ANEW  ][ + ROT ][ – ROT ][ +TILT ][ –TILT ]
```

- The tool can be mounted parallel to any of the X, Y, and Z axes.

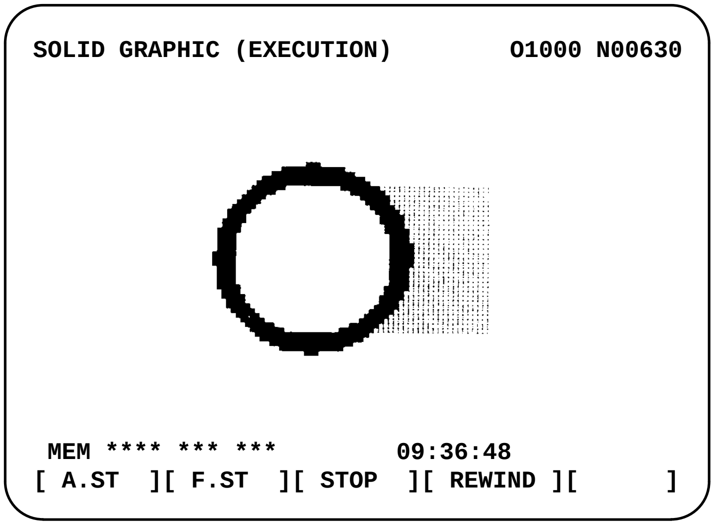
Modification of a coordinate axis (vertical axis)



```
SOLID GRAPHIC (EXECUTION)       O1000 N00630

 MDI **** *** ***        09:36:48
```

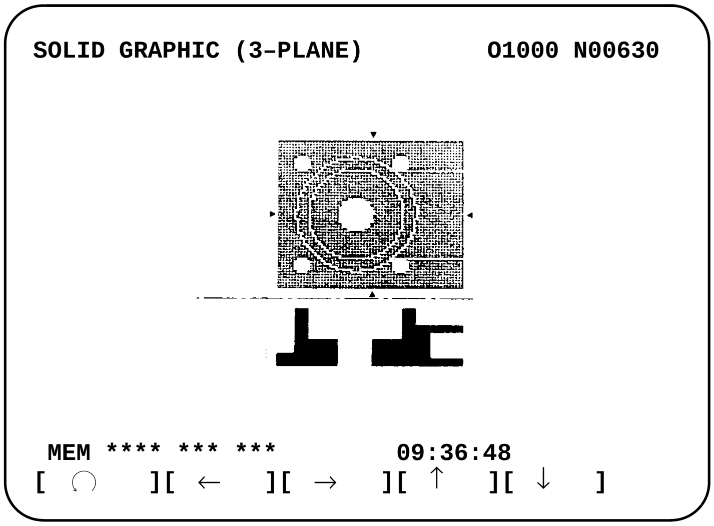
- In addition to three–dimensional drawings, two–dimensional drawings and tri–planar drawings can be created.

Two-dimensional drawing



```
SOLID GRAPHIC (EXECUTION)       O1000 N00630

 MEM **** *** ***        09:36:48
```

Tri-planar drawing



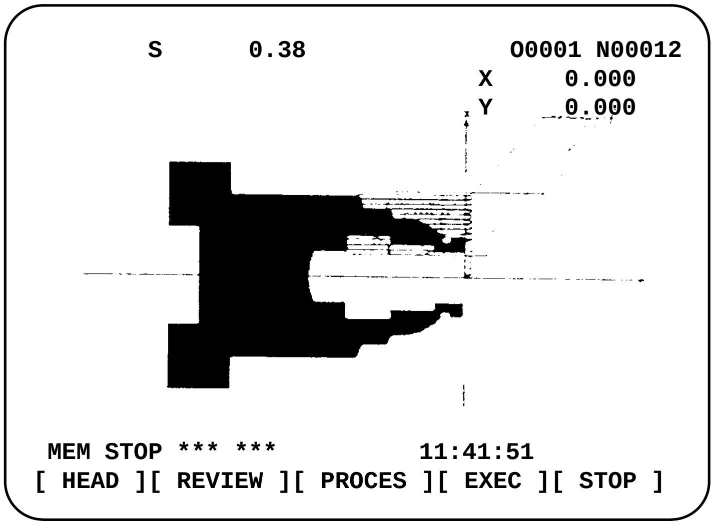
```
SOLID GRAPHIC (3–PLANE)        O1000 N00630

 MEM **** *** ***        09:36:48
[ ∩   ][ ←   ][ →   ][ ↑   ][ ↓   ]
```

Dynamic graphic display function (for T series)

The following two display modes are available:

- **Tool path drawing mode**

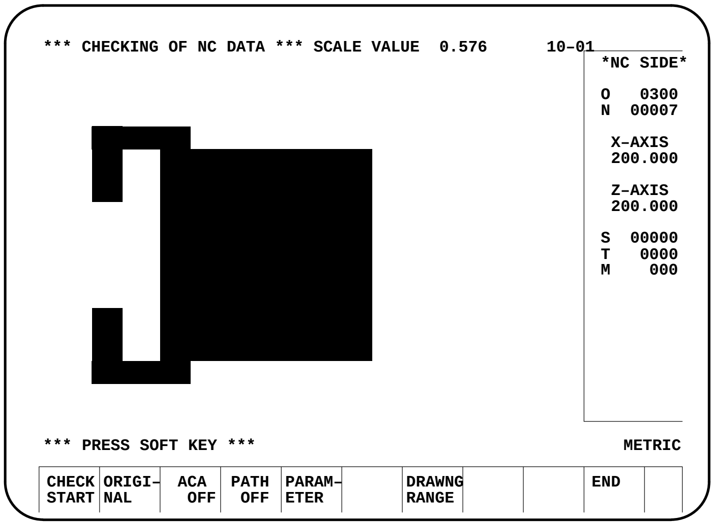
Movement of the tool tip is drawn with fine lines.



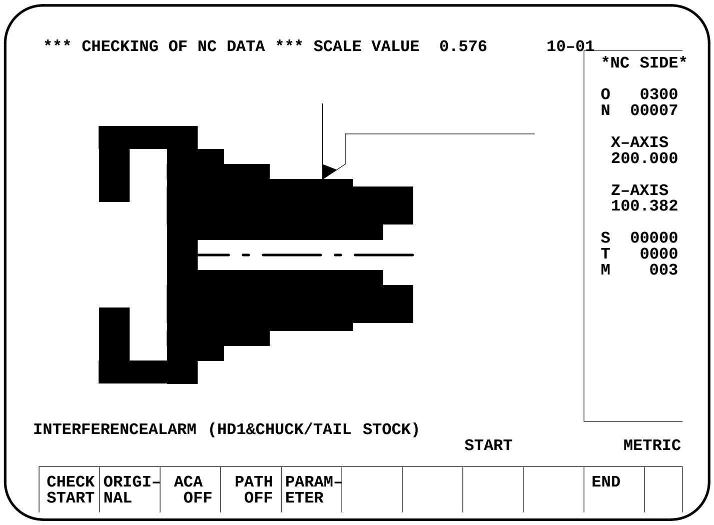
```
        S      0.38               O0001 N00012
                                X     0.000
                                Y     0.000

 MEM STOP *** ***          11:41:51
[ HEAD ][ REVIEW ][ PROCES ][ EXEC ][ STOP ]
```

- **Animated drawing mode**

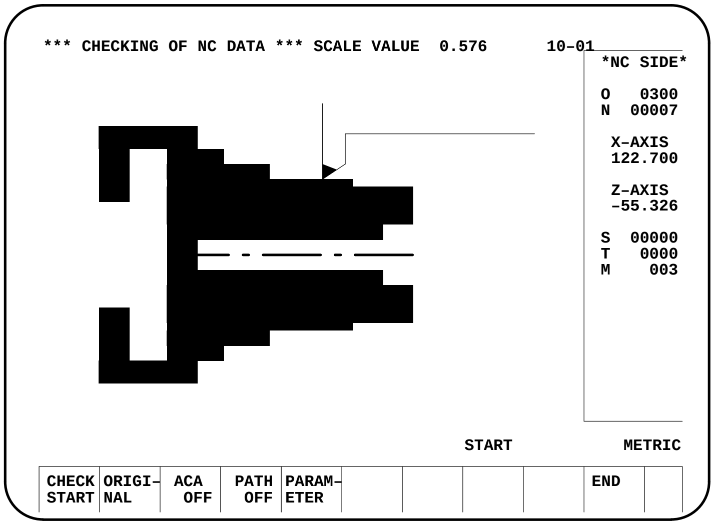
Accurate figures of the material, chuck, and tailstock are displayed on the screen. An animated simulation illustrates how the material will be cut by the tool.



```
*** CHECKING OF NC DATA *** SCALE VALUE  0.576      10–01
                                                     *NC SIDE*
                                                     O   0300
                                                     N  00007
                                                      X–AXIS
                                                      200.000
                                                      Z–AXIS
                                                      200.000
                                                     S  00000
                                                     T   0000
                                                     M    000
*** PRESS SOFT KEY ***                                  METRIC
```



```
*** CHECKING OF NC DATA *** SCALE VALUE  0.576      10–01
                                                     *NC SIDE*
                                                     O   0300
                                                     N  00007
                                                      X–AXIS
                                                      200.000
                                                      Z–AXIS
                                                      100.382
                                                     S  00000
                                                     T   0000
                                                     M    003
INTERFERENCEALARM (HD1&CHUCK/TAIL STOCK)
                                       START            METRIC
```



```
*** CHECKING OF NC DATA *** SCALE VALUE  0.576      10–01
                                                     *NC SIDE*
                                                     O   0300
                                                     N  00007
                                                      X–AXIS
                                                      122.700
                                                      Z–AXIS
                                                      –55.326
                                                     S  00000
                                                     T   0000
                                                     M    003
                                       START            METRIC
```

27.7.3 Background Drawing

M series

The background drawing function enables the drawing of a figure for one program while machining a workpiece under the control of another program.

Explanations

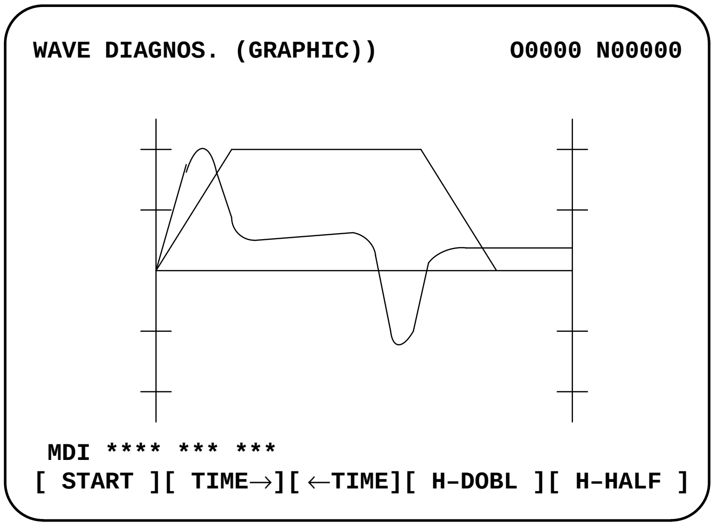
- **Program selection**

Immediately after entering background drawing mode with operation of MDI key, the program which was selected previously remains selected. Any program can be selected for background drawing, by using the background drawing screen.

Parameter setting and drawing method are same as synamic graphic display.

27.8
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.



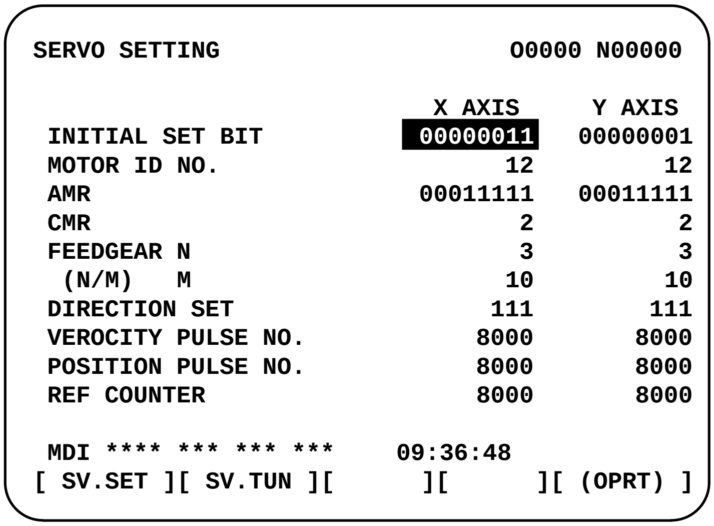
```
WAVE DIAGNOS. (GRAPHIC))         O0000 N00000

 MDI **** *** ***
[ START ][ TIME→][ ←TIME][ H–DOBL ][ H–HALF ]
```

On this screen, the sampling period (6 to 32767 ms) and drawing start conditions can be specified.

27.9
SCREENS FOR SERVO DATA AND SPINDLE DATA

27.9.1
Servo Setting Screen

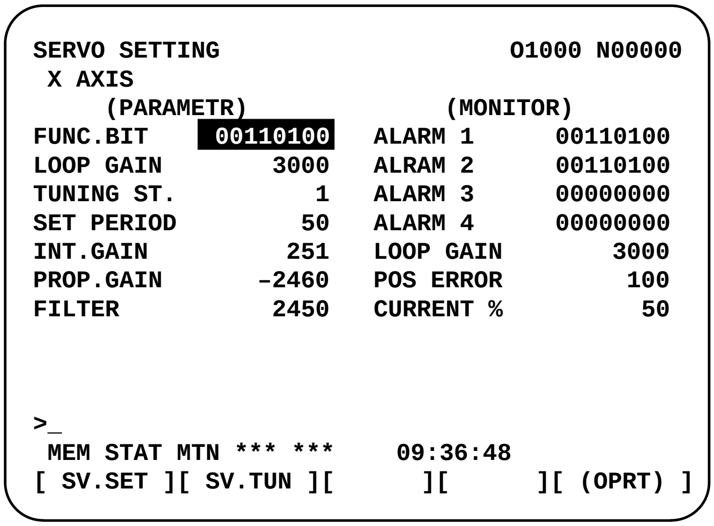
On the servo setting screen, parameters required for standard initialization of the servo motor are listed. The parameters can also be set.



```
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) ]
```

27.9.2
Servo Adjustment Screen

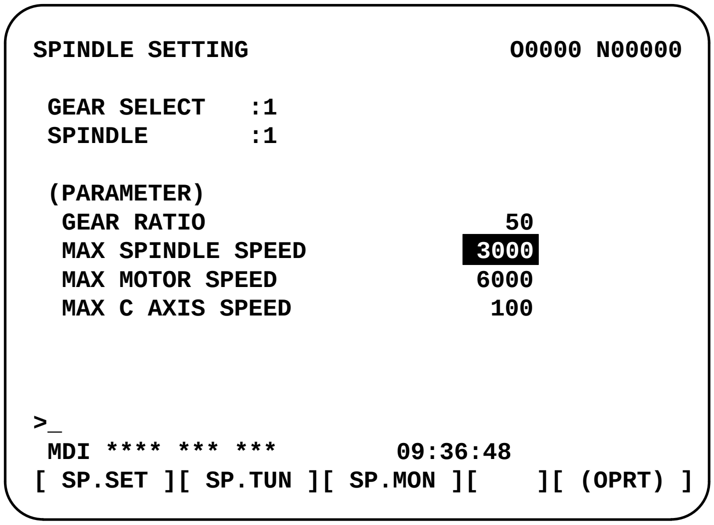
On the servo adjustment screen, parameters required for basic adjustment of the servo motor and statuses being monitored are listed for each axis.



```
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) ]
```

27.9.3
Spindle Setting Screen

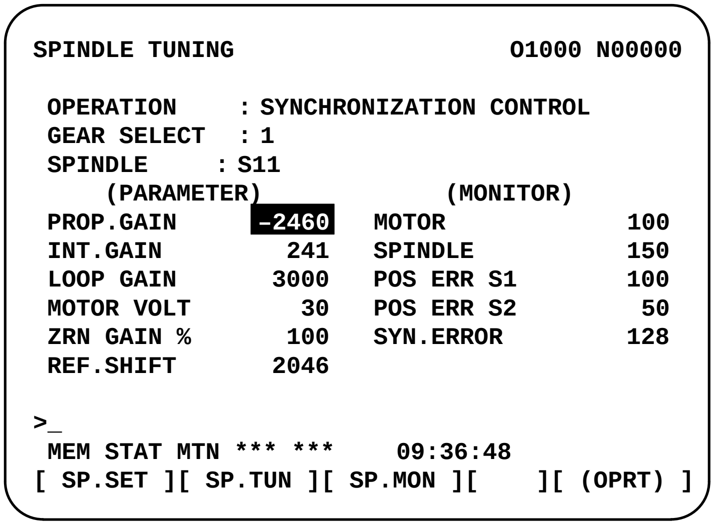
On the spindle setting screen, parameters required for standard initialization of the serial spindle are listed. The parameters can also be set. This screen is only for the main spindle connected to the first amplifier.



```
SPINDLE SETTING                    O0000 N00000

 GEAR SELECT   :1
 SPINDLE       :1

 (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) ]
```

27.9.4
Spindle Adjustment Screen

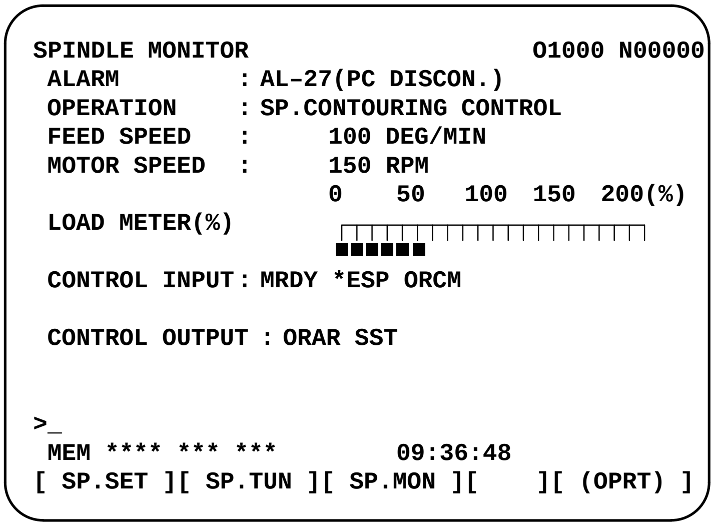
On the spindle adjustment screen, parameters required for basic adjustment of the serial spindle and statuses being monitored are listed. The screen is only for the main spindle connected to the first amplifier.



```
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   POS ERR S2        50
 ZRN GAIN %      100   SYN.ERROR        128
 REF.SHIFT      2046

>_
 MEM STAT MTN *** ***    09:36:48
[ SP.SET ][ SP.TUN ][ SP.MON ][    ][ (OPRT) ]
```

27.9.5 Spindle Monitor Screen

On the spindle monitor screen, various data items related to the spindle are listed. This screen is only for the main spindle of the first amplifier.



```
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) ]
```

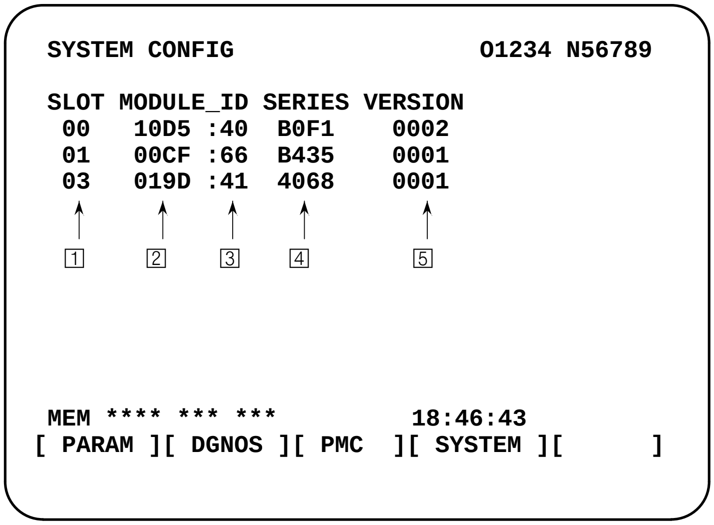
27.10 SYSTEM CONFIGURATION DISPLAY FUNCTION

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   10D5 :40  B0F1    0002
  01   00CF :66  B435    0001
  03   019D :41  4068    0001







 MEM **** *** ***          18:46:43
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][      ]
```

1 **Slot No.**

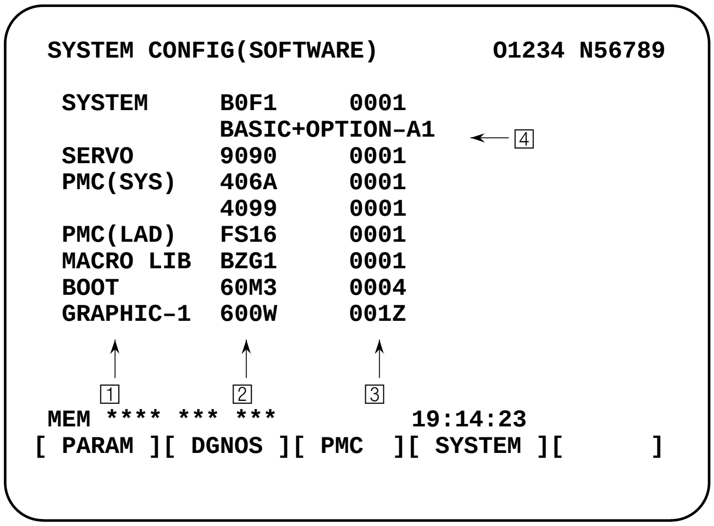
2 **Module ID**

3 **Software ID**

4 **Software series**

5 **Software edition**

Software information



```
 SYSTEM CONFIG(SOFTWARE)         O1234 N56789

  SYSTEM     B0F1     0001
             BASIC+OPTION–A1
  SERVO      9090     0001
  PMC(SYS)   406A     0001
             4099     0001
  PMC(LAD)   FS16     0001
  MACRO LIB  BZG1     0001
  BOOT       60M3     0004
  GRAPHIC–1  600W     001Z


 MEM **** *** ***          19:14:23
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][      ]
```

1 **Software type**

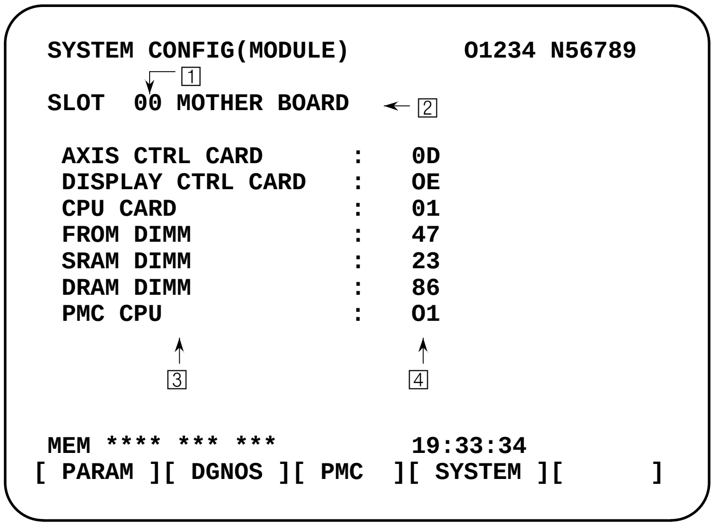
2 **Software series**

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

 SLOT  00 MOTHER BOARD

  AXIS CTRL CARD      :   0D
  DISPLAY CTRL CARD   :   OE
  CPU CARD            :   01
  FROM DIMM           :   47
  SRAM DIMM           :   23
  DRAM DIMM           :   86
  PMC CPU             :   O1



 MEM **** *** ***         19:33:34
[ PARAM ][ DGNOS ][ PMC  ][ SYSTEM ][      ]
```

$\boxed{1}$ **Slot number (The slot number corresponds to the number displayed on the slot information screen.)**

$\boxed{2}$ $\boxed{1}$ **Name of the PC board inserted in the slot**

$\boxed{3}$ $\boxed{2}$ **Hardware (modules) mounted on the PC board**

$\boxed{4}$ $\boxed{3}$ **Types of hardware (modules), mounted/not mounted**

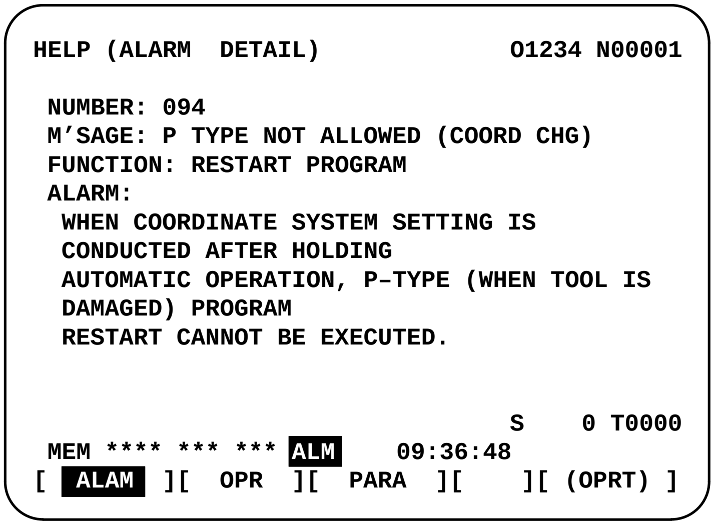
27.11 HELP FUNCTION

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:

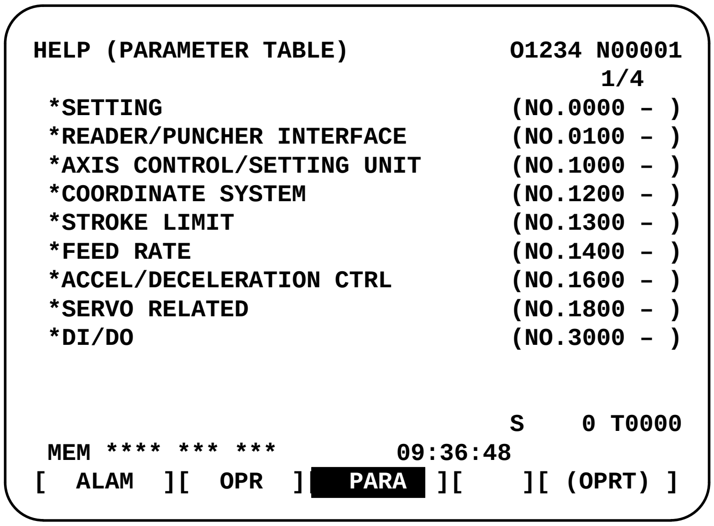
- On the alarm detail screen, detailed information on the alarm currently activated is displayed. Using this information, the operator can identify the cause of the alarm and what action to take. Any alarm information can be displayed on this screen.
- On the operation instruction screen, when the operator 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 94) is activated.**



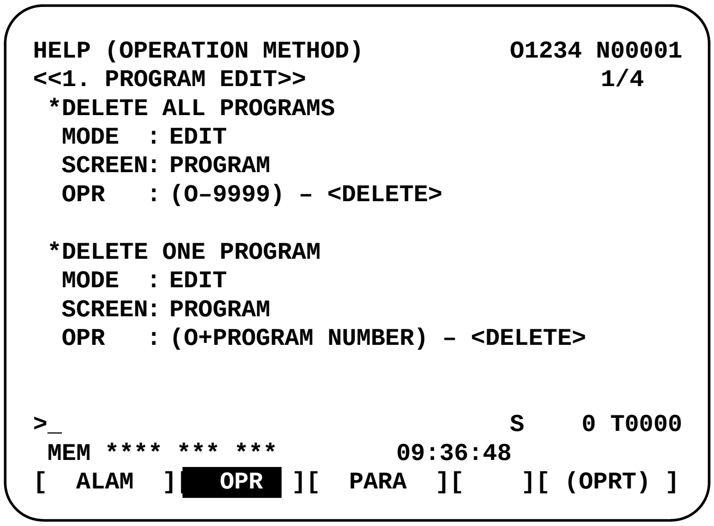
```
HELP (ALARM  DETAIL)             O1234 N00001

 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) ]
```

27.12
DATA PROTECTION KEY

A data protection key can be installed on the machine side for protection of various NC data. The following three 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.

27.13
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 : 97/09/25
  09:27:55

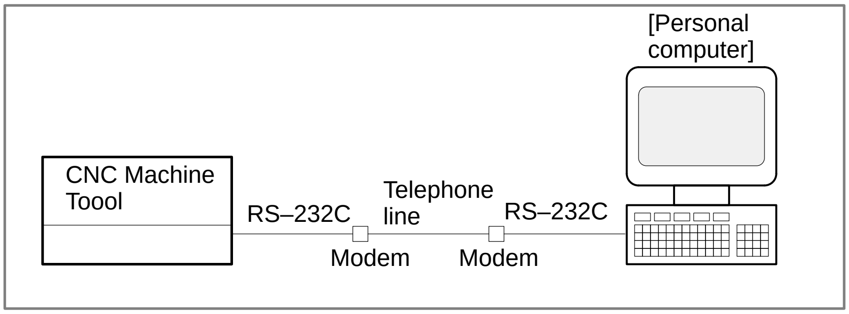
The history data can be output to an input/output device, connected via the reader/punch interface. Previously output history data can be input from an input/output device.

27.14
MACHINING TIME STAMP FUNCTION

When a machining program is executed, the machining time of the main program is displayed on the program machining time display screen. The machining times of up to ten main programs are displayed in hour/minutes/seconds. When more than ten programs are executed, data for the oldest programs is discarded.

27.15
REMOTE DIAGNOSIS

The remote diagnosis function allows CNC status monitoring and modification to CNC data to be performed remotely by menu–based operation. The remote diagnosis function, operating under MS–DOS, is installed on a standard personal computer, connected as a service terminal to the CNC via the RS–232C interface, over a telephone line, and so on.



The remote diagnosis terminal software is sold separately.
The remote diagnosis function provides the following capabilities :

- **CNC programs**
  - ❑ Computer → CNC
    - CNC command data for verification
    - Searching for a specified program
    - Part program
    - Deleting a specified program
    - Deleting all programs
  - ❑ CNC → computer
    - Part program
    - Displaying a program directory
    - Program number of a program being executed
    - Sequence number of a sequence being executed
  - ❑ Computer → CNC
    - Parameter
    - Pitch error data
    - Tool offset value
    - Custom macro variable
    - Selecting a display screen
    - Memory contents
    - PMC data
    - Displaying a specified message
    - All parameters

❑ CNC → computer
  - Alarm information
  - Machine position
  - Absolute position
  - Skip position
  - Servo delay
  - Acceleration/deceleration delay
  - Diagnosis
  - Parameter
  - Tool life management data
  - Display screen status
  - Modal information
  - Pitch error data
  - Tool offset value
  - Custom macro variable
  - Memory contents
  - Ladder program
  - Actual feedrate
  - Status
  - A/D conversion
  - PMC data
  - Screen character data
  - Printed circuit board information
  - Ladder title
  - Series and edition of PMC/ladder
  - All parameters

❑ File function selection
  - Listing files
  - Referring a file
  - Deleting a file
  - Copying a file
  - Renaming a file
  - Linking a file
  - Changing the current directory
  - Creating a directory
  - Deleting a directory

NOTE

An arrow "→" indicates the direction of data flow.

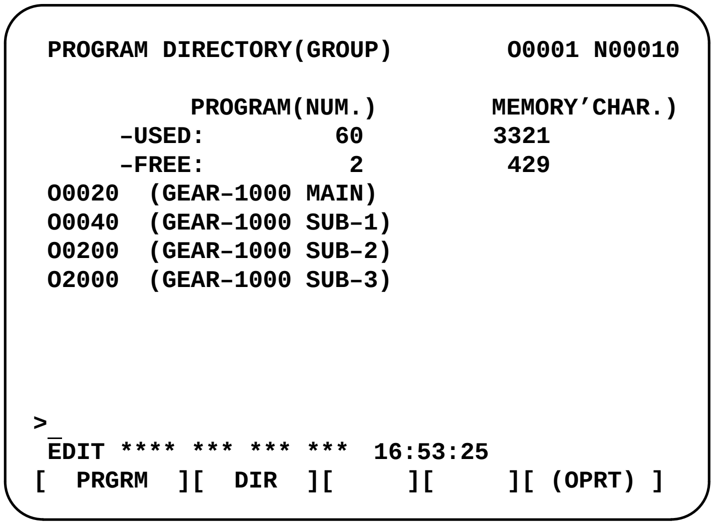
27.16
DIRECTORY DISPLAY AND PUNCH FOR A SPECIFIED GROUP

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*”



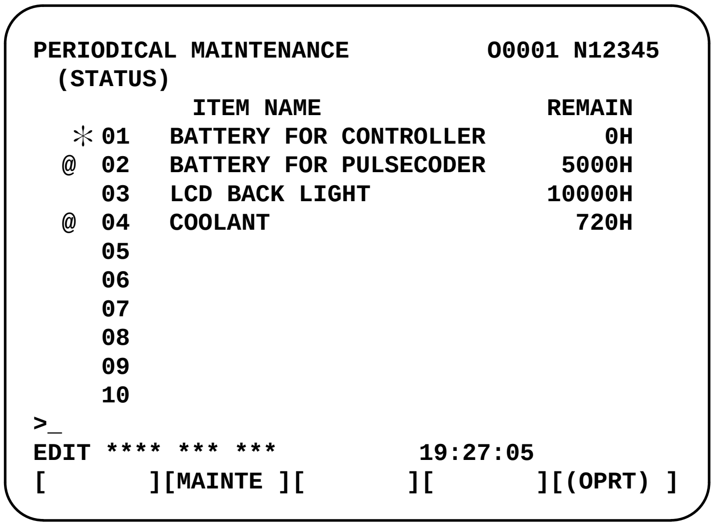
```
 PROGRAM DIRECTORY(GROUP)        O0001 N00010

           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) ]
```

27.17
CLEARING THE SCREEN

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.

27.18 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) ]
```

27.19 TOUCH PAD

A pen input device/touch pad, manufactured by Fujitsu Limited, is used on the LCD display as follows:

(1) The soft keys below the 10.4–inch color LCD/MDI panel (F0 to F9, FR, and FL) are replaced by the soft keys on the touch pad.

(2) The cursor displayed on the 10.4–inch color LCD is controlled from the touch pad.

(3) A touch–pad–type software machine operator's panel, realized by C executor, can be used.

(4) A touch–pad–type calculator, realized by C executor, can be used.

(5) A C executor application program can be created by using the touch pad.

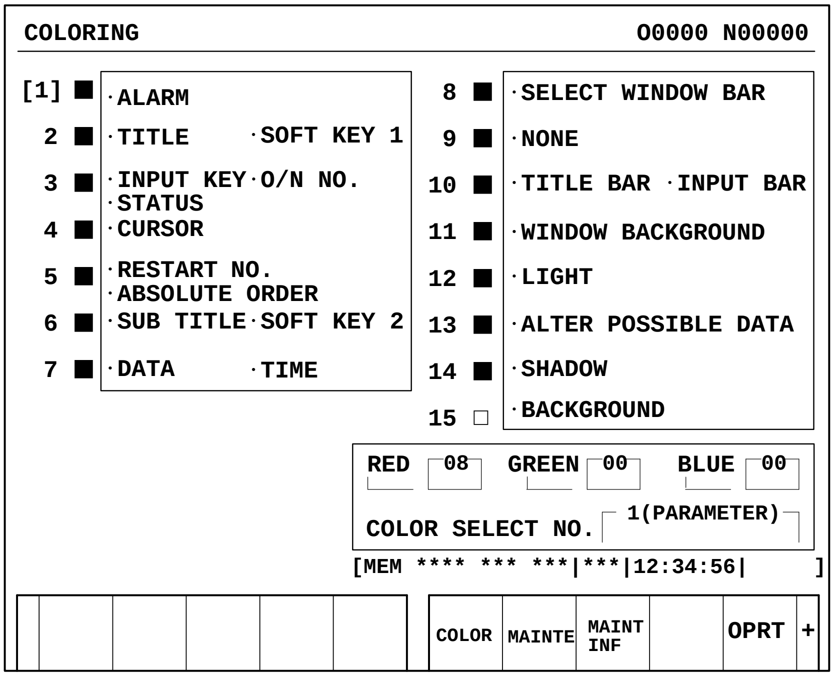
27.20 MAINTENANCE INFORMATION SCREEN

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.)

27.21 COLOR SETTING SCREEN

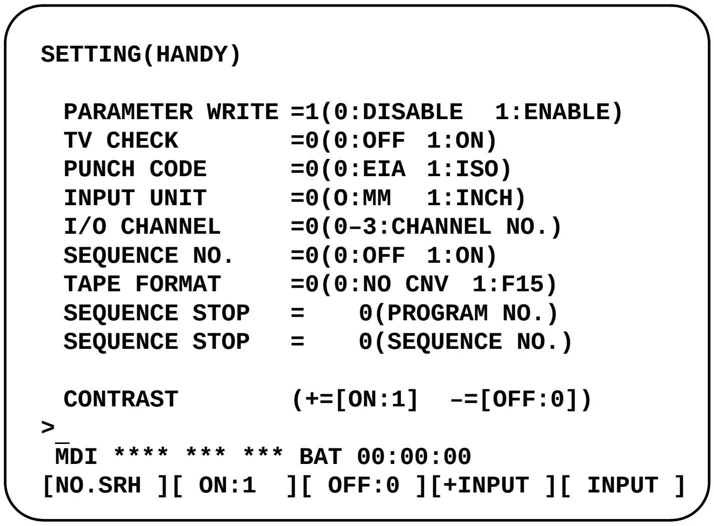
When the VGA graphic control function is supported, the VGA screen colors can be set on the color setting screen.



```
COLORING                                        O0000 N00000

[1] ■ ·ALARM                           8 ■ ·SELECT WINDOW BAR
 2  ■ ·TITLE        ·SOFT KEY 1        9 ■ ·NONE
 3  ■ ·INPUT KEY·O/N NO.              10 ■ ·TITLE BAR ·INPUT BAR
      ·STATUS
 4  ■ ·CURSOR                         11 ■ ·WINDOW BACKGROUND
 5  ■ ·RESTART NO.                    12 ■ ·LIGHT
      ·ABSOLUTE ORDER
 6  ■ ·SUB TITLE·SOFT KEY 2           13 ■ ·ALTER POSSIBLE DATA
 7  ■ ·DATA       ·TIME               14 ■ ·SHADOW
                                      15 □ ·BACKGROUND

                                RED  08   GREEN  00   BLUE  00

                                COLOR SELECT NO.  1(PARAMETER)
                              [MEM **** *** ***|***|12:34:56|    ]
```

27.22 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 ]
```

28

PART PROGRAM STORAGE AND EDITING

28.1 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.

28.2 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.

28.3
EXPANDED 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.

28.4
NUMBER OF REGISTERED PROGRAMS

Number of registered programs can be selected from the following:
63, 125, 200, 400, or 1000.

28.5
PART PROGRAM STORAGE LENGTH

The following part program storage length can be selected:
10, 20, 40, 80, 160, 320, 640, 1280, 2560, or 5120 m.

28.6
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.

28.7
EXTERNAL CONTROL OF I/O DEVICE

Part program registration and punch can be commanded externally.

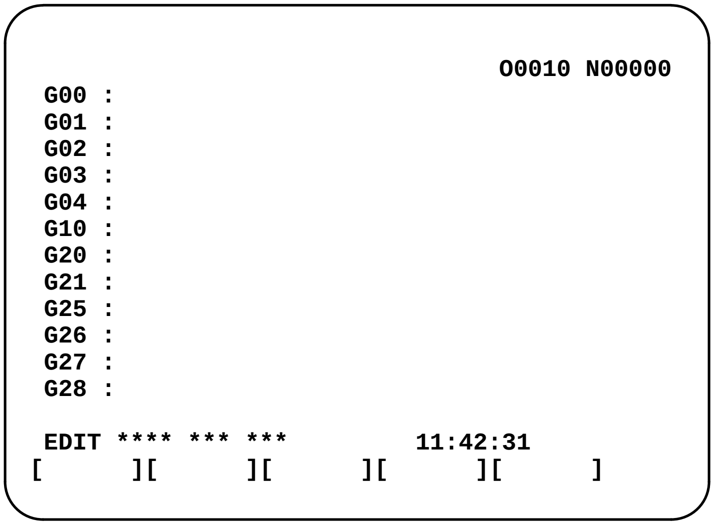
- **Program registration**
  A part program can be registered in memory through the connected input device for background editing using the external read start signal.
- **Program punch**
  A part program can be punched through the connected output device for background editing using the external punch start signal.

28.8 CONVERSATIONAL PROGRAMMING OF FIGURES (ONLY AT 1–PATH CONTROL)

The following two screens can be displayed with graphic data for guidance in programming in the CNC format:

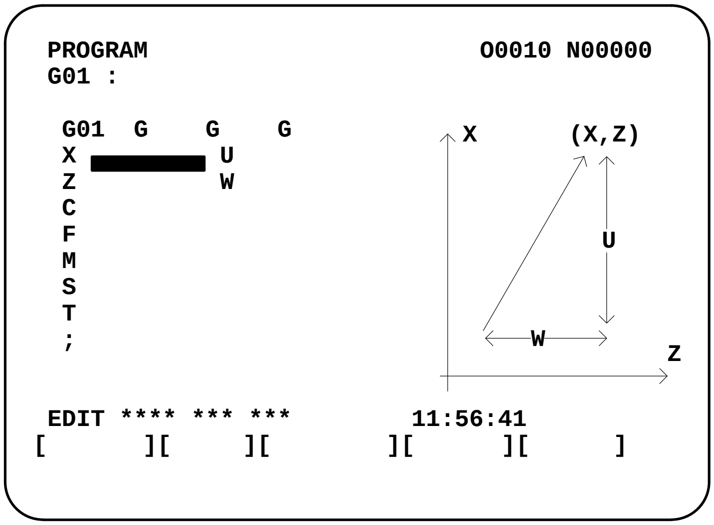
- G code list
- Standard format of a G-code block

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
[      ][     ][       ][      ][      ]
```

28.9 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.

29

DIAGNOSIS FUNCTIONS

29.1
SELF DIAGNOSIS FUNCTIONS

The NC 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

Input/output signals from PMC to CNC, or vice versa, and inner status of the NC can be displayed.

30 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

30.1 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

30.2
INPUT/OUTPUT DEVICES

The following Input/Output devices are prepared, which are connectable to the reader/puncher interface.

30.2.1
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.

30.2.2
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.

30.2.3
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.

30.3
EXTERNAL PROGRAM INPUT

By using the external program input start signal, a program can be loaded from an input unit into CNC memory.

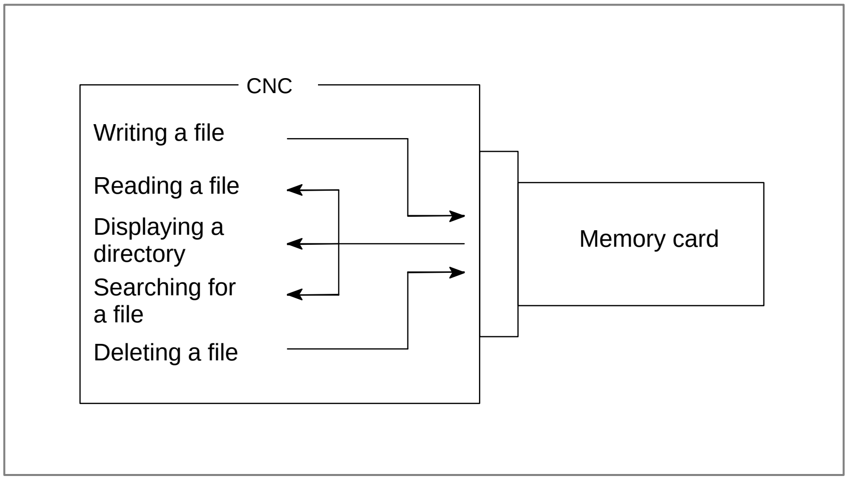
When an input unit such as the FANUC Handy File or FANUC Floppy Cassette is being used, a file can be searched for using the workpiece number search signals, after which the program can be loaded into CNC memory.

30.4
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.



30.5 DNC1 CONTROL

DNC1 is a poprietary communication network allowing information exchange between the cell controller and CNC machine tools.

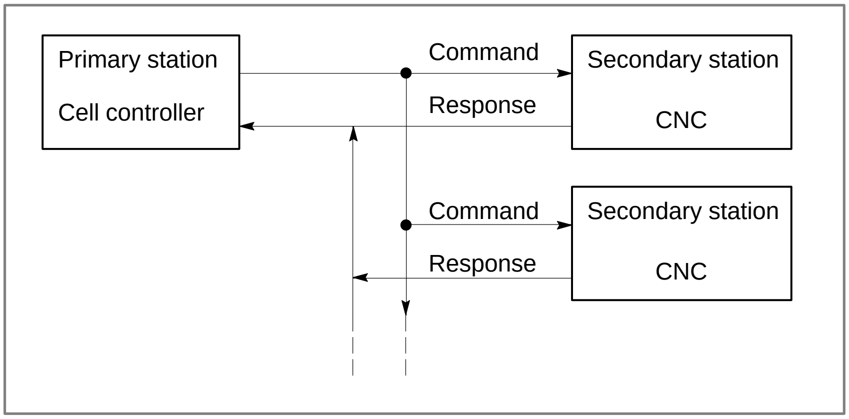
DNC1 is classified into two mode, Mode–1 and Mode–2, by the connection models.

Explanations

- **Mode–1**

In the Mode–1, the cell controller plays as a primary station and controls multiple CNC machine tools in the multi–point–connection.

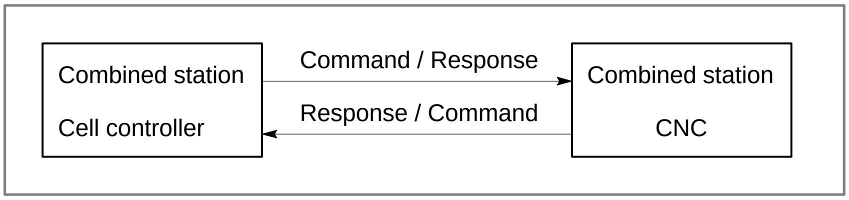
This mode is usually used to establish small scale FMS.



- **Mode–2**

In the Mode–2, the cell controller and CNC play as a combined station in the point–to–point connection.

This mode is usually used for drip feeding of continuous small blocks in the DNC operation such as mold machining.



NOTE

The combined station has combined functions of primary station and secondary station.

- **Primary station**
  This station controls data link and has responsibility of control and recovery to the other station on the data link. This station corresponds to the cell controller in the multi–point connection.
- **Secondary station**
  This station executes data link control according to the command of the primary station. This station corresponds to the CNC in the multi–point connection.
- **Combined station**
  This station executes peer–to–peer communication concerning to the data link control.

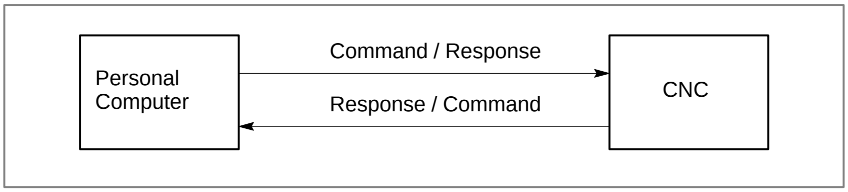
30.6 DNC2 CONTROL (ONLY AT 1–PATH CONTROL)

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.



30.7
DATA SERVER

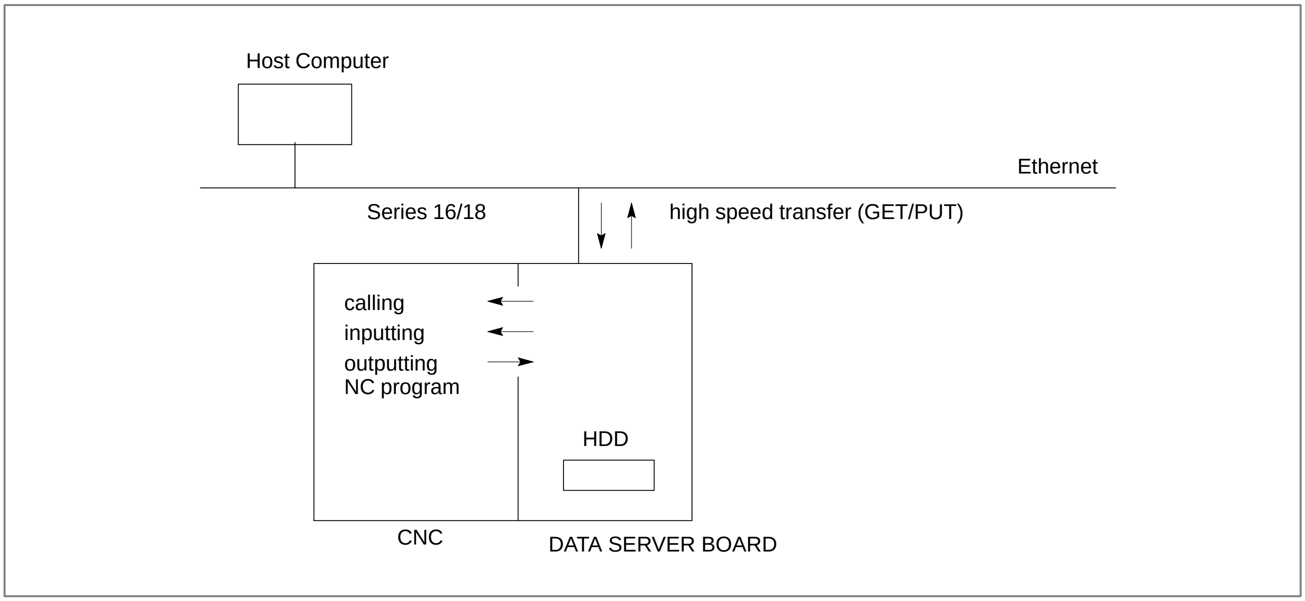
The dara server has the following features:

(1) Drive high–speed machining operation by calling the subprogram from a built–in hard disk on the DATA SERVER BOARD(described as “HDD” below).

(2) Input a NC program in the Host Computer into the HDD by using FTP. Output a NC program in the HDD into the Host Computer by using FTP.

(3) Input a NC program in the HDD into the memory of the CNC. Output a NC program in the memory of the CNC into the HDD.

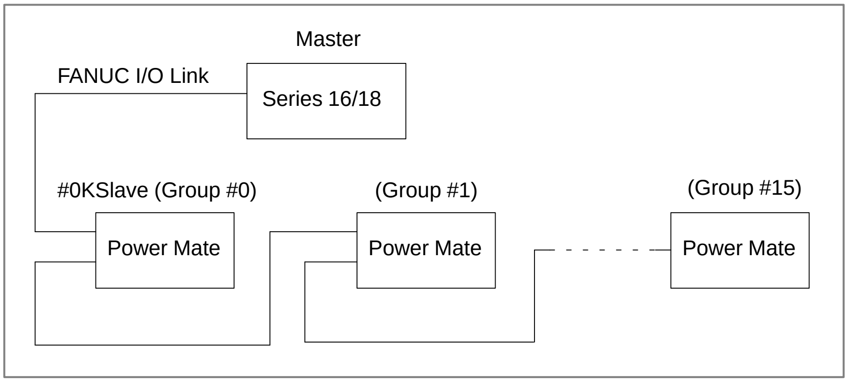
(4) Delete NC program and display the table of NC programs in the HDD.



30.8

DATA INPUT/OUTPUT FUNCTION BASED ON THE I/O LINK AND DATA INPUT/OUTPUT FUNCTION B BASED ON THE I/O LINK

Power Mate programs, parameters, macro variables, and diagnostic (PMC) data are input/output using FANUC I/O Link.
With FANUC I/O Link, slaves in groups 0 to 15 can be connected, enabling data input/output to and from a maximum of 16 Power Mates.
The ordinary data input/output function based on I/O Link can only be executed in the foreground. When data input/output function B based on I/O Link is used, the external I/O device control function is associated with I/O Link so that an input/output group number and program number can be specified from the PMC. The external I/O device control function operates in the background. Therefore, when no other background operation is being performed, data can be input/output, regardless of the NC mode and the currently selected screen.



The programs, parameters, macro variables, and diagnostic (PMC) data of a slave Power Mate are stored in tape format within the part program storage length; these data items are stored as master program data in a master program memory area.

Data input/output can be performed between the master and a slave of a selected group. When the ordinary data input/output function based on I/O Link is used, a group is selected by means of parameter setting. When data input/output function B based on I/O Link is used, a group is selected by issuing the DI signal. Data input/output cannot be performed between the master and more than one group at a time.

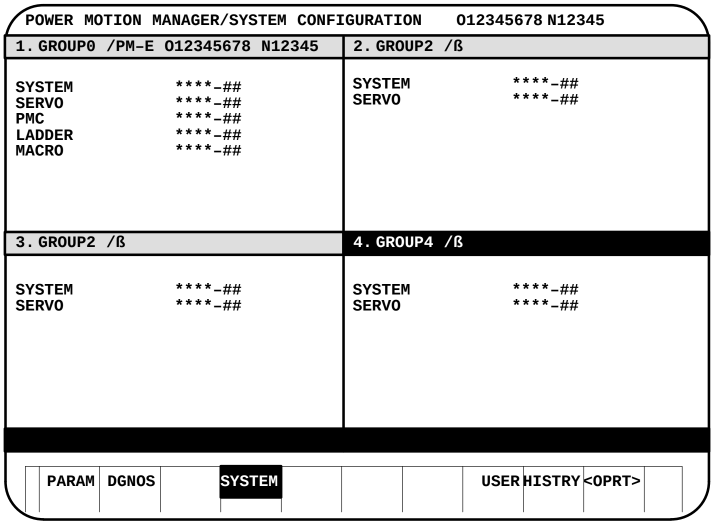
30.9

POWER MOTION MANAGER

When the power motion series is used as an additional (slave) axis of the CNC, the power motion manager enables the display and setting of data from the CNC. Up to eight slave units can be connected.

The power motion manager supports the following functions:

1) Current position display (absolute/machine coordinate)
2) Parameter display and setting
3) Diagnosis
4) System configuration screen
5) Alarm



```
POWER MOTION MANAGER/SYSTEM CONFIGURATION    O12345678 N12345
1. GROUP0 /PM–E O12345678 N12345    2. GROUP2 /ß

SYSTEM          ****–##             SYSTEM          ****–##
SERVO           ****–##             SERVO           ****–##
PMC             ****–##
LADDER          ****–##
MACRO           ****–##

3. GROUP2 /ß                        4. GROUP4 /ß

SYSTEM          ****–##             SYSTEM          ****–##
SERVO           ****–##             SERVO           ****–##
```

The sample screen shows the data for four units, displayed on a 12 soft key type device. The same data can also be displayed on a 7 soft key type device.

31 SAFETY FUNCTIONS

31.1 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.

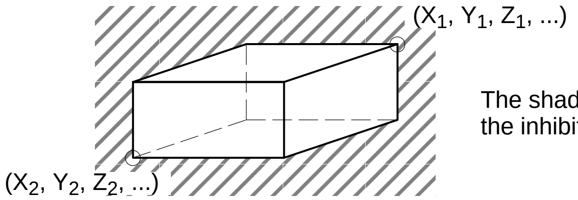
31.2 OVERTRAVEL FUNCTIONS

31.2.1 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.

31.2.2 Stored Stroke Check 1

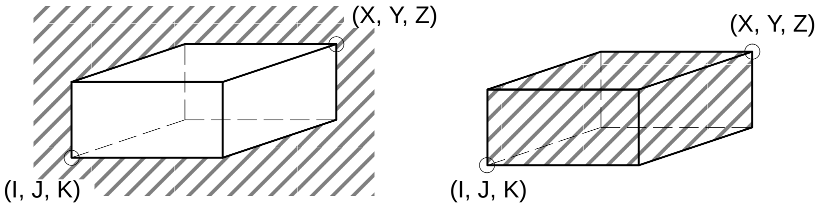
The movable section of the machine is parameter set in machine coordinates value. If the machine moves beyond the preset range, it decelerates to a stop and alarm is displayed. (This function is valid after manual reference point return at power on.)
This function can be used instead of hardware overtravel 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.



The shaded part is the inhibition area.

31.2.3 **M series** Stored Stroke Check 2 (G22, G23)

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.



Inhibition area outside **Inhibition area inside**

Format

G22 X_ Y_ Z_ I_ J_ K_ ;

On/off of stored stroke check 2 is commanded by program as follows:

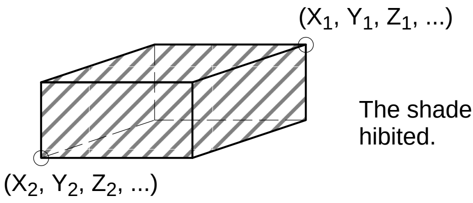
G22 : Stored stroke check function on

G23 : Stored stroke check function off

31.2.4 Stored Stroke Checks 3

M series

The space within the range specified with parameters is inhibited.



The shaded space is inhibited.

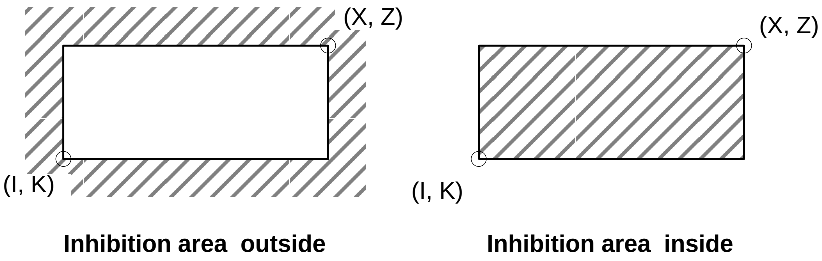
31.2.5 Stored Stroke Checks 2 and 3

T series

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 2 (G22, G23)

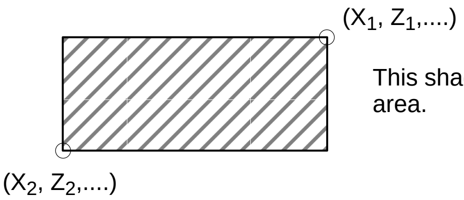
The designation of the forbidden area can be specified by parameters or program.
The forbidden area can be changed for each workpiece. Selection between inside or outside as the forbidden area is made by parameters.



Inhibition area outside **Inhibition area inside**

Stored stroke check 3

Inside the area specified by parameters is the forbidden area.



This shaded area 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

31.2.6

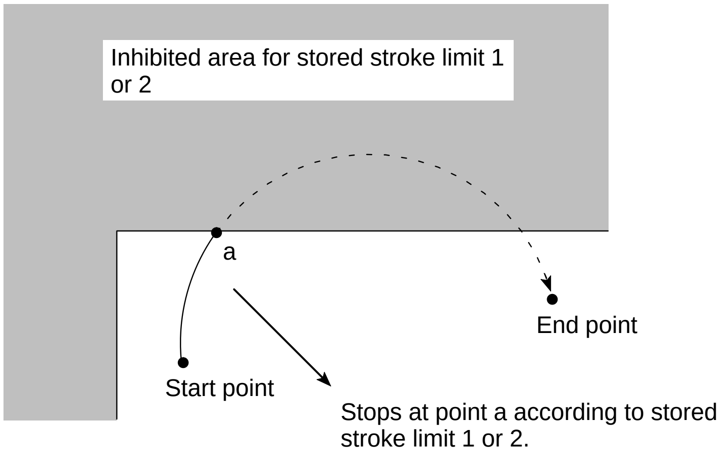
Stroke Limit Check Before Movement

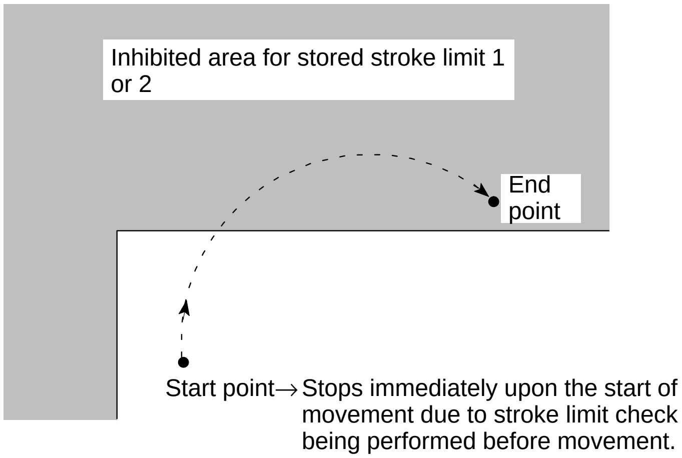
This function calculates the movement end point at the start of movement in a block, during automatic operation, based on the current machine position and the specified amount of travel, to check whether the end point falls within the inhibited area for stored stroke limit 1, 2, or 3. If the end point falls within an inhibited area, movement for that block is stopped immediately upon the start of movement and an alarm is issued.

NOTE

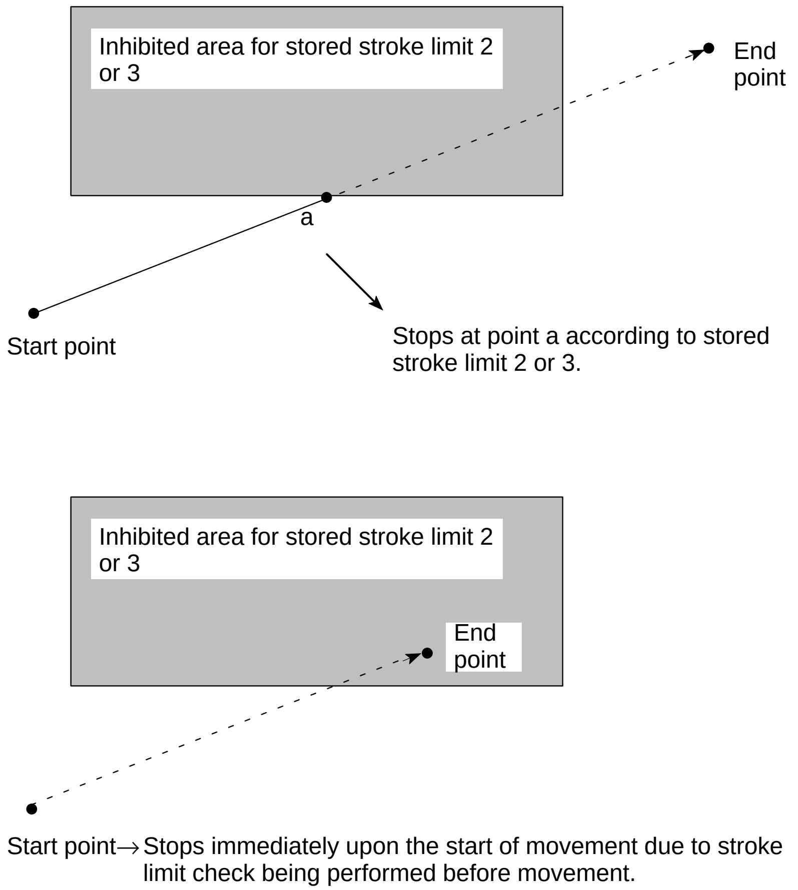
This function checks only whether the end point falls within an inhibited area. It does not check whether the tool passes through an inhibited area between the start and end points. However, an alarm is issued upon a tool's entering an inhibited area according to stored stroke limit 1, 2, or 3.

Example 1)





Example 2)



Stops at point a according to stored stroke limit 2 or 3.

Start point→Stops immediately upon the start of movement due to stroke limit check being performed before movement.

31.2.7 Externally Setting the Stroke Limit

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.

31.2.8 Chuck/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.

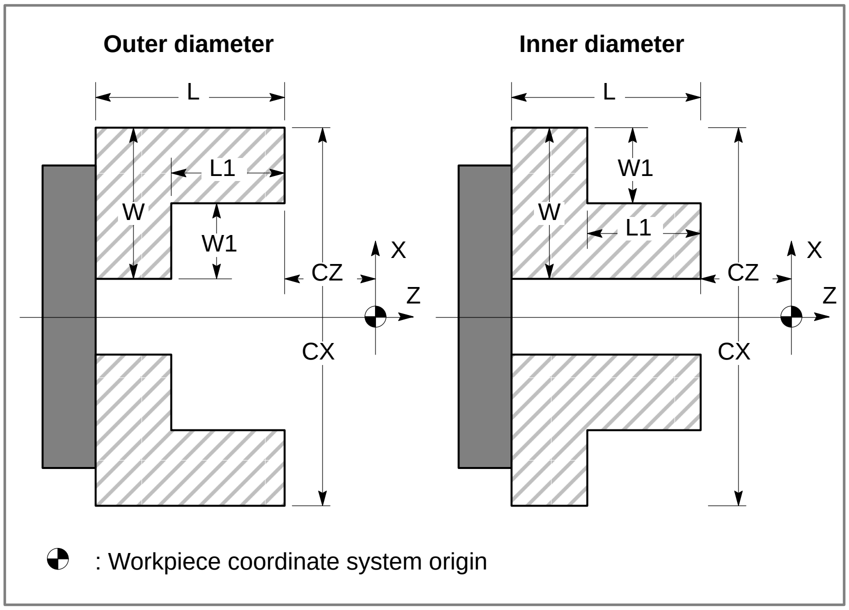
When a tool enters the area of entry prohibition during cutting, the travelling of tool is stopped and an alarm message is displayed. The tool can be escaped from the prohibition area by moving in the opposite direction to that on entry. The Yes/No of this function is selected by the G22 (Stored stroke 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.

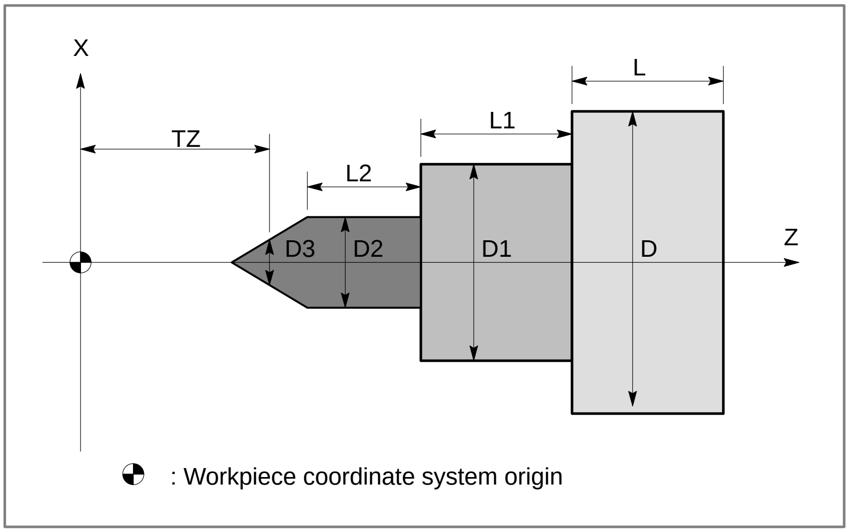
Explanations

- **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.

31.3
INTERLOCK

31.3.1
Interlock per 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.

31.3.2
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.

31.3.3
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.

31.3.4
Start Lock

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.

31.4 EXTERNAL DECELERATION

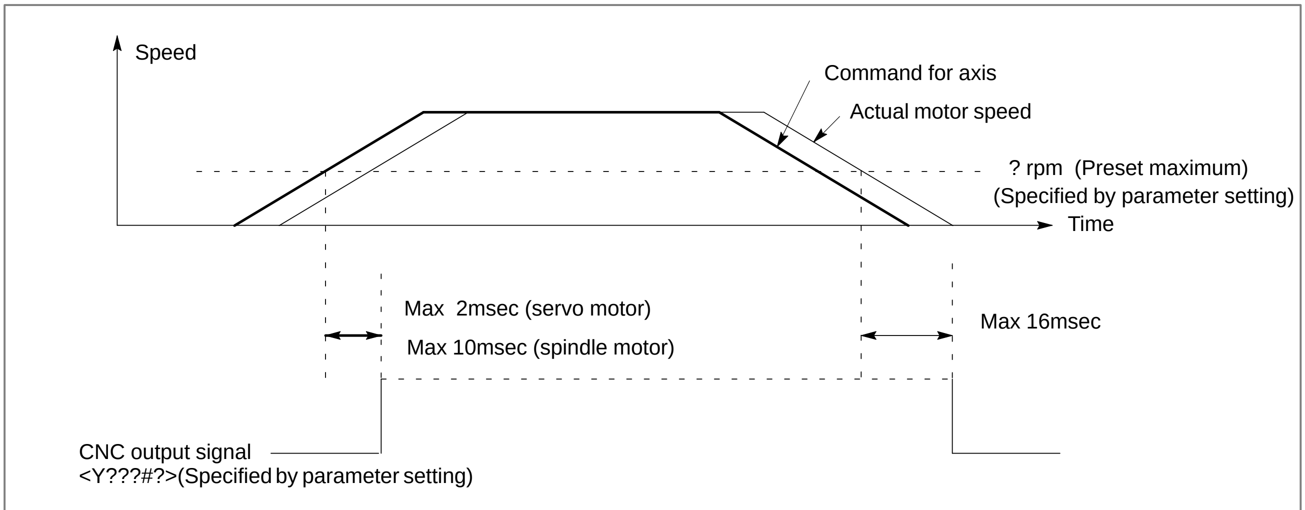
Feed rate can be decelerated by an external deceleration signal from the machine side. A feed rate after deceleration can be set by parameter. External deceleration is prepared every axis and every direction.
When the tool is to be moved in the reverse direction, futile time may not be wasted since no 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.

31.5 ABNORMAL LOAD DETECTION

When a cutting tool collides with the machine body or is damaged during cutting, the load torque applied to the servo 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.

31.6 SERVO/SPINDLE MOTOR SPEED DETECTION

The servo axis and spindle motor speeds are monitored. If the speed of an axis exceeds a preset maximum (specified by parameter setting), the corresponding signal is output to a Y address (specified by parameter setting) of the PMC.
The following diagram illustrates the signal output state.



Note) The status of each signal is updated every 8 msec.
(Fluctuations in the speed of less than 8 msec duration cannot be detected, therefore.)

32 STATUS OUTPUT

32.1 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.

32.2 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.

32.3 REWINDING SIGNAL

This signal shows that tape reader or main program in memory is rewinding.

32.4 ALARM SIGNAL

This signal is transmitted when the NC comes under an alarm status.

32.5 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.

32.6 AUTOMATIC OPERATION SIGNAL

This signal is sent out when it is under automatic operation.

32.7 AUTOMATIC OPERATION START SIGNAL

This signal is sent out when automatic operation is being activated.

32.8 FEED HOLD SIGNAL

This signal is sent out when automatic operation is held by feed hold.

32.9 RESET SIGNAL

This signal is sent out to show that the NC has been reset.

32.10 IN–POSITION SIGNAL

This signal shows that an axis is under in–position status. This signal is output for all axes.

32.11 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.

32.12
AXIS MOVE DIRECTION SIGNAL

This signal is output to show move direction of each axis. This signal is output for each axis.

32.13
RAPID TRAVERSING SIGNAL

This signal shows that the move command is done under rapid traverse.

32.14
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.

32.15
THREAD CUTTING SIGNAL

This signal shows that the machine is under thread cutting mode (G33) or thread cutting cycle (T series).

32.16
CONSTANT SURFACE SPEED CONTROL SIGNAL

This signal shows that the machine is under constant surface speed control mode (G96).

32.17
INCH INPUT SIGNAL

This signal shows that input is done under inch input mode (G20).

32.18
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

32.19
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 ten position switch signals can be output.

This signal is transmitted when the NC comes under an alarm status.

33 EXTERNAL DATA INPUT

The external data input is as follows.

- External tool compensation
- External program number search
- External work 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

33.1
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.

33.2
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.

33.3
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.
The external data input function allows NC operation by data sent from outside the NC (for example from the machine side).

33.4
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.

33.5
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.

33.6
EXTERNAL OPERATOR'S MESSAGE

Message to the operator is given from outside the NC, and the message is displayed.
The message is sent after the message number (0 to 999). Only one message with message number can be sent at a single time. Maximum 255 characters can be used for a single message.
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.

33.7
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.

34 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, <SHIFT>, 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 <SHIFT> to the CNC. This produces the same results as with actual key operations.

35 PERSONAL COMPUTER FUNCTION

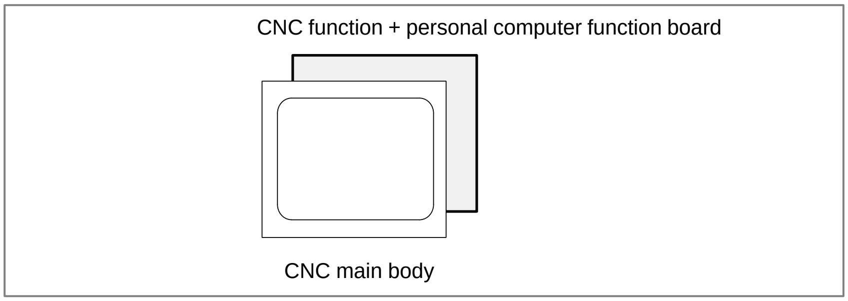
The open CNC allows the machine tool builder to incorporate a high–level man–machine interface, such as conversational automatic programming and conversational operation that makes maximum use of the machine tool builder's know–how.

The personal computer function can be realized in either of two ways: By using the IBM PC–compatible personal computer function that is built into the CNC printed circuit board, combined with a 10.4" color LCD, or by connecting a commercially available IBM PC–compatible personal computer via the high–speed serial bus.

35.1 BUILT–IN PERSONAL COMPUTER FUNCTION

The CNC's built–in personal computer function has the following features:

- High compatibility with the IBM PC(*1)
- Windows 95® operating system, which supports many software products for the IBM PC(*1)
- Optional ISA expansion unit, enabling the installation of commercially available expansion boards (ISA specification) for the IBM PC(*1)
- Installation combined with the CNC
- Direct connection to the CNC via a bus, enabling the high–speed exchange of a wide range of information
- Highly reliable design based on design technologies realized through the development of CNCs



Personal computer software for open CNC

Item	Specification	Remarks
Operating system	Windows®95	
Dedicated drivers	NC program memory driver NC data file driver	Optional Optional
Dedicated library	CNC/PMC data window	
Package software	CNC maintenance package	
	CNC basic operation package	Optional
Development tools	Visual C ++ *1 Visual Basic® *1	Microsoft Corporation Microsoft Corporation

Personal computer hardware for open CNC (personal computer function built into CNC)

Item	Specification	Remarks
CPU	Intel Pentium^{TM} or Intel i486^{TM}DX4 *1	Select either CPU.
Main memory	32M bytes max. 24M bytes max.	For Pentium For i486
Hard disk	1G bytes	Built–in
Display	10.4” color TFT LCD (with a touch pad) 80 characters x 25 lines (ANK) 640 x 480 dots	Simultaneous display of 256 colors *3 A touch pad is optional.
Ports	PCMCIA × 1 slot Full keyboard × 1 Serial (RS–232C) × 1 Mouse × 1 Floppy disk × 1	A touch pad is connected to serial port 1.
	Serial (RS–232C) ×1 (additional) Parallel × 1	Optional Optional
Expansion slots	ISA expansion slot (half size) × 2 *2	Optional
Ambient temperature	Operating: +5°C to +40°C Non–operating: –20°C to +60°C	
Ambient relative humidity	30% to 90% No condensation	

*1 Intel, i486, and Pentium are registered trademarks of Intel Corporation.
Windows and Visual Basic are registered trademarks of Microsoft Corporation.
Visual C++ is a trademark of Microsoft Corporation.
IBM is a registered trademark of IBM Corporation.

*2 Expansion boards for IBM PC are to be provided by the machine tool builder.

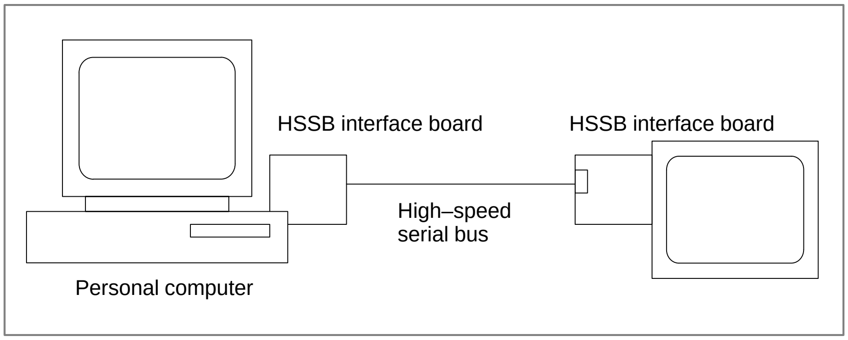
*3 A dedicated driver is required to display more than 16 colors (tones).

35.2 HIGH–SPEED SERIAL BUS (HSSB)

The high–speed serial bus is a serial interface used to transfer data at high speed between the CNC control unit and a personal computer installed on the operator panel side.

By installing a dedicated interface board in a commercially available IBM PC–compatible personal computer, the CNC control unit can be connected to the personal computer via the high–speed serial bus. The high–speed serial bus has the following features:

- Large amounts of data can be transferred between the personal computer and CNC control unit at high speed.
- A highly reliable optical fiber cable is used for connection.
- The machine tool builder can select an appropriate personal computer according to the specifications of the machine system.



Hardware for open CNC (system with commercially available personal computer connected to CNC via high–speed serial bus)

Item	Specification	Remarks
Interface board on CNC	For option slot	Applicable model: 210*i*
Interface board on personal computer	ISA specification Power supply: +5 V only	
Connection cable	Optical fiber cable	Maximum length: 50 m
Personal computer requirements	CPU: 486 or better At least one ISA slot	The installation environment shall satisfy the conditions described in the manual supplied with the personal computer.

III. AUTOMATIC PROGRAMMING FUNCTION

1

OUTLINE OF CONVERSATIONAL AUTOMATIC PROGRAMMING

FANUC provides conversational automatic programming functions for lathes and machining centers. Two conversational automatic programming functions are provided for lathes: Conversational automatic programming function II (called CAP II) and Super CAP T, which use different input methods. For machining centers, Super CAP M is provided.

With these conversational automatic programming functions, the user can enter data and create machining programs easily by following the displayed illustrations and instructions.

In addition, program checking and modifications can be performed easily.

2

CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION FOR LATHES

There are two conversational automatic programming functions for lathes: CAP II and Super CAP T and Super CAP II T.
The difference between the two functions is in the program input method. CAP II uses the symbolic FAPT method for inputting programs. In Super CAP T and Super CAP II T. programs are entered by selecting machining types.

2.1 CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION II (CAP II)

2.1.1 Features

CAP II is a conversational automatic programming function for lathes. It has the following features:

- Part figures can be input in a batch by using the symbolic keys.
- Even complicated part figures can be input by using the automatic intersection calculation function.
- The automatic process determination function creates necessary processes automatically.
- Any cutting direction or area can be specified.
- NC data can be created without superfluous movement, such as cutting through air.
- By using MTF, NC data suitable for the particular machine being used can be created.

2.1.2 Applicable Machines

CAP II can be used with the following lathes:

- 1–spindle/1–turret lathe
- 1–spindle/2–turret lathe
- 2–spindle (main spindle and sub spindle)/1–turret lathe
- Lathe with Y–axis/C–axis machining functions
- Lathe with chasing tool
- Vertical lathe

2.1.3

Outline of the Conversational Automatic Programming Function

Machining types

In CAP II, the following machining types can be determined automatically or selected manually:

- Outer surface rough machining
- Inner surface rough machining
- Outer surface semifinish machining
- Inner surface semifinish machining
- Outer surface finishing
- Inner surface finishing
- Grooving/residual machining
- Threading
- Cutting off
- Bar feed
- Center drilling/drilling/reaming/tapping
- C–axis center drilling/drilling/tapping
- C–axis front face nothing
- C–axis cylindrical grooving
- Y–axis center drilling/drilling/tapping(*1)
- Y–axis pattern machining(*1)
- Y–axis contouring(*1)
- Auxiliary machining(*1)(*2)

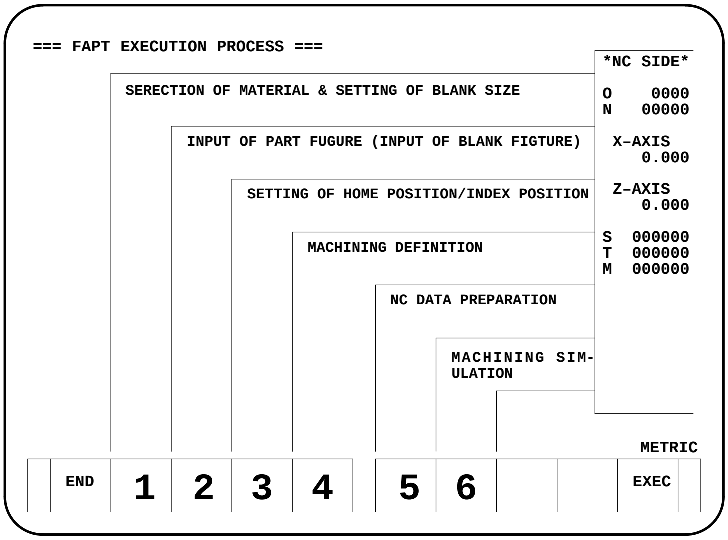
NOTE

*1 Y–axis machining and miscellaneous machining are not determined automatically.

*2 A subprogram can be called from the conversational program.

Basic menu screen

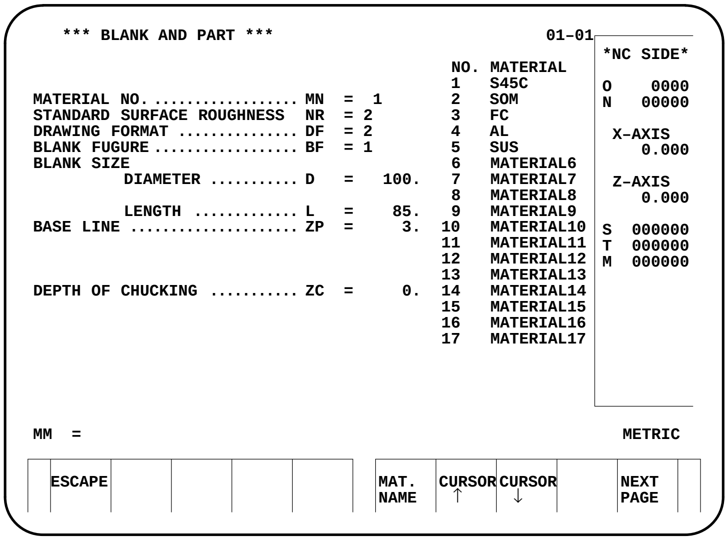
Operations with CAP II always begin with the basic menu screen shown at the following. If the user cannot determine the next operation on a conversational screen, the user can press the leftmost soft key [ESCAPE] to return to the display of this basic menu screen.



```
*NC SIDE*
O    0000
N   00000
 X–AXIS
    0.000
 Z–AXIS
    0.000
S  000000
T  000000
M  000000
```

Material selection and blank size setting screen

When a size is entered, a guide figure can be drawn by pressing the [HELP] soft key.



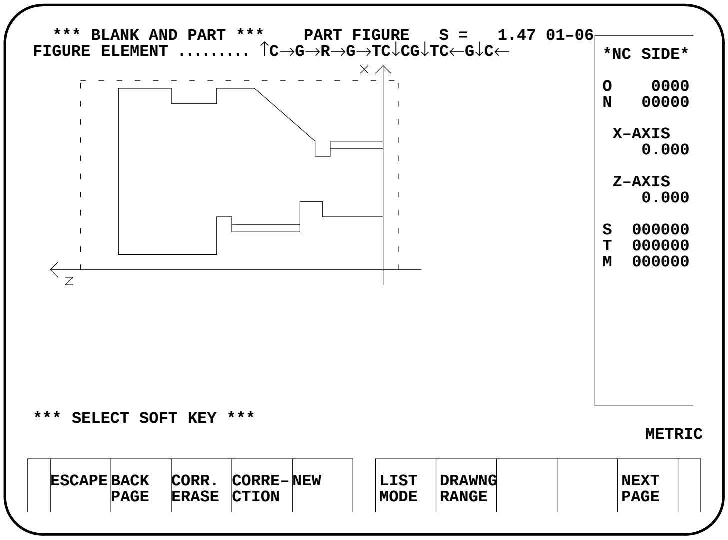
```
   *** BLANK AND PART ***                          01–01
                                            NO. MATERIAL
                                             1   S45C
MATERIAL NO. ................. MN  =  1       2   SOM
STANDARD SURFACE ROUGHNESS  NR  = 2         3   FC
DRAWING FORMAT .............. DF  = 2         4   AL
BLANK FUGURE ................. BF  = 1         5   SUS
BLANK SIZE                                   6   MATERIAL6
         DIAMETER .......... D   =   100.    7   MATERIAL7
                                             8   MATERIAL8
         LENGTH ............ L   =    85.    9   MATERIAL9
BASE LINE .................... ZP  =     3.  10   MATERIAL10
                                            11   MATERIAL11
                                            12   MATERIAL12
                                            13   MATERIAL13
DEPTH OF CHUCKING .......... ZC  =     0.  14   MATERIAL14
                                            15   MATERIAL15
                                            16   MATERIAL16
                                            17   MATERIAL17

MM  =                                                 METRIC

*NC SIDE*
O    0000
N   00000
 X–AXIS
    0.000
 Z–AXIS
    0.000
S  000000
T  000000
M  000000
```

Part figure input screen

Part figures are input in a batch by using symbolic keys (↑ , →, ↓ , ←, ↗ , ↘ , ↙ , ↖ , ↷, and ↶).

Functions are available for simplifying part figure input; these functions include the automatic intersection calculation, pocket calculator format numeric calculation, continuous groove input, chamfering batch input, and figure copy functions.

The input figures are displayed directly on the screen so that they can be checked easily.



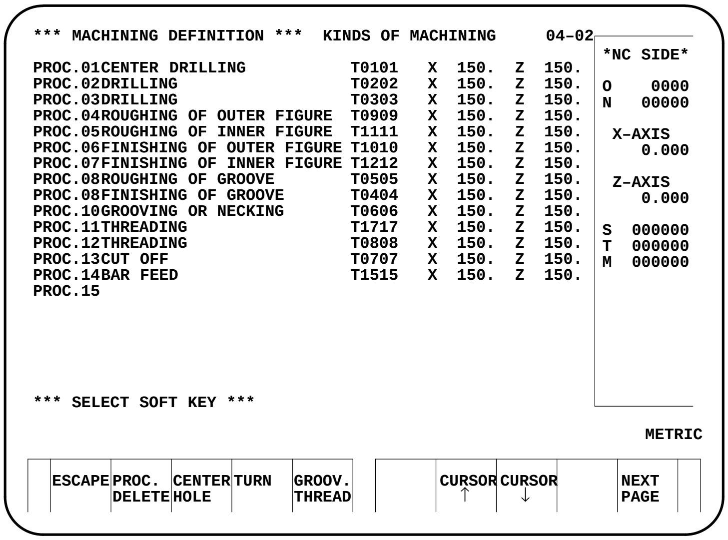
```
  *** BLANK AND PART ***    PART FIGURE   S =   1.47 01–06
FIGURE ELEMENT ......... ↑C→G→R→G→TC↓CG↓TC←G↓C←
                                                        *NC SIDE*

                                                        O    0000
                                                        N   00000

                                                         X–AXIS
                                                            0.000

                                                         Z–AXIS
                                                            0.000

                                                        S  000000
                                                        T  000000
                                                        M  000000

*** SELECT SOFT KEY ***
                                                           METRIC

  ESCAPE BACK  CORR.  CORRE– NEW     LIST  DRAWNG        NEXT
         PAGE  ERASE  CTION          MODE  RANGE         PAGE
```

Process directory display

The automatic process determination function automatically creates the processes shown on the following.

The automatic process determination function automatically determines machining types, tool data, cutting areas, and cutting conditions.

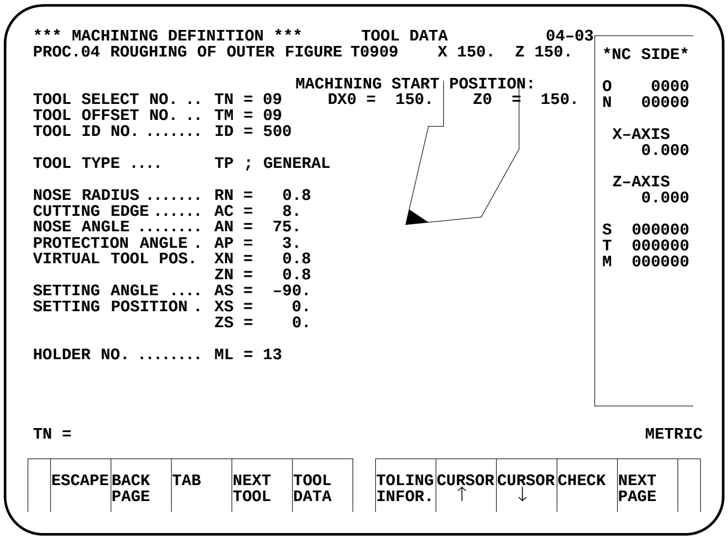
On this screen, processes and edit operations (deletion and insertion of processes) can be selected manually.



```
*** MACHINING DEFINITION ***  KINDS OF MACHINING     04–02
                                                        *NC SIDE*
PROC.01CENTER DRILLING           T0101   X  150.  Z  150.
PROC.02DRILLING                  T0202   X  150.  Z  150.   O    0000
PROC.03DRILLING                  T0303   X  150.  Z  150.   N   00000
PROC.04ROUGHING OF OUTER FIGURE  T0909   X  150.  Z  150.
PROC.05ROUGHING OF INNER FIGURE  T1111   X  150.  Z  150.    X–AXIS
PROC.06FINISHING OF OUTER FIGURE T1010   X  150.  Z  150.       0.000
PROC.07FINISHING OF INNER FIGURE T1212   X  150.  Z  150.
PROC.08ROUGHING OF GROOVE        T0505   X  150.  Z  150.    Z–AXIS
PROC.08FINISHING OF GROOVE       T0404   X  150.  Z  150.       0.000
PROC.10GROOVING OR NECKING       T0606   X  150.  Z  150.
PROC.11THREADING                 T1717   X  150.  Z  150.   S  000000
PROC.12THREADING                 T0808   X  150.  Z  150.   T  000000
PROC.13CUT OFF                   T0707   X  150.  Z  150.   M  000000
PROC.14BAR FEED                  T1515   X  150.  Z  150.
PROC.15

*** SELECT SOFT KEY ***
                                                           METRIC

  ESCAPE PROC.  CENTER TURN   GROOV.       CURSOR CURSOR       NEXT
         DELETE HOLE          THREAD         ↑      ↓          PAGE
```

Tool data input screen

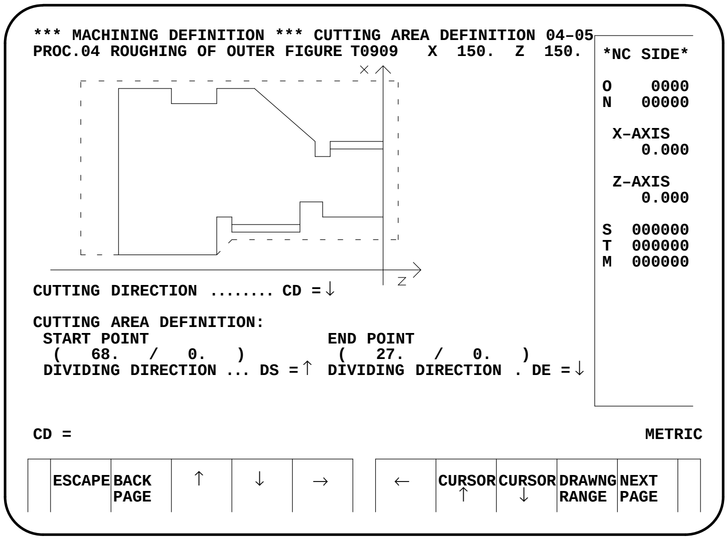
Tool data input and modifications are made on this screen. A tool currently selected is indicated on the screen, allowing the user to easily check tool data.



```
*** MACHINING DEFINITION ***      TOOL DATA          04–03
PROC.04 ROUGHING OF OUTER FIGURE T0909    X 150.  Z 150.    *NC SIDE*

                           MACHINING START POSITION:        O    0000
TOOL SELECT NO. .. TN = 09     DX0 =  150.    Z0  =  150.   N   00000
TOOL OFFSET NO. .. TM = 09
TOOL ID NO. ....... ID = 500                                  X–AXIS
                                                                 0.000
TOOL TYPE ....      TP ; GENERAL
                                                              Z–AXIS
NOSE RADIUS ....... RN =   0.8                                   0.000
CUTTING EDGE ...... AC =   8.
NOSE ANGLE ........ AN =  75.                               S  000000
PROTECTION ANGLE . AP =   3.                                T  000000
VIRTUAL TOOL POS.  XN =   0.8                               M  000000
                   ZN =   0.8
SETTING ANGLE .... AS =  –90.
SETTING POSITION . XS =    0.
                   ZS =    0.

HOLDER NO. ........ ML = 13



TN =                                                         METRIC
```

Screen for setting cutting directions and cutting areas

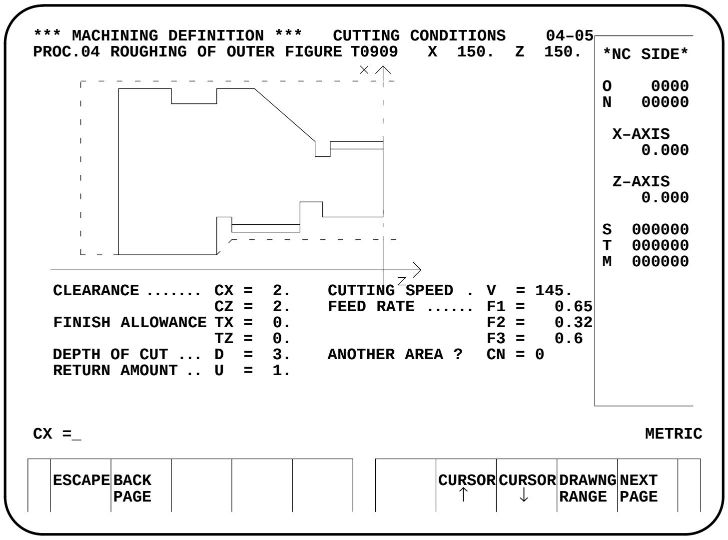
Cutting directions and cutting areas are specified using the arrow keys. Any cutting direction and area can be specified.



```
*** MACHINING DEFINITION *** CUTTING AREA DEFINITION 04–05
PROC.04 ROUGHING OF OUTER FIGURE T0909   X  150.  Z  150.   *NC SIDE*

                                                            O    0000
                                                            N   00000

                                                              X–AXIS
                                                                 0.000

                                                              Z–AXIS
                                                                 0.000

                                                            S  000000
                                                            T  000000
                                                            M  000000

CUTTING DIRECTION  ........ CD =↓

CUTTING AREA DEFINITION:
 START POINT                    END POINT
  (   68.   /   0.   )           (   27.   /   0.   )
 DIVIDING DIRECTION ... DS =↑  DIVIDING DIRECTION . DE =↓


CD =                                                         METRIC
```

Screen for setting cutting conditions

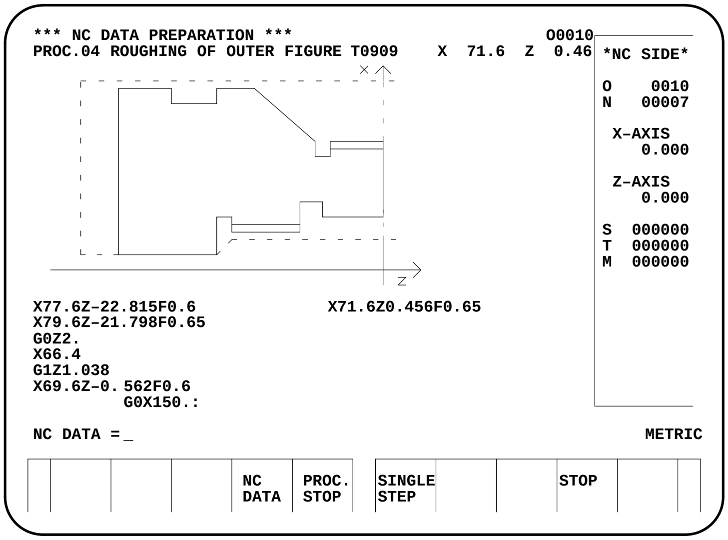
Cutting conditions are input or modified on this screen. The initial values are set automatically according to the parameters and blank material.



```
*** MACHINING DEFINITION ***   CUTTING CONDITIONS    04–05
PROC.04 ROUGHING OF OUTER FIGURE T0909   X  150.  Z  150.   *NC SIDE*

                                                             O    0000
                                                             N   00000

                                                              X–AXIS
                                                                 0.000

                                                              Z–AXIS
                                                                 0.000

                                                             S  000000
                                                             T  000000
                                                             M  000000
  CLEARANCE ....... CX =  2.    CUTTING SPEED . V  = 145.
                    CZ =  2.    FEED RATE ...... F1 =   0.65
  FINISH ALLOWANCE TX =  0.                      F2 =   0.32
                    TZ =  0.                     F3 =   0.6
  DEPTH OF CUT ... D  =  3.    ANOTHER AREA ?   CN = 0
  RETURN AMOUNT .. U  =  1.

CX =_                                                       METRIC
```

NC data creation screen

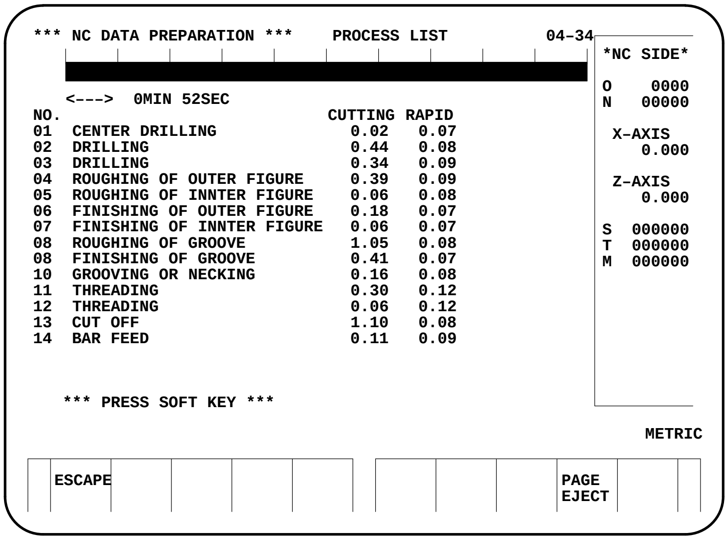
NC data appears, and a tool path is drawn on this screen, allowing the user to easily check NC data. Switching between animated simulation and tool path drawing is enabled with a soft key.



```
*** NC DATA PREPARATION ***                           O0010
PROC.04 ROUGHING OF OUTER FIGURE T0909    X  71.6  Z  0.46  *NC SIDE*

                                                             O    0010
                                                             N   00007

                                                              X–AXIS
                                                                 0.000

                                                              Z–AXIS
                                                                 0.000

                                                             S  000000
                                                             T  000000
                                                             M  000000
X77.6Z–22.815F0.6              X71.6Z0.456F0.65
X79.6Z–21.798F0.65
G0Z2.
X66.4
G1Z1.038
X69.6Z–0. 562F0.6
         G0X150.:

NC DATA =_                                                  METRIC
```

Machining time display screen

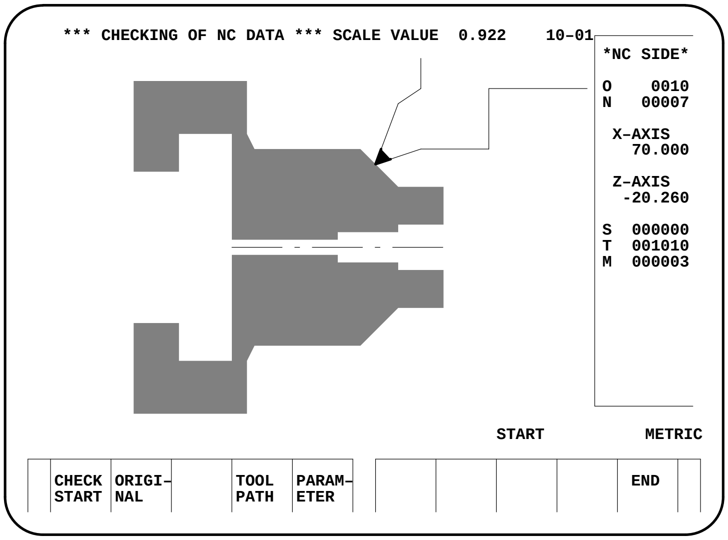
The cutting time and rapid traverse time are displayed for each machining type. A bar chart is displayed so that the user can check the time allotment at a glance.



```
*** NC DATA PREPARATION ***     PROCESS LIST          04–34
                                                            *NC SIDE*

   <–––>  0MIN 52SEC                                        O    0000
NO.                                CUTTING RAPID            N   00000
01  CENTER DRILLING                  0.02   0.07
02  DRILLING                         0.44   0.08             X–AXIS
03  DRILLING                         0.34   0.09                0.000
04  ROUGHING OF OUTER FIGURE         0.39   0.09
05  ROUGHING OF INNTER FIGURE        0.06   0.08             Z–AXIS
06  FINISHING OF OUTER FIGURE        0.18   0.07                0.000
07  FINISHING OF INNTER FIGURE       0.06   0.07
08  ROUGHING OF GROOVE               1.05   0.08            S  000000
08  FINISHING OF GROOVE              0.41   0.07            T  000000
10  GROOVING OR NECKING              0.16   0.08            M  000000
11  THREADING                        0.30   0.12
12  THREADING                        0.06   0.12
13  CUT OFF                          1.10   0.08
14  BAR FEED                         0.11   0.09


   *** PRESS SOFT KEY ***

                                                               METRIC
```

Animated simulation screen

The user can check the memory operation conditions on the screen. A function for checking for interference between the chuck/tailstock and tool is provided. An interference check can be made before actual operation by performing memory operation with the machine lock set to ON.



```
   *** CHECKING OF NC DATA *** SCALE VALUE  0.922    10–01
                                                            *NC SIDE*

                                                            O    0010
                                                            N   00007

                                                             X–AXIS
                                                              70.000

                                                             Z–AXIS
                                                             -20.260

                                                            S  000000
                                                            T  001010
                                                            M  000003

                                               START           METRIC
```

2.2 SUPER CAP T/ SUPER CAP II T

2.2.1 Features

Super CAP T and Super CAP II T are conversational automatic programming functions a for lathes. It has the following features:

- Simple operation
- Program input by selecting machining processes
- Conversational setup operation by following displayed setup instructions
- Direct execution of conversational programs. The program can also be converted to an NC format, then executed.
- Customization by the machine tool builder.
- About Super CAP II T

Super CAP II T is a development of Super CAP T for Series 16*i*/18*i*–TA. It features the following new functions:

1 Background color for the display screen, and 3–D frames for windows and soft keys
2 Buttons appear ”pressed” when selected
3 Machining simulation for turning based on a solid model
(Super CAP T for Series 16*i*/18*i*–TA can be used this function)
4 Tool trajectory drawing based on isometric projection in C–/Y–axis machining simulation
(Super CAP T for Series 16*i*/18*i*–TA can be used this function)
5 Selectable screen display colors, with the saving of up to four color schemes supported

All other functions, such as screen displays, key operations, and machining functions, are the same as those of super CAP T.
Moreover, machining programs, tool data, and conversational function parameters created with Super CAP T can also be used with Super CAP II T.

2.2.2 Applicable Machines

Super CAP T and Super CAP II T can be used with the following lathes:

- 1–spindle/1–turret lathe
- 1–spindle/2–turret lathe
- Facing 2–spindle/2–turret lathe (The two turrets operate independently of each other.)
- 2–spindle (main spindle and sub spindle)/1–turret lathe
- Lathe with Y–axis/C–axis machining functions

2.2.3
Outline of the Conversational Automatic Programming Function

Machining types

In Super CAP T and Super CAP II T , the following machining types can be selected:

- Bar machining
- Pattern repeating
- Residual machining
- End facing
- Threading
- Grooving
- Necking
- Center drilling/drilling/reaming/boring/tapping
- Single action

NOTE
A command equivalent to one block of an NC program can be input conversationally.

- Subcall

NOTE
A subprogram can be called from the conversational program.

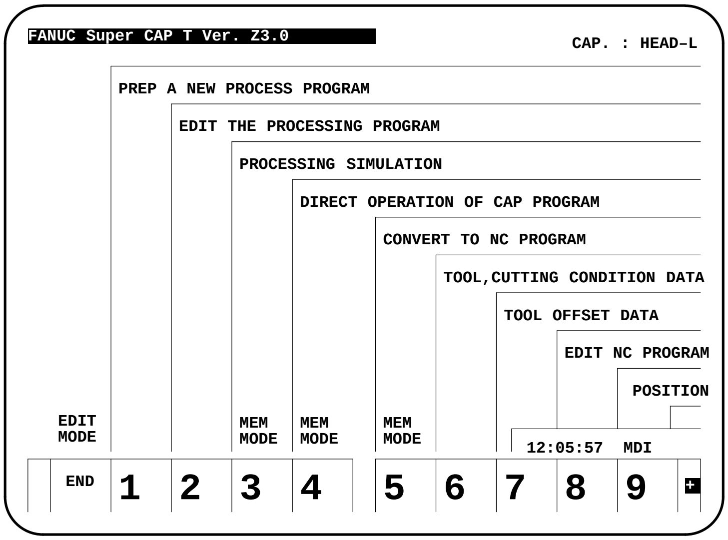
- Auxiliary processes and transfer process

NOTE
The machine tool builder can include machine–specific operations in the conversational function.

- M code/end process
- C–axis center drilling/drilling/reaming/boring/tapping
- C–axis grooving
- C–axis nothing
- C–axis cylindrical machining
- Y–axis center drilling/drilling/reaming/tapping
- Y–axis milling

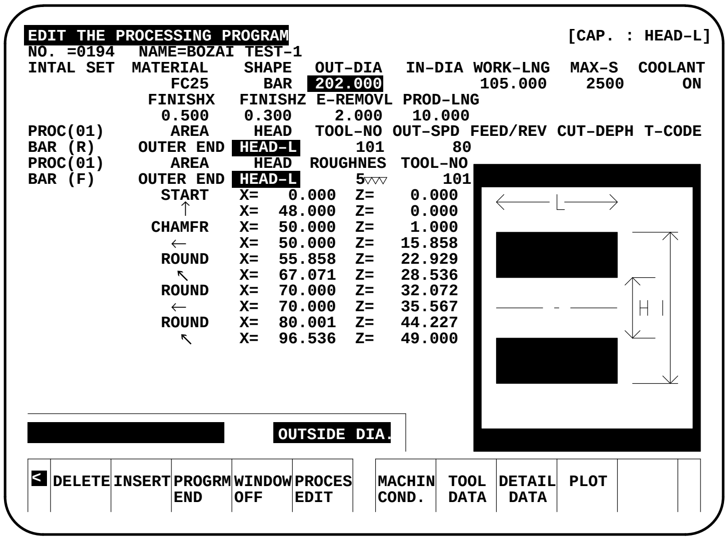
Basic menu screen

Operations with Super CAP T and Super CAP II always begin with the basic menu screen shown here. If the user cannot determine the next operation on a conversational screen, the user can press the leftmost soft key ◁ to return the display to this basic menu screen.
Following each screens are displayed in Super CAP T.
For Super CAP II T some screens are different.



Conversational program input screen

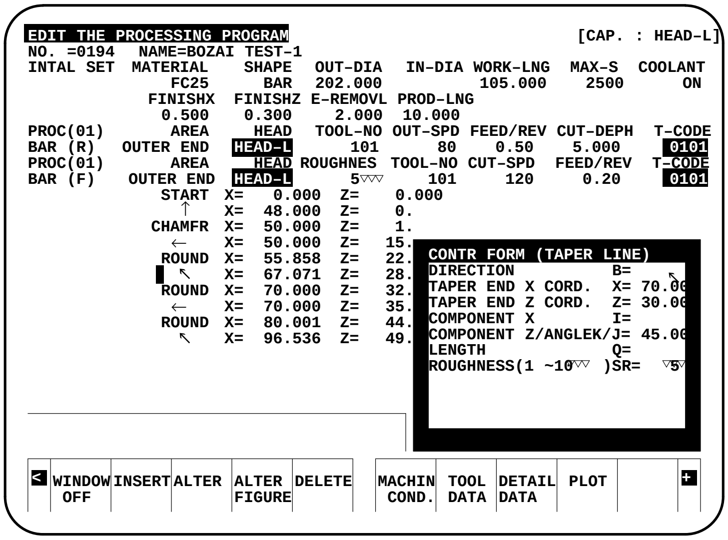
A program can be input easily by following the instructions displayed in a window.



```
EDIT THE PROCESSING PROGRAM                                [CAP. : HEAD–L]
NO. =0194   NAME=BOZAI TEST–1
INTAL SET  MATERIAL     SHAPE   OUT–DIA   IN–DIA WORK–LNG  MAX–S  COOLANT
                FC25      BAR   202.000          105.000    2500       ON
              FINISHX   FINISHZ E–REMOVL PROD–LNG
                0.500     0.300    2.000   10.000
PROC(01)        AREA       HEAD  TOOL–NO OUT–SPD FEED/REV CUT–DEPH T–CODE
BAR (R)      OUTER END  HEAD–L       101       80
PROC(01)        AREA       HEAD ROUGHNES  TOOL–NO
BAR (F)      OUTER END  HEAD–L      5▽▽▽      101
               START   X=    0.000  Z=    0.000
                 ↑     X=   48.000  Z=    0.000
              CHAMFR   X=   50.000  Z=    1.000
                 ←     X=   50.000  Z=   15.858
               ROUND   X=   55.858  Z=   22.929
                 ↖     X=   67.071  Z=   28.536
               ROUND   X=   70.000  Z=   32.072
                 ←     X=   70.000  Z=   35.567
               ROUND   X=   80.001  Z=   44.227
                 ↖     X=   96.536  Z=   49.000

                             OUTSIDE DIA.
```

Conversational program input screen (inputting a machining profile)

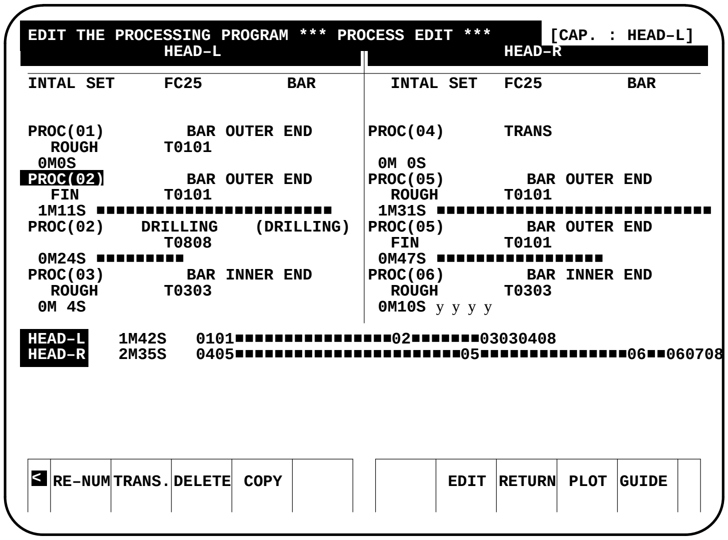
A machining profile can be input easily by using intersection automatic calculation and pocket calculator format calculation.
The input profile is displayed directly on the screen so that the user can check the profile easily.



```
EDIT THE PROCESSING PROGRAM                                  [CAP. : HEAD–L]
NO. =0194  NAME=BOZAI TEST–1
INTAL SET  MATERIAL    SHAPE  OUT–DIA   IN–DIA WORK–LNG  MAX–S  COOLANT
                FC25     BAR  202.000          105.000    2500       ON
             FINISHX  FINISHZ E–REMOVL PROD–LNG
               0.500    0.300    2.000  10.000
PROC(01)        AREA     HEAD  TOOL–NO OUT–SPD FEED/REV CUT–DEPH  T–CODE
BAR (R)   OUTER END    HEAD–L      101      80     0.50    5.000    0101
PROC(01)        AREA     HEAD ROUGHNES  TOOL–NO CUT–SPD  FEED/REV  T–CODE
BAR (F)    OUTER END   HEAD–L     5▽▽▽      101     120      0.20    0101
               START  X=   0.000  Z=    0.000
                 ↑    X=  48.000  Z=    0.
              CHAMFR  X=  50.000  Z=    1.
                 ←    X=  50.000  Z=   15.  CONTR FORM (TAPER LINE)
               ROUND  X=  55.858  Z=   22.  DIRECTION           B=    ↖
                 ↖    X=  67.071  Z=   28.  TAPER END X CORD.   X= 70.0
               ROUND  X=  70.000  Z=   32.  TAPER END Z CORD.   Z= 30.0
                 ←    X=  70.000  Z=   35.  COMPONENT X         I=
               ROUND  X=  80.001  Z=   44.  COMPONENT Z/ANGLEK/J= 45.0
                 ↖    X=  96.536  Z=   49.  LENGTH              Q=
                                            ROUGHNESS(1 ~10▽▽▽ )SR=  ▽5▽
```

Process directory display screen

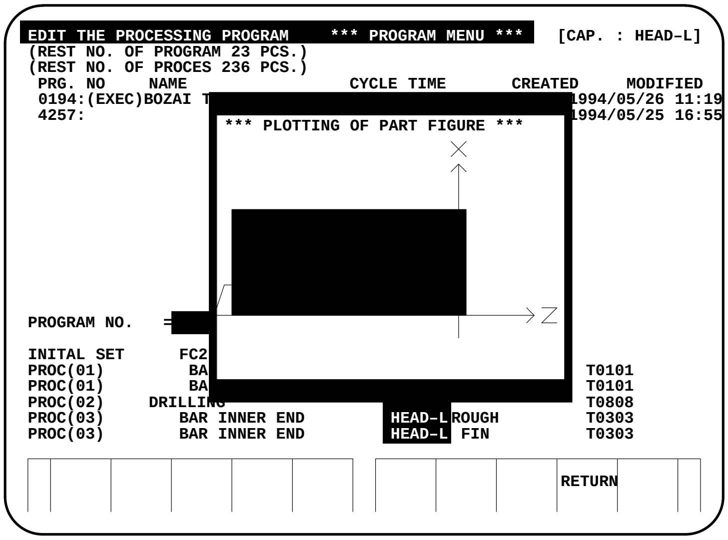
The entire machining program can be checked easily from the process directory display screen that also indicates the execution time for each process.In addition, editing operations including movement, copy, and deletion of processes can be performed on this screen.



```
EDIT THE PROCESSING PROGRAM *** PROCESS EDIT ***     [CAP. : HEAD–L]
                  HEAD–L                               HEAD–R
INTAL SET     FC25        BAR     INTAL SET  FC25         BAR


PROC(01)        BAR OUTER END    PROC(04)      TRANS
  ROUGH       T0101
 0M0S                             0M 0S
PROC(02)        BAR OUTER END    PROC(05)        BAR OUTER END
  FIN         T0101                ROUGH       T0101
 1M11S ■■■■■■■■■■■■■■■■■■■■■■■■    1M31S ■■■■■■■■■■■■■■■■■■■■■■■■■■■■■
PROC(02)    DRILLING    (DRILLING) PROC(05)        BAR OUTER END
              T0808                FIN         T0101
 0M24S ■■■■■■■■■                   0M47S ■■■■■■■■■■■■■■■■■
PROC(03)        BAR INNER END    PROC(06)        BAR INNER END
  ROUGH       T0303                ROUGH       T0303
 0M 4S                             0M10S y y y y

HEAD–L   1M42S   0101■■■■■■■■■■■■■■■■02■■■■■■■03030408
HEAD–R   2M35S   0405■■■■■■■■■■■■■■■■■■■■■■■■05■■■■■■■■■■■■■■■■06■■060708
```

Program directory screen

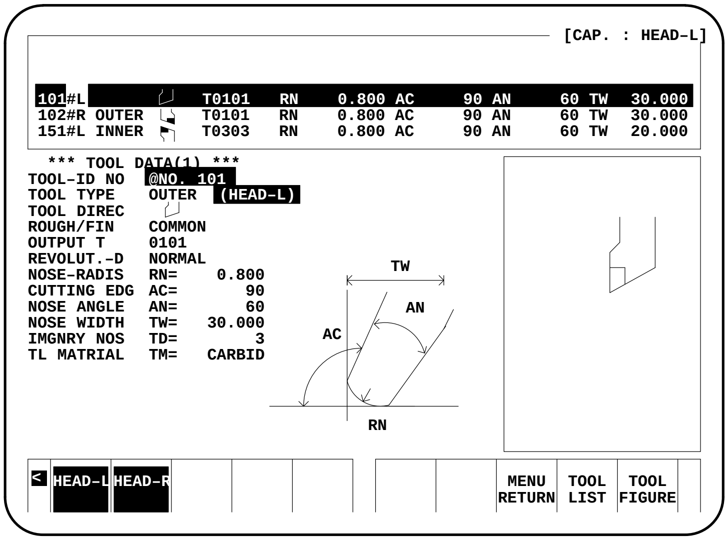
Programs created conversationally are listed on the program directory screen. The user can choose from these programs.The figure produced by a specific program can be displayed in a window for checking.



```
EDIT THE PROCESSING PROGRAM     *** PROGRAM MENU ***   [CAP. : HEAD–L]
(REST NO. OF PROGRAM 23 PCS.)
(REST NO. OF PROCES 236 PCS.)
 PRG. NO    NAME                         CYCLE TIME      CREATED     MODIFIED
 0194:(EXEC)BOZAI T                                             1994/05/26 11:19
 4257:                                                          1994/05/25 16:55
                    *** PLOTTING OF PART FIGURE ***

PROGRAM NO.   =

INITAL SET      FC2
PROC(01)         BA                                             T0101
PROC(01)         BA                                             T0101
PROC(02)     DRILLING                                           T0808
PROC(03)        BAR INNER END            HEAD–L ROUGH           T0303
PROC(03)        BAR INNER END            HEAD–L  FIN            T0303
```

Tool/cutting condition/ pre–tool automatic determination

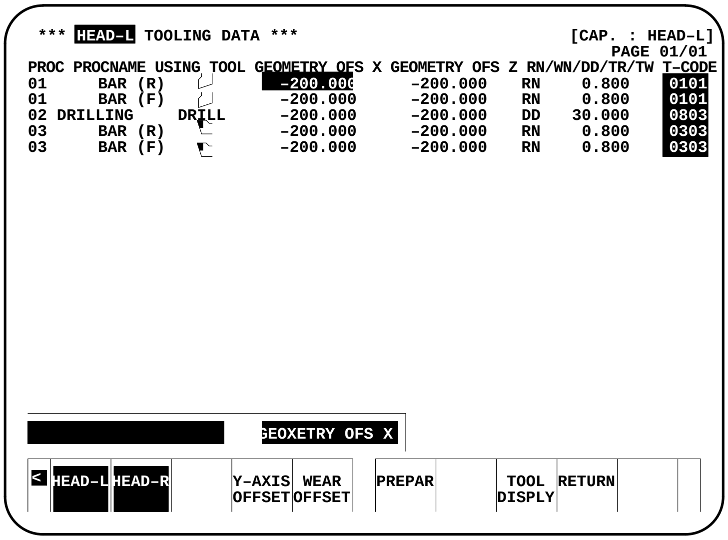
Tool data, cutting condition data, and pre–tool data can be input easily by following the instructions displayed on the screen.Once data is input, necessary data for machining is determined automatically.



```
                                                      [CAP. : HEAD–L]

 101#L                T0101   RN    0.800 AC     90 AN      60 TW   30.000
 102#R OUTER          T0101   RN    0.800 AC     90 AN      60 TW   30.000
 151#L INNER          T0303   RN    0.800 AC     90 AN      60 TW   20.000
  *** TOOL DATA(1) ***
TOOL–ID NO   @NO. 101
TOOL TYPE    OUTER  (HEAD–L)
TOOL DIREC
ROUGH/FIN    COMMON
OUTPUT T     0101
REVOLUT.–D   NORMAL
NOSE–RADIS   RN=     0.800
CUTTING EDG  AC=        90
NOSE ANGLE   AN=        60
NOSE WIDTH   TW=    30.000
IMGNRY NOS   TD=         3
TL MATRIAL   TM=    CARBID
```

Tooling screen

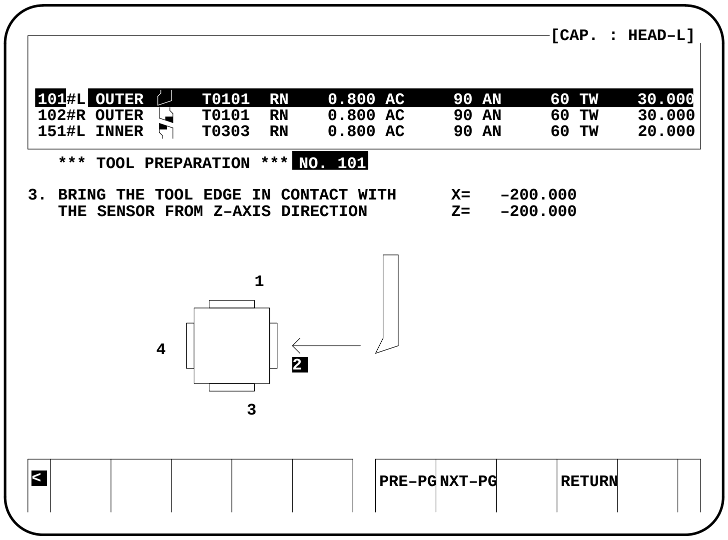
Tool assignment to the turret and tool offset measurement for each tool can be performed easily on the tooling screen which lists the tools used in the machining program.



```
 *** HEAD–L TOOLING DATA ***                                  [CAP. : HEAD–L]
                                                                 PAGE 01/01
PROC PROCNAME USING TOOL GEOMETRY OFS X GEOMETRY OFS Z RN/WN/DD/TR/TW T–CODE
01      BAR (R)             –200.000       –200.000    RN     0.800     0101
01      BAR (F)             –200.000       –200.000    RN     0.800     0101
02 DRILLING     DRILL       –200.000       –200.000    DD    30.000     0803
03      BAR (R)             –200.000       –200.000    RN     0.800     0303
03      BAR (F)             –200.000       –200.000    RN     0.800     0303

                                GEOXETRY OFS X
```

Setup instructions

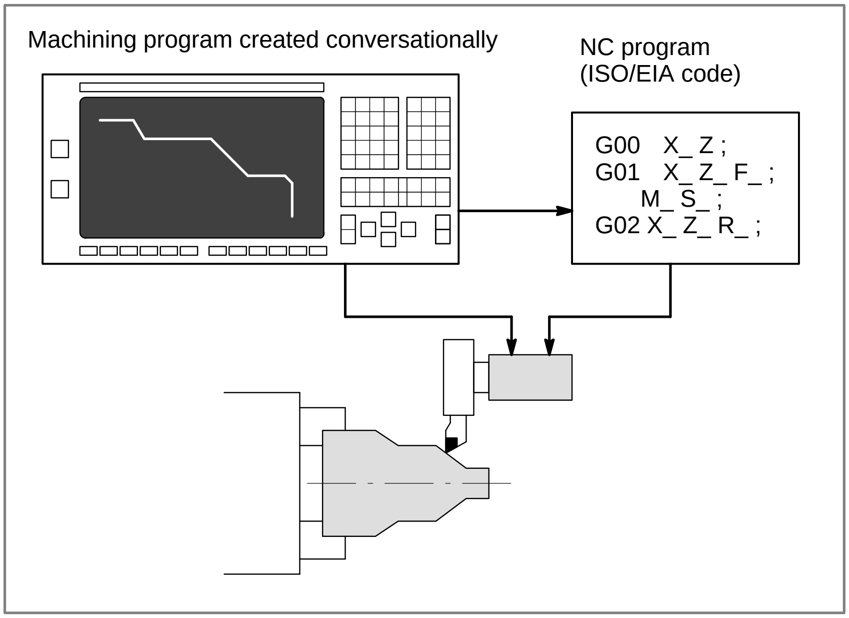
By following the setup instructions displayed conversationally, tool geometry compensation, tool–change position, chuck barrier, and tailstock barrier can be set easily.



```
                                                            [CAP. : HEAD–L]


 101#L OUTER       T0101   RN     0.800 AC     90 AN     60 TW     30.000
 102#R OUTER       T0101   RN     0.800 AC     90 AN     60 TW     30.000
 151#L INNER       T0303   RN     0.800 AC     90 AN     60 TW     20.000
    *** TOOL PREPARATION *** NO. 101

3.  BRING THE TOOL EDGE IN CONTACT WITH      X=   –200.000
    THE SENSOR FROM Z–AXIS DIRECTION         Z=   –200.000
```

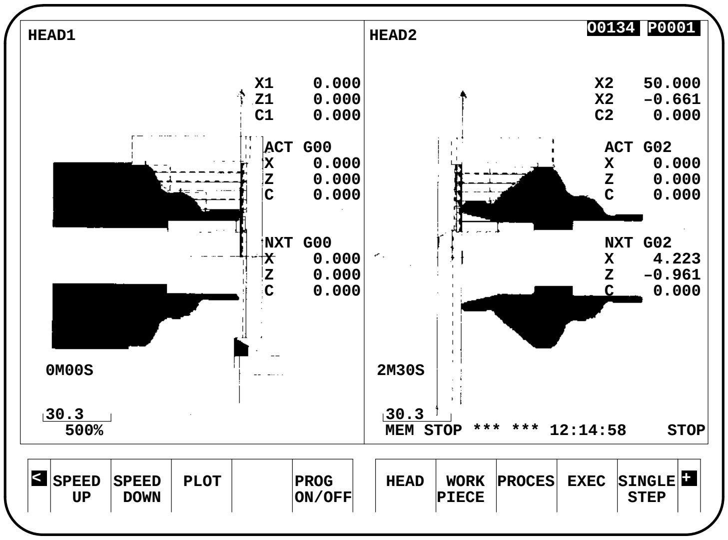
NC program output function

The machining program created conversationally can be run directly. The program can also be converted and executed as an NC program. Furthermore, when modifications are made to the NC program obtained by the conversion, a much more efficient machining program can be created.



Machining simulation

A variety of machining simulations, such as simultaneous animated simulation for the facing 2–spindle 2–path lathe, animated simulation for the 1–spindle/2–turret 2–path lathe, and animated simulation of C–axis/Y–axis machining can be performed.


```
HEAD1                                   HEAD2               O0134 P0001

                        X1     0.000                        X2   50.000
                        Z1     0.000                        X2   -0.661
                        C1     0.000                        C2    0.000

                        ACT G00                             ACT G02
                        X      0.000                        X     0.000
                        Z      0.000                        Z     0.000
                        C      0.000                        C     0.000

                        NXT G00                             NXT G02
                        X      0.000                        X     4.223
                        Z      0.000                        Z    -0.961
                        C      0.000                        C     0.000

  0M00S                                 2M30S

  30.3                                  30.3
   500%                                 MEM STOP *** *** 12:14:58     STOP
```

Adding machine–specific unique know–how

By using the software package provided for the macro compiler/executor, auxiliary operations specific to the machine, such as measurement on the machine, bar feeder, cutting–off, and transfer of blanks, can easily be included in the conversational function.The newly added processes can be displayed and edited the same as with other existing processes.

2.3 SYMBOLIC CAP T

2.3.1 Features of Symbolic CAP T

Symbolic CAP T is a conversational automatic programming function for lathes. It has the following features:

- User–friendly operation procedure that is easy to use, even by beginners
  Symbolic CAP T is easy even for beginners to use, thanks to its support of graphical menus (icons) and mouse–driven conversational processing. In addition, it provides a wealth of help messages.
- Operating procedure for experienced operators
  Symbolic CAP T commands can be entered from the keyboard, enableing experienced operators to operate the system quickly.
- Customizing
  Symbolic CAP T can be customized. It is possible to assign a frequently used function to a menu that is constantly displayed, allowing that function to be called merely by clicking a button.
- One–click switching of screen
  The screen can be switched between that for maching definition and that for figure preparation simply by clicking button. Therefore, even if a figure is found to be missing during machining definition, figure input need not be repeated from the very beginning.
- Symbolic figure input
  Blank and part figures can be entered using conventional symbolic figure input based on arrows.
- Fully automatic process determination
  An optimum machining type, a tool, a portion to be cut, and machining conditions can be automatically selected simply by entering blank and part figures. Even beginners can create machining programs quickly and easily.
- Machining simulation
  Three–dimensional animated machining simulation and tool path drawing are performed simultaneously. Animated machining simulation allows the operator to actually observe the progress of machining. Tool path drawing enables the operator to check the tool path generated by a machining program in detail.,

2.3.2 Applicable Machines

Symbolic CAP T can be used with the following lathes, simply by adding an optional module:

- Basic module VI
  - ❑ 1 spindle, 1 turret (1 path)
  - ❑ 2 spindles, 1 turret (1 path)(*)
- C–/Y–axis module(*)
  - ❑ Lathe supporting Y–/C–axis machining
- 2–path control module(*)
  - ❑ 1 spindle, 2 turrets (2 paths)
  - ❑ 2 spindles, 2 turrets (2 paths)

* Under development

2.3.3 Conversational Automatic Programming Function

Figure input

- Symbolic input
- CAD input
- CAD data input (DXF)

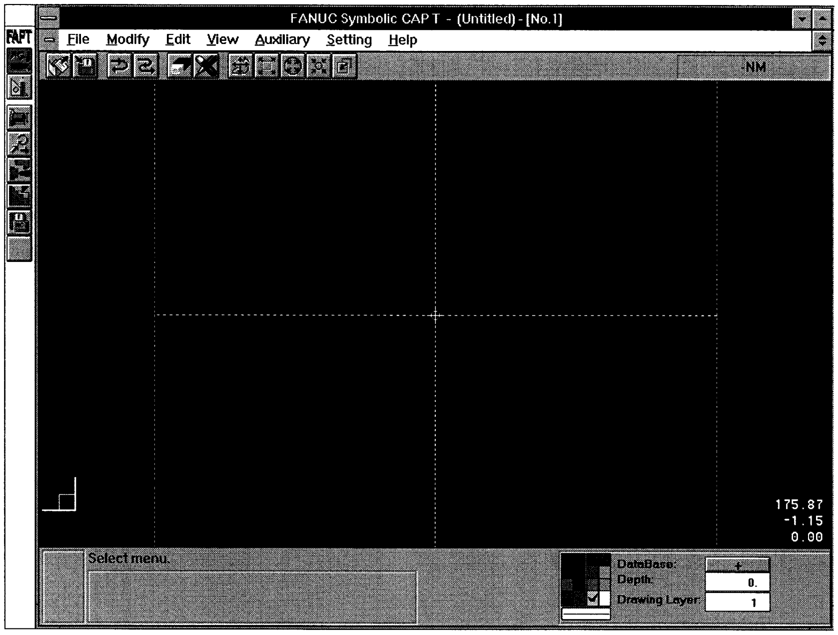
Machining type

Symbolic CAP T can automatically determine the following machining types. It also supports manual selection.

- Basic module/2–path control module
  - Center hole machining (center drilling, drilling, reaming, tapping)
  - Roughing (outer, inner, or end surface) (normal or reverse tool)
  - Semi–finishing (outer, inner, or end surface) (normal or reverse tool)
  - Finishing (outer, inner, or end surface) (normal or reverse tool)
  - Rough grooving (outer, inner, or end surface)
  - Finish grooving (outer, inner, or end surface)
  - Rough/finish grooving (outer, inner, or end surface)
  - Threading (outer, inner, or front surface)
- C–/Y–axis module
  - Milling and hole machining using the C– and Y–axes

Symbolic CAP T basic screen

- Symbolic CAP T is easy to use for all users, from beginners to experts, thanks to its support of graphical menus (icons) and mouse–driven conversational processing.



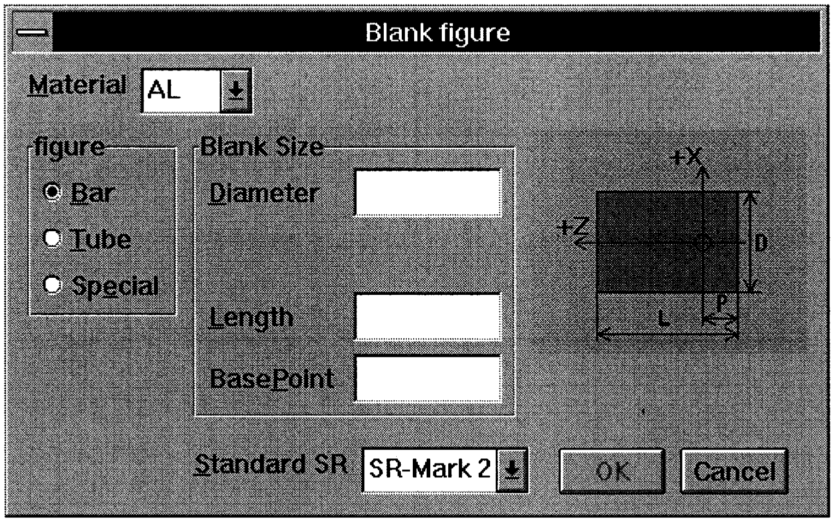
175.87
-1.15
0.00

Select menu.

DataBase: +
Depth: 0.
Drawing Layer: 1

Material selection and blank dimension setting screen

- Blanks having standard shapes, such as bars and tubes, can be specified easily.
- Complicated blank figures can be prepared in the same way as the figures for parts.



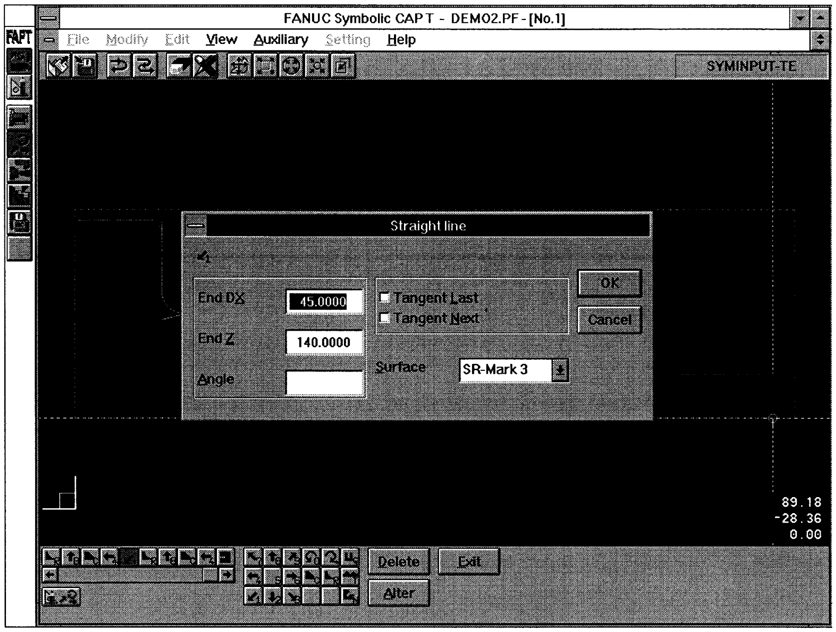
Material: AL

figure: ● Bar ○ Tube ○ Special

Standard SR: SR-Mark 2

Part figure input screen

- Figures for parts can easily be input by using symbolic keys such as arrows.
- The input figures are drawn on the screen, thus making checking easier.



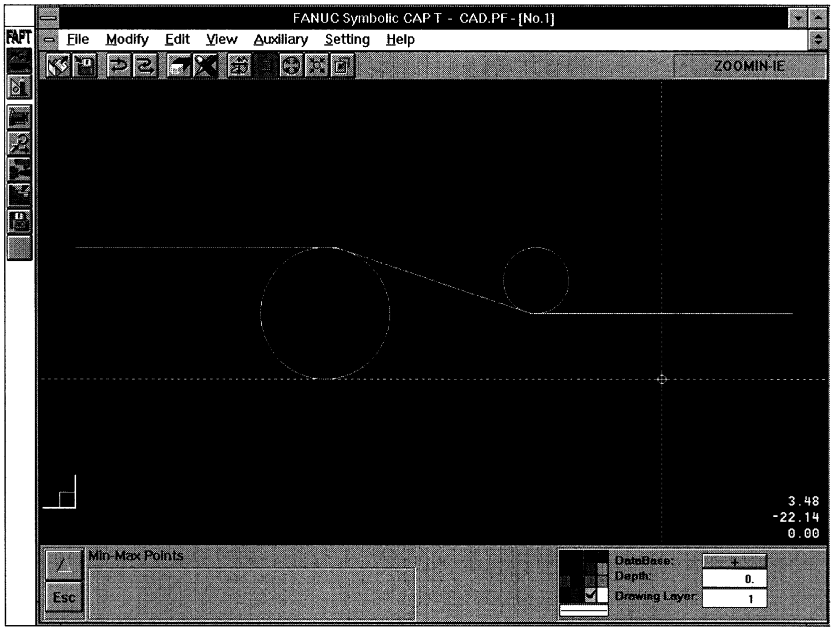
FANUC Symbolic CAP T - DEMO2.PF - [No.1]

Straight line

End DX 45.0000
End Z 140.0000
Angle
☐ Tangent Last
☐ Tangent Next
Surface SR-Mark 3

89.18
-28.36
0.00

- CAD input facilitates the input of complicated figures.



FANUC Symbolic CAP T - CAD.PF - [No.1]

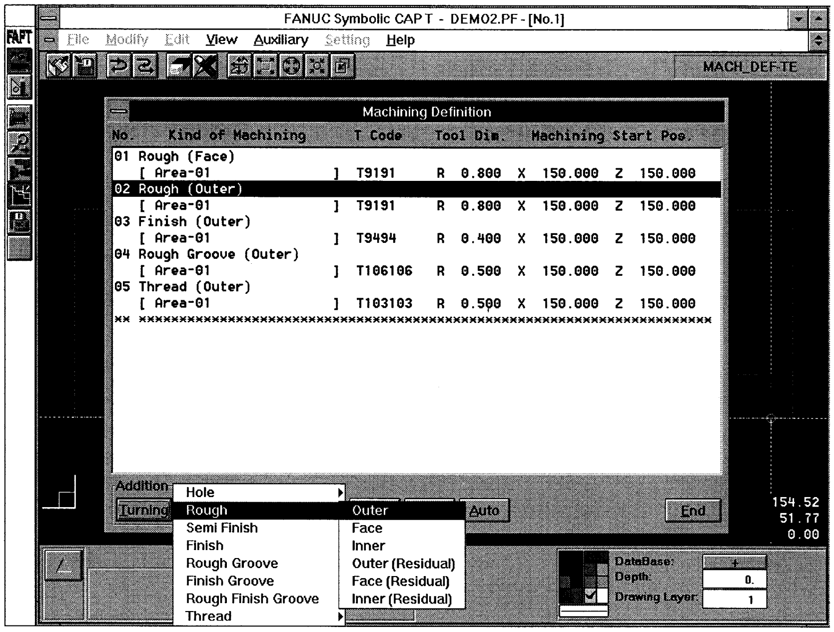
Min-Max Points

DataBase: +
Depth: 0.
Drawing Layer: 1

3.48
-22.14
0.00

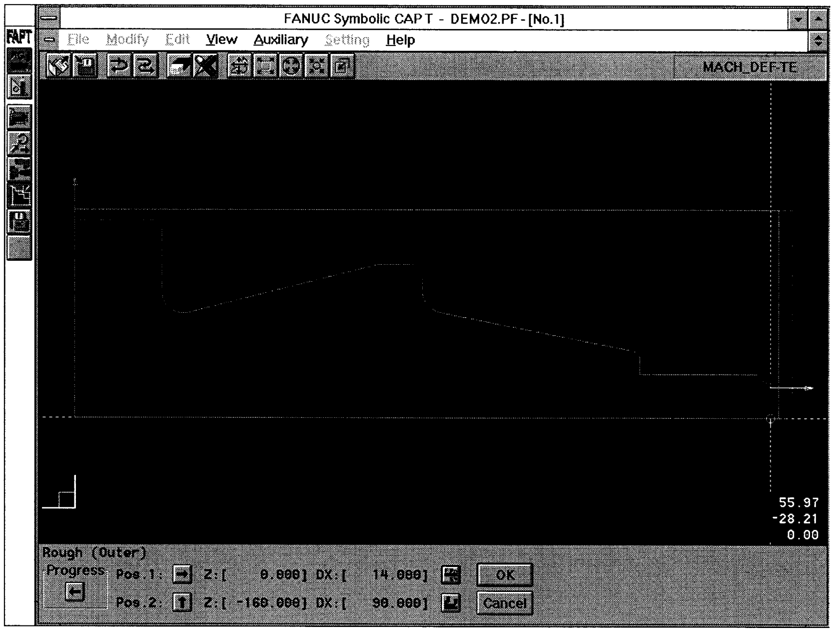
Machining definition

- The automatic process determination function automatically creates the processes shown below.
- The automatic process determination function automatically determines the machining type, tool data, cutting area, and cutting conditions.
- The operator can easily modify the automatically determined machining type, tool data, cutting area, or cutting conditions, or manually create processes, using the screen shown below.



```
No.    Kind of Machining        T Code    Tool Dim.   Machining Start Pos.
01 Rough (Face)
   [ Area-01                ]  T9191     R  0.800  X  150.000  Z  150.000
02 Rough (Outer)
   [ Area-01                ]  T9191     R  0.800  X  150.000  Z  150.000
03 Finish (Outer)
   [ Area-01                ]  T9494     R  0.400  X  150.000  Z  150.000
04 Rough Groove (Outer)
   [ Area-01                ]  T106106   R  0.500  X  150.000  Z  150.000
05 Thread (Outer)
   [ Area-01                ]  T103103   R  0.500  X  150.000  Z  150.000
** ********************************************************************
```

- The specified cutting area is drawn as shown below, thus allowing the area to be checked visually.



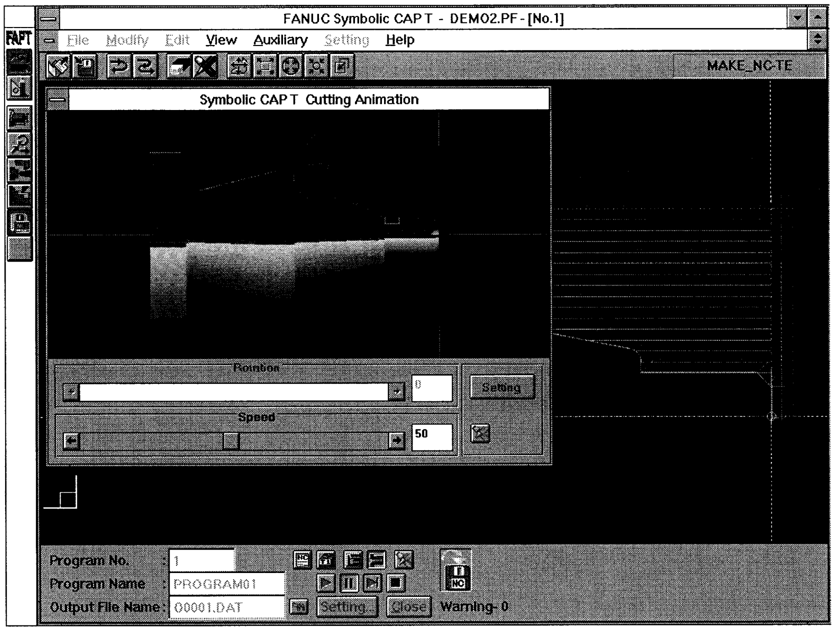
Rough (Outer)

Pos.1: Z:[0.000] DX:[14.000]

Pos.2: Z:[-160.000] DX:[90.000]

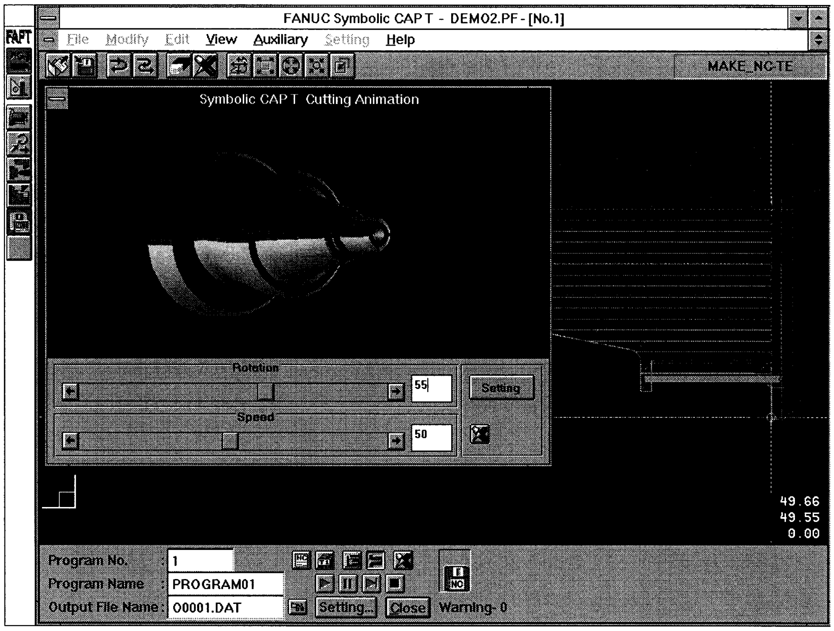
NC data preparation

- A tool path can be created without redundancy.
- The format of the NC data can be customized using an NC machine file.
- Three–dimensional animated machining simulation and tool path drawing allow the operator to observe the progress of the machining.



Rotation: 0
Speed: 50
Program No. : 1
Program Name : PROGRAM01
Output File Name : O0001.DAT
Warning- 0

- The completed part figure can be checked from its end surface, by rotating the workpiece.



Rotation: 55
Speed: 50
49.66
49.55
0.00
Program No. : 1
Program Name : PROGRAM01
Output File Name : O0001.DAT
Warning- 0

3 CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION FOR MACHINING CENTERS

Super CAP M and Super CAP II M are provided as the conversational automatic programming function for machining centers. As with Super CAP T, Super CAP M and Super CAP II M use a machining process selection method for input.

Super CAP II M is a development of conventional Super CAP M.
It features greatly improved display screens that use VGA graphics, while its operation is simple, in the same way as Super CAP M.

3.1 FEATURES

Super CAP M and Super CAP II M are performed by using conversational control software and a custom macro program. In Super CAP M and Super CAP II M, therefore, many macro instructions dedicated to conversational operation are provided as well as the standard custom macro instructions. FANUC supplies a standard macro library to the machine tool builder. The machine tool builder can make additions and modifications to the standard macro library to develop unique functions. The following sections explain the specifications of the conversational automatic programming function of the standard macro library supplied by FANUC.

- **About Super CAP II M**

Super CAP II M is a development of Super CAP M. It features the following new functions:

(1) Background color for the display screen; 3–D frames for the windows and soft keys.

(2) Finer and faster machining simulation based on a solid model.

(3) Cross–sectional display of a product based on a solid model.

(4) Selectable screen colors; up to four color schemes can be registered

(5) Improved visual interface for contour machining, with display of the contour figure during contour programming.

All other functions, such as screen displays, key operations, and machining functions, are the same as those of super CAP M.

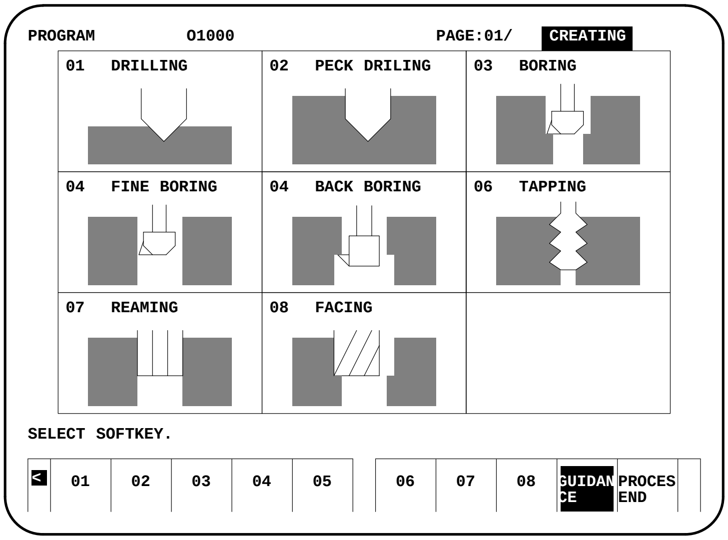
Moreover, machining programs, tool data, and conversational function parameters created with Super CAP M can also be used with Super CAP II M.

3.2
OUTLINE OF THE MACRO LIBRARY

Machining type

In Super CAP M and Super CAP II M, the following machining types can be selected:

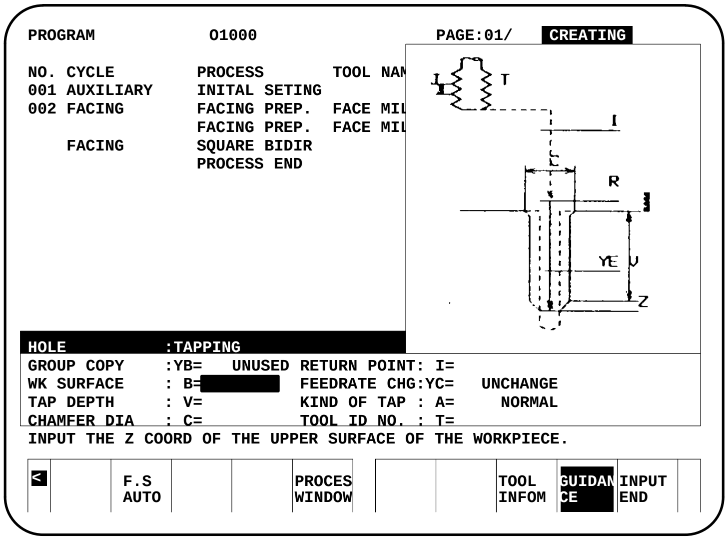
- Drilling (eight types + hole position menu: Nine types)
- Facing (six types)
- Side facing (eight types including contour side)
- Pocketing and grooving (eight types including contour pocketing)
- 2 + 1/2 machining (eight types)
- NC language (eight types)
- Machining of multiple workpieces (five types)
- U axis machining (eight types)
  Following each screens are displayed in Super CAP M.
  For Super CAP II M some screens are different.



```
PROGRAM         O1000                     PAGE:01/  CREATING

SELECT SOFTKEY.
```

Conversational program input screen

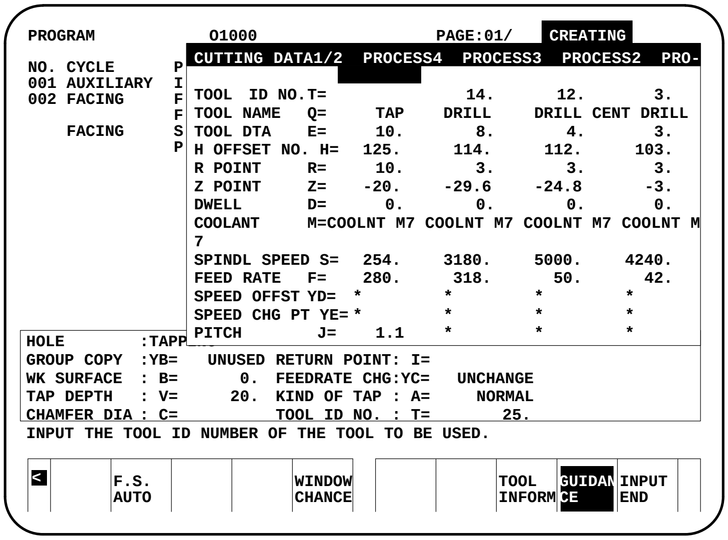
An easy–to–understand guide figure and message are displayed for each input item on the screen.



```
PROGRAM           O1000                   PAGE:01/  CREATING

NO. CYCLE        PROCESS       TOOL NAM
001 AUXILIARY    INITAL SETING
002 FACING       FACING PREP.  FACE MIL
                 FACING PREP.  FACE MIL
    FACING       SQUARE BIDIR
                 PROCESS END

HOLE           :TAPPING
GROUP COPY     :YB=   UNUSED RETURN POINT: I=
WK SURFACE     : B=          FEEDRATE CHG:YC=   UNCHANGE
TAP DEPTH      : V=          KIND OF TAP : A=     NORMAL
CHAMFER DIA    : C=          TOOL ID NO. : T=
INPUT THE Z COORD OF THE UPPER SURFACE OF THE WORKPIECE.
```

Tool/cutting condition/ pre–tool automatic determination

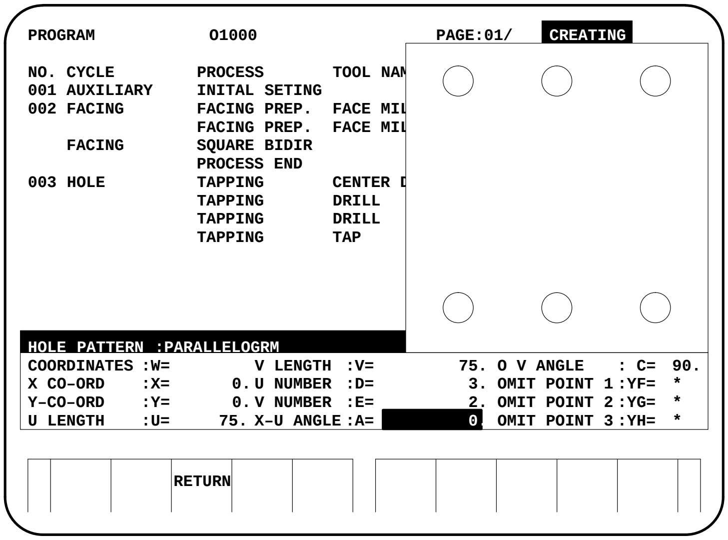
Tool data, cutting condition data, and pre–tool data can be input easily by following the instructions displayed on the screen.
Once data is input, necessary data for machining is determined automatically.



```
PROGRAM            O1000                  PAGE:01/   CREATING
                  CUTTING DATA1/2  PROCESS4  PROCESS3  PROCESS2  PRO-
NO. CYCLE       P
001 AUXILIARY   I
002 FACING      F TOOL  ID NO.T=                14.       12.       3.
                F TOOL NAME   Q=     TAP      DRILL     DRILL CENT DRILL
    FACING      S TOOL DTA    E=     10.         8.        4.       3.
                P H OFFSET NO. H=  125.       114.      112.      103.
                  R POINT     R=     10.         3.        3.       3.
                  Z POINT     Z=    -20.     -29.6     -24.8      -3.
                  DWELL       D=      0.         0.        0.       0.
                  COOLANT     M=COOLNT M7 COOLNT M7 COOLNT M7 COOLNT M
                  7
                  SPINDL SPEED S=  254.      3180.     5000.     4240.
                  FEED RATE   F=    280.       318.       50.      42.
                  SPEED OFFST YD=  *         *         *         *
                  SPEED CHG PT YE= *         *         *         *
HOLE         :TAPP PITCH       J=    1.1     *         *         *
GROUP COPY  :YB=   UNUSED RETURN POINT: I=
WK SURFACE  : B=       0.  FEEDRATE CHG:YC=   UNCHANGE
TAP DEPTH   : V=      20.  KIND OF TAP : A=     NORMAL
CHAMFER DIA : C=           TOOL ID NO. : T=       25.
INPUT THE TOOL ID NUMBER OF THE TOOL TO BE USED.
```

Immediate checking of input data

Input data is indicated graphically, and so checking can be done immediately. If a data value exceeds an allowable limit, an alarm message appears on the screen and the cursor appears at the position of the data in question.

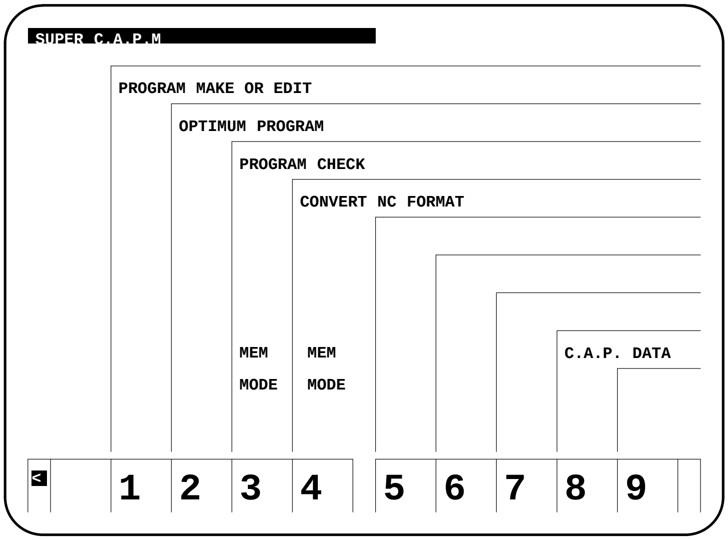


```
PROGRAM            O1000                  PAGE:01/   CREATING

NO. CYCLE          PROCESS       TOOL NAM
001 AUXILIARY      INITAL SETING
002 FACING         FACING PREP.  FACE MIL
                   FACING PREP.  FACE MIL
    FACING         SQUARE BIDIR
                   PROCESS END
003 HOLE           TAPPING       CENTER D
                   TAPPING       DRILL
                   TAPPING       DRILL
                   TAPPING       TAP

HOLE PATTERN  :PARALLELOGRM
COORDINATES :W=         V LENGTH  :V=          75. O V ANGLE    : C=  90.
X CO-ORD    :X=       0. U NUMBER  :D=           3. OMIT POINT 1 :YF=  *
Y-CO-ORD    :Y=       0. V NUMBER  :E=           2. OMIT POINT 2 :YG=  *
U LENGTH    :U=      75. X-U ANGLE :A=           0  OMIT POINT 3 :YH=  *
```

3.3
OUTLINE OF THE CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION

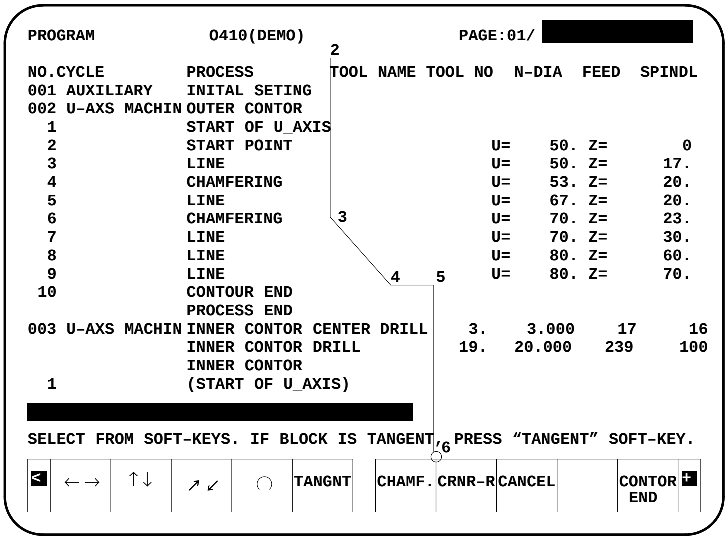
Basic menu screen

Operations with Super CAP M and Super CAP II M always begin with the following basic menu screen. When the leftmost soft key ◁ on a conversational screen is pressed, the display is returned to the previous screen.



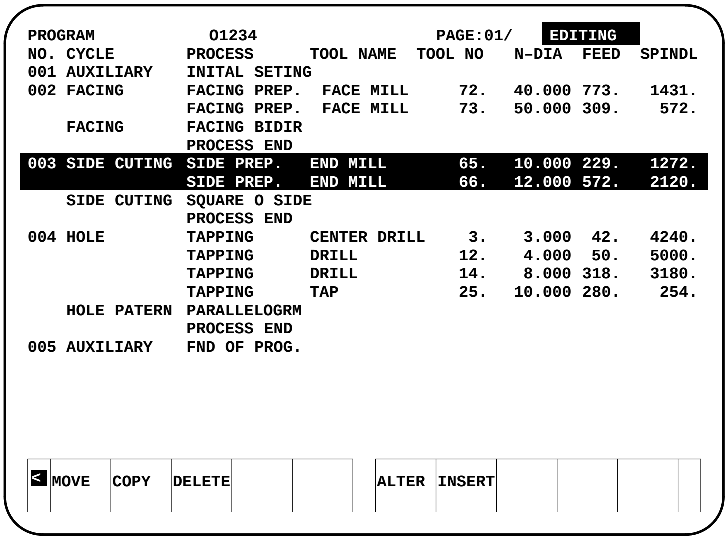
Conversational program input screen (inputting contours)

Even a complicated machining profile can be input easily by using the symbolic input and automatic intersection calculation functions.
The input profile is displayed directly on the screen so that the user can easily check the profile.


```
PROGRAM          O410(DEMO)                PAGE:01/

NO.CYCLE        PROCESS        TOOL NAME TOOL NO  N-DIA  FEED  SPINDL
001 AUXILIARY   INITAL SETING
002 U-AXS MACHIN OUTER CONTOR
  1             START OF U_AXIS
  2             START POINT                  U=    50. Z=       0
  3             LINE                         U=    50. Z=     17.
  4             CHAMFERING                   U=    53. Z=     20.
  5             LINE                         U=    67. Z=     20.
  6             CHAMFERING                   U=    70. Z=     23.
  7             LINE                         U=    70. Z=     30.
  8             LINE                         U=    80. Z=     60.
  9             LINE                         U=    80. Z=     70.
 10             CONTOUR END
                PROCESS END
003 U-AXS MACHIN INNER CONTOR CENTER DRILL    3.    3.000    17     16
                INNER CONTOR DRILL          19.   20.000   239    100
                INNER CONTOR
  1             (START OF U_AXIS)

SELECT FROM SOFT-KEYS. IF BLOCK IS TANGENT, PRESS “TANGENT” SOFT-KEY.
```

Conversational program editing screen

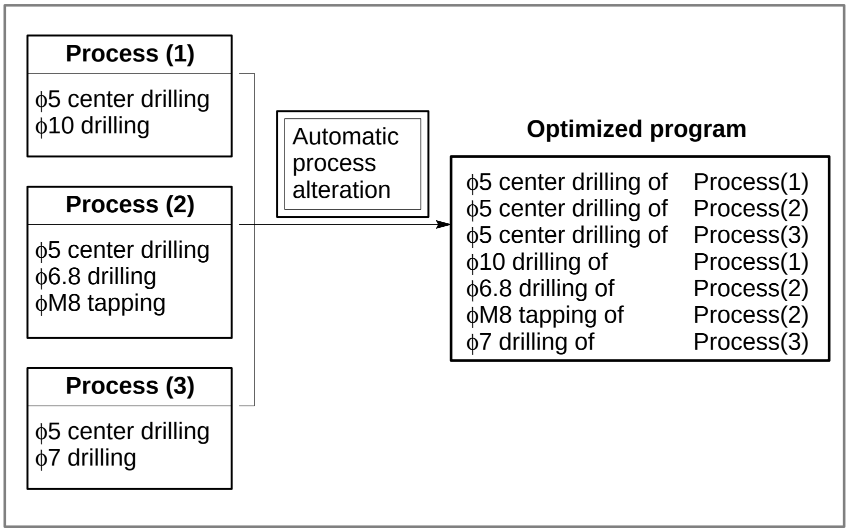
Programmed data is listed in an easy–to–understand form.
On this screen, editing operations such as movement, copy, and deletion of processes can be performed.



```
PROGRAM           O1234                  PAGE:01/   EDITING
NO. CYCLE         PROCESS      TOOL NAME  TOOL NO   N-DIA  FEED  SPINDL
001 AUXILIARY     INITAL SETING
002 FACING        FACING PREP.  FACE MILL     72.   40.000 773.   1431.
                  FACING PREP.  FACE MILL     73.   50.000 309.    572.
    FACING        FACING BIDIR
                  PROCESS END
003 SIDE CUTING   SIDE PREP.   END MILL       65.   10.000 229.   1272.
                  SIDE PREP.   END MILL       66.   12.000 572.   2120.
    SIDE CUTING   SQUARE O SIDE
                  PROCESS END
004 HOLE          TAPPING      CENTER DRILL    3.    3.000  42.   4240.
                  TAPPING      DRILL          12.    4.000  50.   5000.
                  TAPPING      DRILL          14.    8.000 318.   3180.
                  TAPPING      TAP            25.   10.000 280.    254.
    HOLE PATERN   PARALLELOGRM
                  PROCESS END
005 AUXILIARY     FND OF PROG.
```

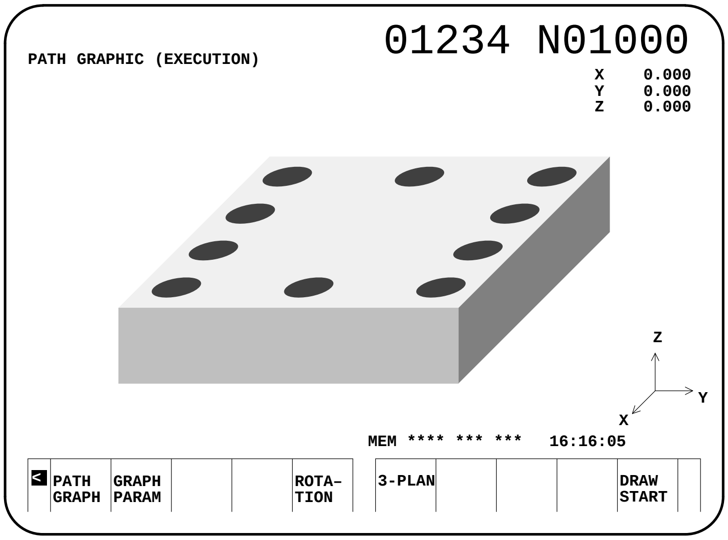
Process optimization edit function

The machining order can be changed automatically to reduce the number of times tools are changed. The machining order can also be specified manually.
With these functions, the time required for machining can be reduced.



Full graphic function

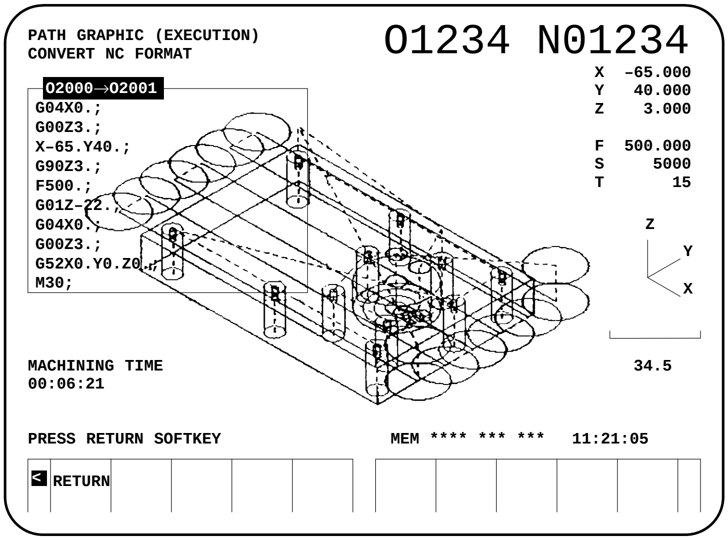
Machining profiles, tool figures, and tool paths can be drawn in the isometric mode, biplane drawing mode, and so forth.
In addition, an animated simulation function is provided to display a solid drawing of the workpiece figure to be machined.



3.4 OTHER OPTIONAL FUNCTIONS

NC program output function

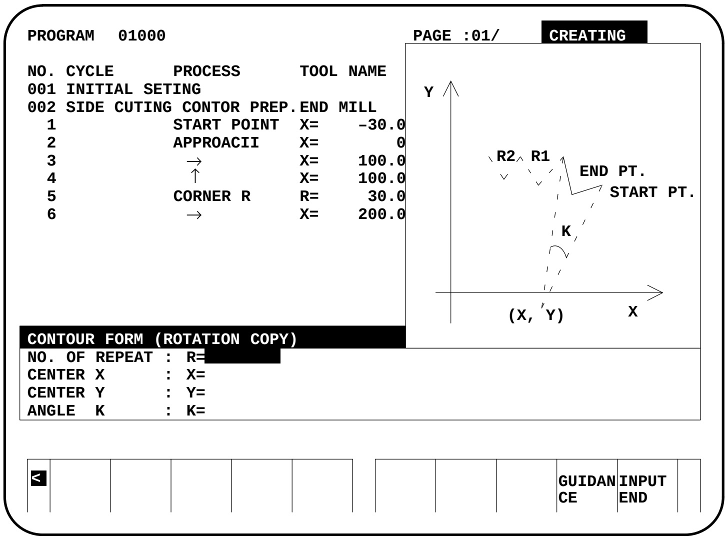
A machining program created conversationally can be run directly. The program can also be converted and then executed as an NC program form. Furthermore, when modifications are made to the NC program obtained by the conversion, a more efficient machining program can be created.



```
PATH GRAPHIC (EXECUTION)              O1234 N01234
CONVERT NC FORMAT
                                                   X  -65.000
 O2000→O2001                                       Y   40.000
 G04X0.;                                           Z    3.000
 G00Z3.;
 X-65.Y40.;                                        F  500.000
 G90Z3.;                                           S     5000
 F500.;                                            T       15
 G01Z-22.;
 G04X0.;
 G00Z3.;
 G52X0.Y0.Z0.;
 M30;

MACHINING TIME                                        34.5
00:06:21

PRESS RETURN SOFTKEY             MEM **** *** ***   11:21:05
```

Contour repeat function

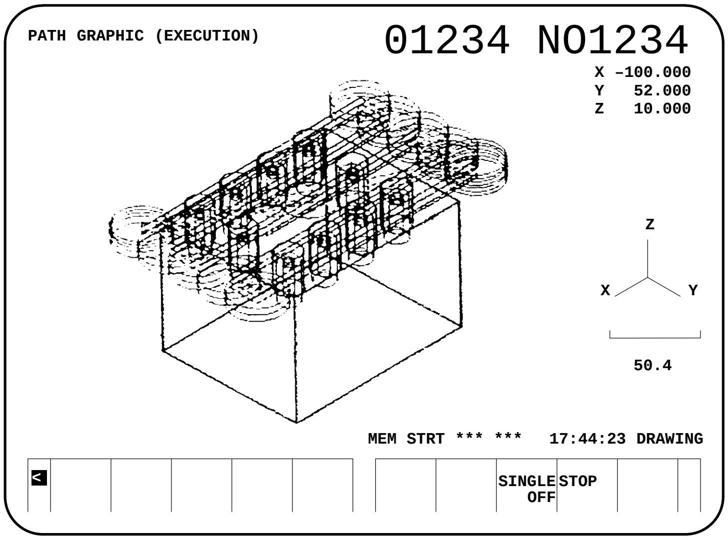
In programming for an arbitrary figure, a certain section of a figure can be repeated more than once. Three types of repetitions are available: Inversion, rotation, and parallel displacement. By combining these types, program data can be utilized more than once in a program.



```
PROGRAM   01000                        PAGE :01/    CREATING

NO. CYCLE      PROCESS      TOOL NAME
001 INITIAL SETING
002 SIDE CUTING CONTOR PREP.END MILL
  1            START POINT  X=    -30.0
  2            APPROACII    X=        0
  3              →          X=    100.0
  4              ↑          X=    100.0
  5            CORNER R     R=     30.0
  6              →          X=    200.0

CONTOUR FORM (ROTATION COPY)
NO. OF REPEAT :  R=
CENTER X      :  X=
CENTER Y      :  Y=
ANGLE  K      :  K=
```

Background drawing function

A machining program can be created and checked using the drawing function while another program is being executed for machining.
By using this function, the NC can be used more efficiently.



PATH GRAPHIC (EXECUTION) O1234 NO1234

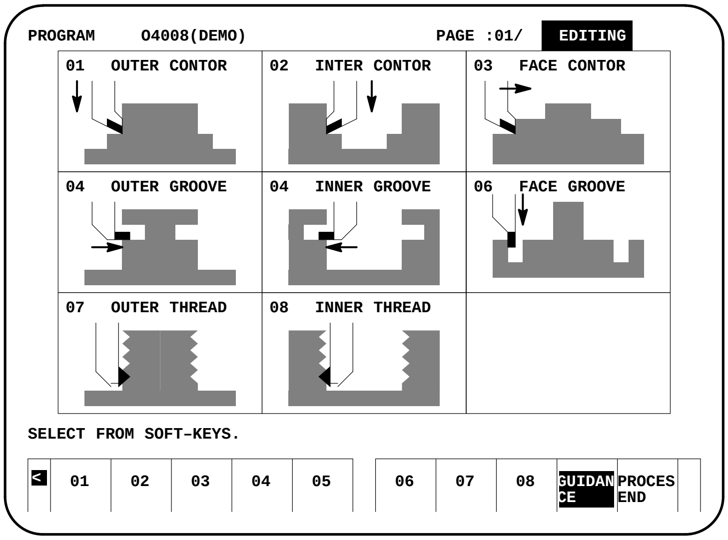
X –100.000
Y 52.000
Z 10.000

MEM STRT *** *** 17:44:23 DRAWING

U–axis conversational function

Programs for the following cutting operations with the lathe can be input conversationally:

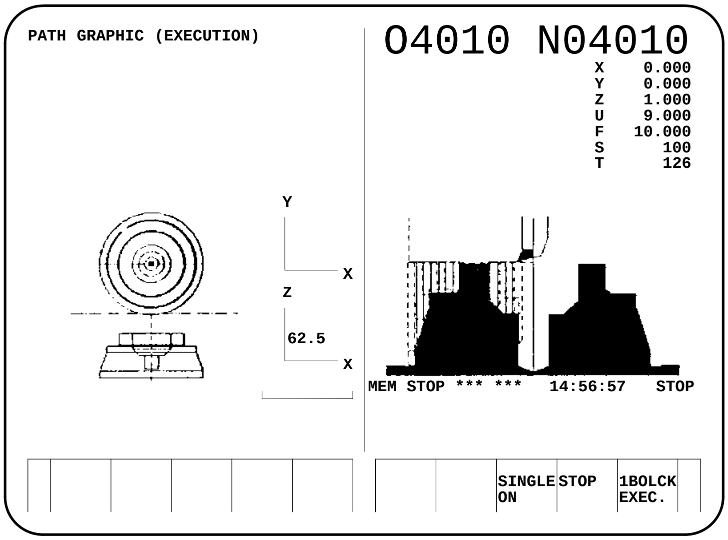
- Contouring (outer surface/inner surface/end face)
- Grooving (outer surface/inner surface/end face)
- Threading (external/internal)



PROGRAM O4008(DEMO) PAGE :01/ EDITING

SELECT FROM SOFT–KEYS.

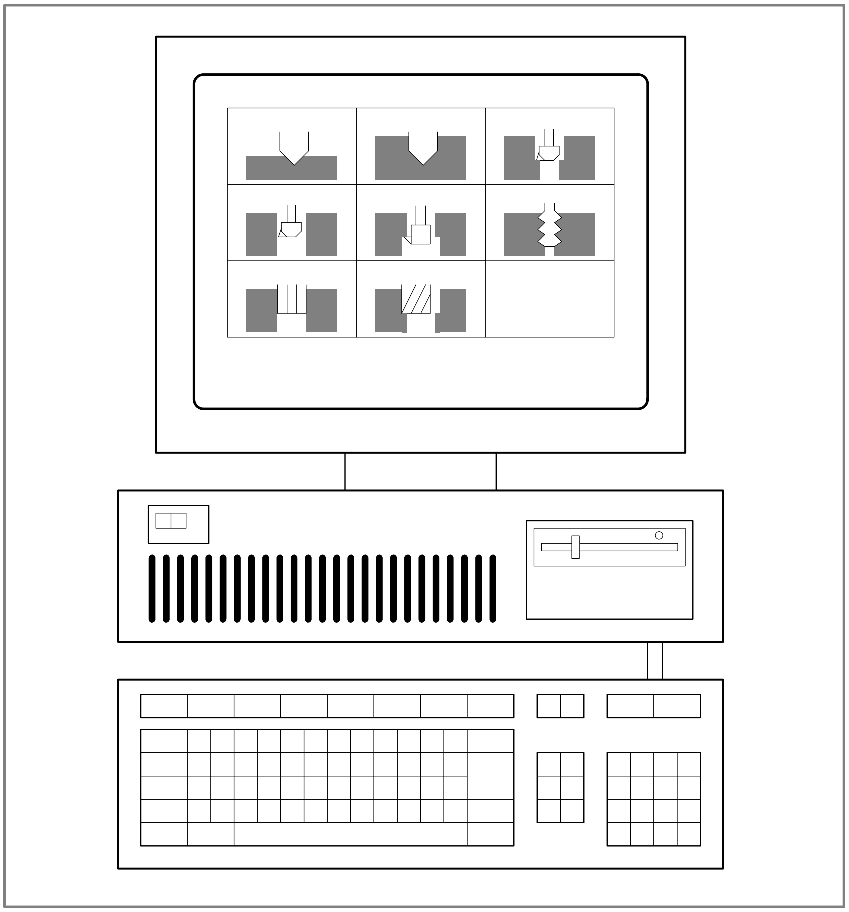
In addition, simulation for the machining profile, removal, tool path, and others functions are enabled during cutting.



```
PATH GRAPHIC (EXECUTION)                O4010 N04010
                                              X     0.000
                                              Y     0.000
                                              Z     1.000
                                              U     9.000
                                              F    10.000
                                              S       100
                                              T       126
MEM STOP *** ***    14:56:57    STOP
```

Conversational C language programming function

When a custom macro program is replaced with a program coded in C, programs such as those for automatic tool setting and automatic cutting condition setting can be executed at high–speed.
Programs are developed on a personal computer.



3.5
SYMBOLIC CAP M

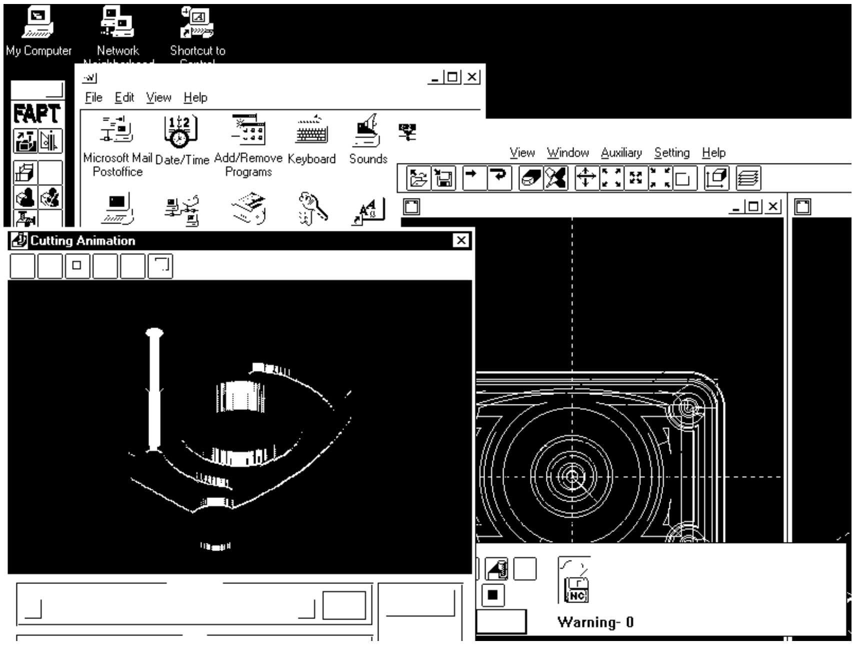
Features

Symbolic CAP M is conversational automatic programming software for machining centers. It has the following features:

- **User–friendly operation that is easy to use, even by beginners**
  Symbolic CAP M is easy even for beginners to use, thanks to its support of graphical menus (icons) and mouse–driven conversational processing. In addition, it features a large library of help messages.
- **Operating procedure for experienced operators**
  Symbolic CAP M commands can be entered directly from the keyboard, enabling experienced operators to operate the system quickly.
- **Customizing**
  Symbolic CAP M can be customized. It is possible to assign a frequently used function to a tool bar that is constantly displayed on the screen, allowing that function to be called merely by clicking the corresponding button.
- **Symbolic figure input**
  Blank and part figures can be entered using conventional symbolic figure input based on arrows.
- **Fully automatic process determination**
  The optimum machining type, tool, and machining conditions can be automatically selected simply by entering blank and part figures. Thus, even beginners can create machining programs quickly and easily.
- **Machining simulation**
  Three–dimensional animated machining simulation and tool path drawing are performed simultaneously. Animated machining simulation allows the operator to observe a simulation of the machining progress. Tool path drawing enables the operator to check the tool path generated by a machining program in detail.

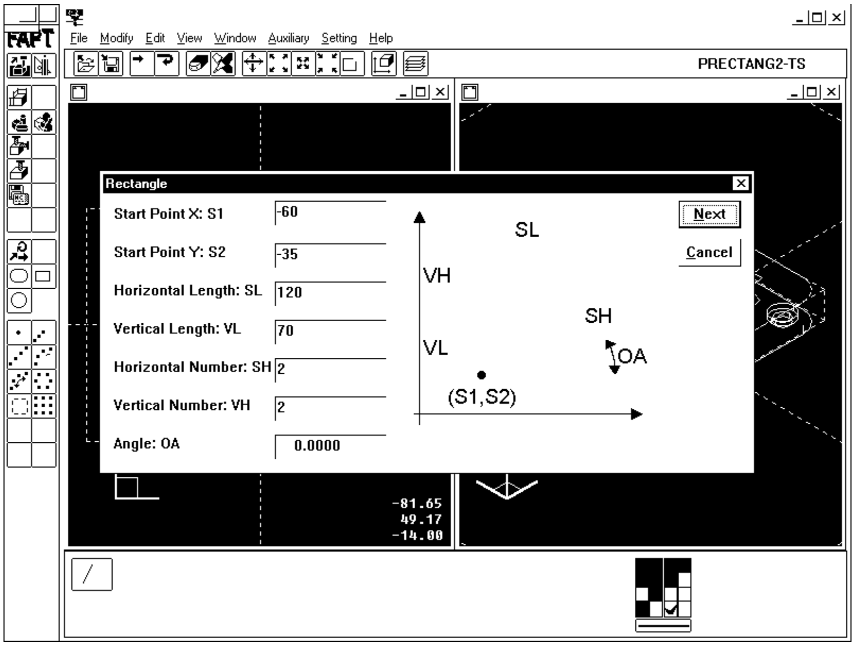
Windows® compatibility

Symbolic CAP M can run under Windows®, thus offering a user–friendly operating environment with icons, graphics, and windows.



Powerful figure input function

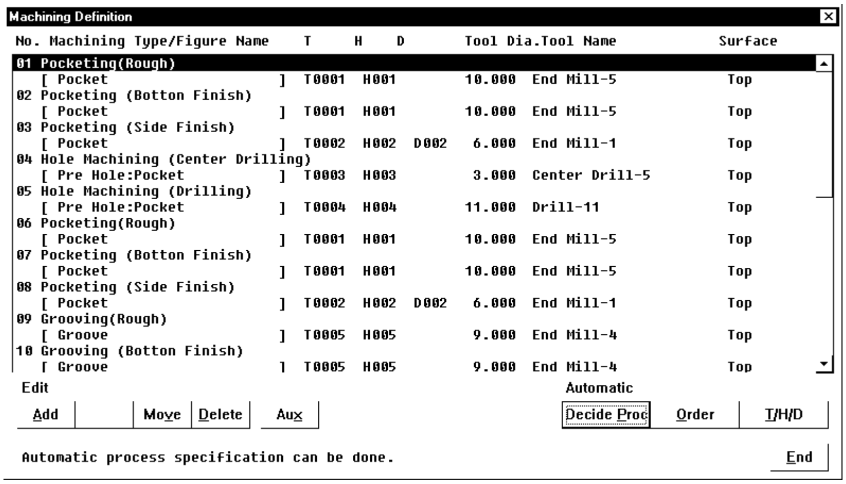
Figures can be programmed easily, by means of symbolic input for contour figures and menu–based conversational input for pattern figures such as holes. CAD data can also be input as figures.



Start Point X: S1 -60
Start Point Y: S2 -35
Horizontal Length: SL 120
Vertical Length: VL 70
Horizontal Number: SH 2
Vertical Number: VH 2
Angle: OA 0.0000

Fully automatic process determination

An optimum machining process can be automatically determined simply by entering blank and part figures.

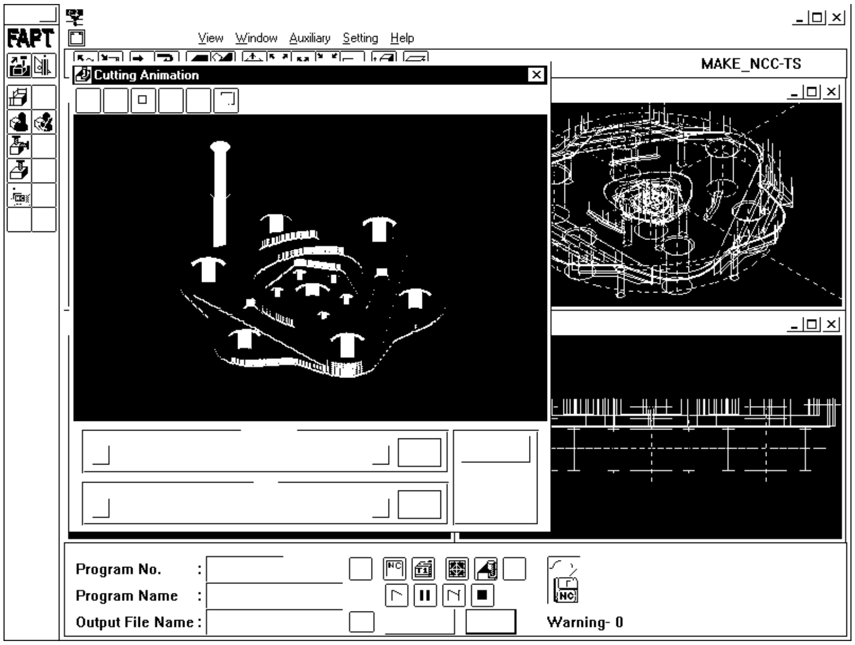


```
No. Machining Type/Figure Name     T      H     D        Tool Dia.Tool Name            Surface
01 Pocketing(Rough)
   [ Pocket                   ]  T0001  H001          10.000  End Mill-5             Top
02 Pocketing (Botton Finish)
   [ Pocket                   ]  T0001  H001          10.000  End Mill-5             Top
03 Pocketing (Side Finish)
   [ Pocket                   ]  T0002  H002  D002     6.000  End Mill-1             Top
04 Hole Machining (Center Drilling)
   [ Pre Hole:Pocket          ]  T0003  H003           3.000  Center Drill-5         Top
05 Hole Machining (Drilling)
   [ Pre Hole:Pocket          ]  T0004  H004          11.000  Drill-11               Top
06 Pocketing(Rough)
   [ Pocket                   ]  T0001  H001          10.000  End Mill-5             Top
07 Pocketing (Botton Finish)
   [ Pocket                   ]  T0001  H001          10.000  End Mill-5             Top
08 Pocketing (Side Finish)
   [ Pocket                   ]  T0002  H002  D002     6.000  End Mill-1             Top
09 Grooving(Rough)
   [ Groove                   ]  T0005  H005           9.000  End Mill-4             Top
10 Grooving (Botton Finish)
   [ Groove                   ]  T0005  H005           9.000  End Mill-4             Top
```

Automatic process specification can be done.

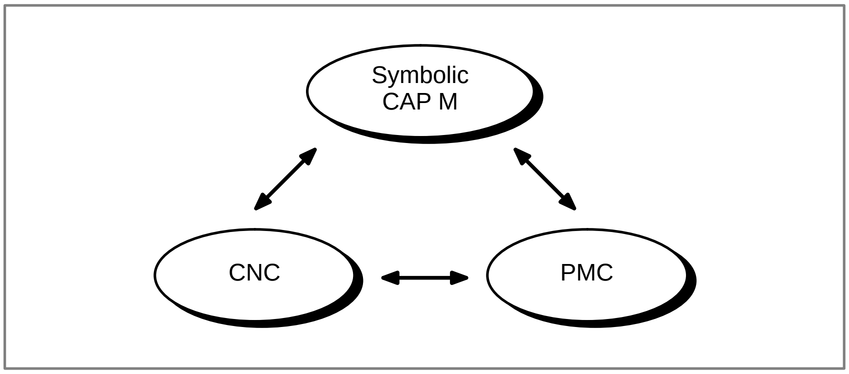
Real machining simulation

High–speed real machining simulation, using solid models, can be performed based on a created machining program, enabling the contents of the program to be checked easily.



NC connection module (development scheduled)

A large amount of data can be transferred to and from the CNC or PMC via an open data bus. This function is used to enable the direct manipulation of conversational programs and setup operations such as the measurement of tool offsets.



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 Y : 0.001 mm	X : 0.00005 mm Y : 0.0001 mm
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 Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
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 Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
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 Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
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
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
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, 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 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. 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, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step

NOTE

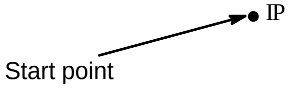
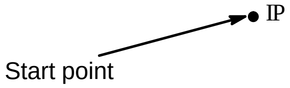
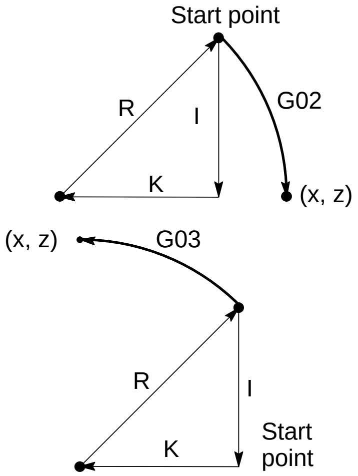
*1 The feedrate range shown above are limitations depending on CNC interpolation capacity.
As a whole system, limitations depending on servo system must also be considered.

B FUNCTIONS AND TAPE FORMAT LIST

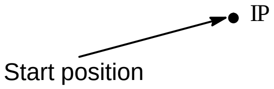
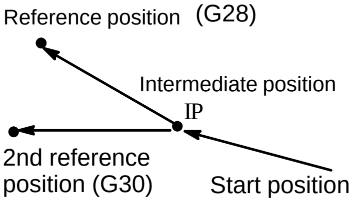
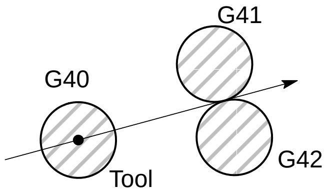
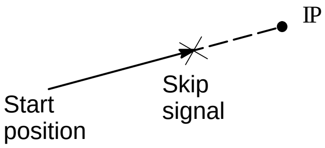
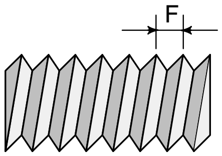
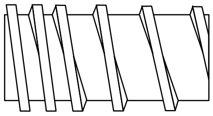
B.1 T SERIES

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)

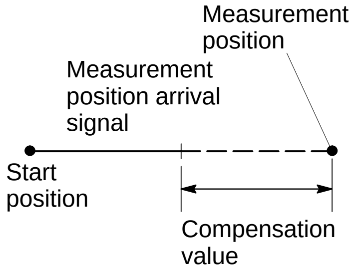
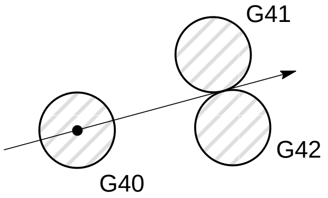
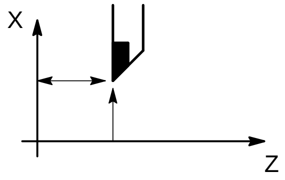
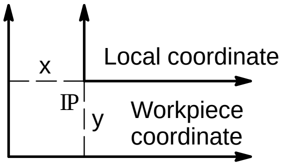
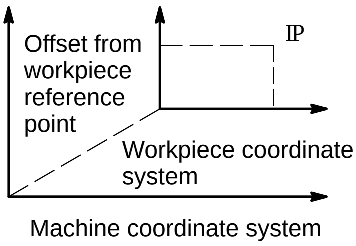
(1/5)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_\ Y\_\begin{Bmatrix}R\_\\I\_\ J\_\end{Bmatrix}F\_\ ;$ $G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_\ Z\_\begin{Bmatrix}R\_\\I\_\ K\_\end{Bmatrix}F\_\ ;$ $G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_\ Z\_\begin{Bmatrix}R\_\\J\_\ K\_\end{Bmatrix}F\_\ ;$
Dwell (G04)		$G04\begin{Bmatrix}X\_\\P\_\end{Bmatrix}\ ;$
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_ Z_ R_ Q_ ; P=1000+Geometry offset number Tool wear offset value G10 P_ X_ Z_ R_ Q_ ; P=Wear offset number

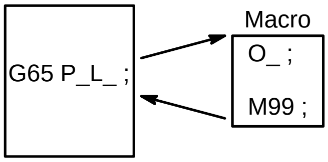
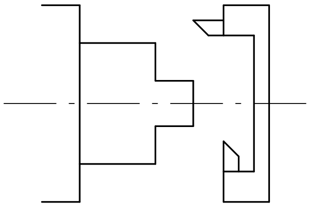
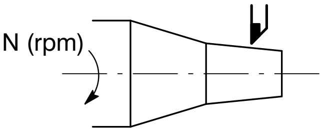
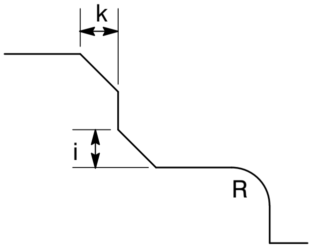
(2/5)

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 return (G30)		G28 IP_ ; G30 IP_ ;
Cutter compensation (G40, G41, G42)		$\left\{\begin{matrix}G41\\G42\end{matrix}\right\}$ P_ ; P : Tool offset number G40 : Cancel
Skip function (G31)		G31 IP_ F_;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_;
Variable–lead threading		G34 IP_ F_K_;

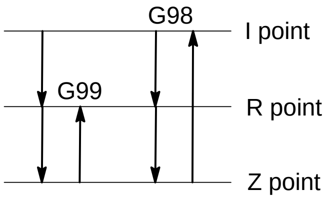
(3/5)

Functions	**Illustration**	**Tape format**
Automatic tool compensation (G36, G37)		G36 X xa ; G37 Z za ;
Tool–tip radius compensation (G40, G41, G42)		{ G41 / G42 } IP_ ; G40 : Cancel
Coordinate system setting Spindle speed setting (G50)		G50 X_ Z_ ; Coordinate system setting G50 S_ ; 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)		{ G54 / : / G59 } IP _ ;

(4/5)

Functions	Illustration	Tape format
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
Mirror image for double turret (G68, G69)		G68 ; Mirror image for double turret on G69 ; Mirror image cancel
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_;
Feed per minute (G98) Feed per revolution (G99)	mm/min inch/min mm/rev inch/rev	G98 … F_ ; G99 … F_ ;
Constant surface speed control (G96/G97)	m/min or feet/min 	G96 S_ ; G97 ; Cancel
Chamfering, Corner R		$X\_\ ;\left\{\begin{matrix}C\pm k\\R\_\end{matrix}\right\}P\_\ ;$ $Z\_\ ;\left\{\begin{matrix}C\pm i\\R\_\end{matrix}\right\}P\_\ ;$

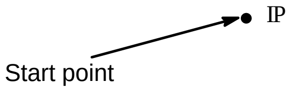
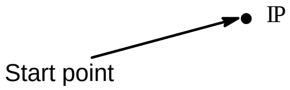
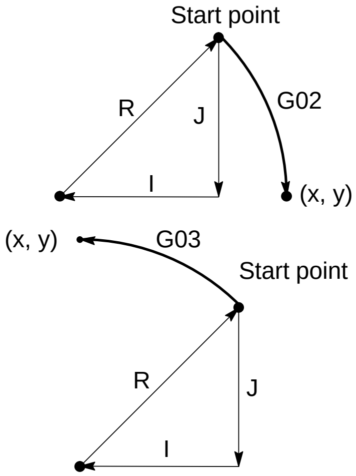
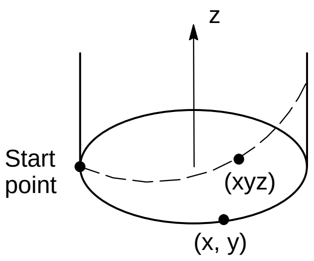
(5/5)

Functions	Illustration	Tape format
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.14. 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_ ; G94 X_ Z_ K_ F_ ; $\left\{\begin{matrix}G90\\G92\end{matrix}\right\}$ X_ Z_ I_ F_ ;
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_ ;

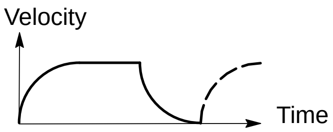
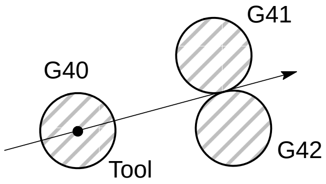
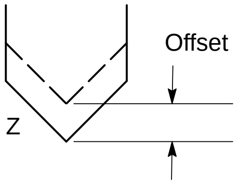
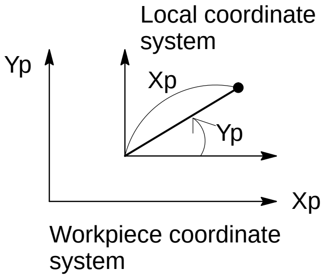
B.2
M SERIES

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)

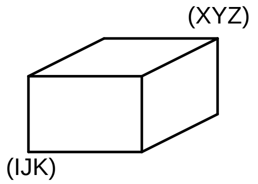
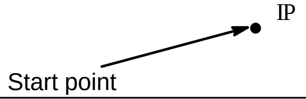
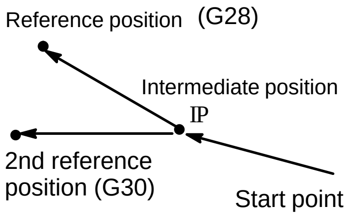
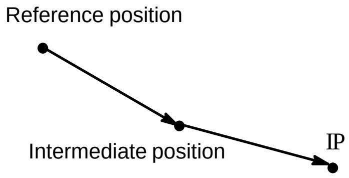
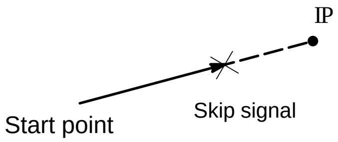
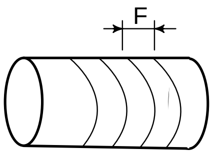
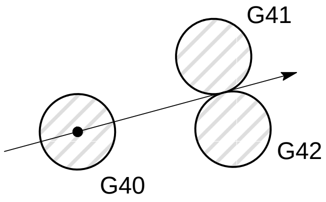
(1/6)

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	$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Y\_ \begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix} \alpha\_F\_ ;$ $G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Z\_ \begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix} \alpha\_F\_ ;$ $G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\ Z\_ \begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix} \alpha\_F\_ ;$ α: Any address other than that of a circular interpolation axis
Dwell (G04) (In case of X–Y plane)		$G04 \begin{Bmatrix} X\_ \\ P\_ \end{Bmatrix} ;$
Cylindrical interpolation (G07.1)		G07.1 IP_R_ ; Cylindrical interpolation mode R : Radius of cylinder G07.1 IP 0 ; Cylindrical interpolation mode cancel

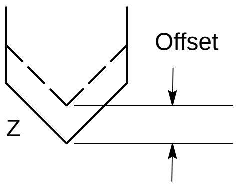
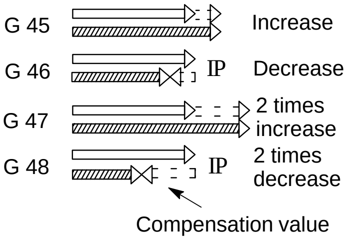
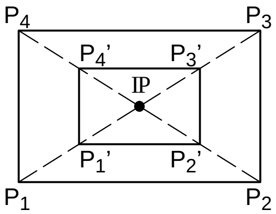
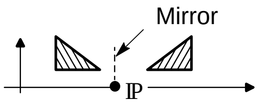
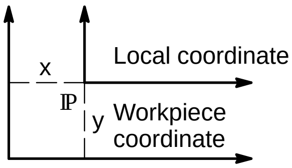
(2/6)

Functions	Illustration	Tape format
Look–ahead control (G08)		G08 P1 ; Look–ahead control mode on G08 P0 ; Look–ahead control mode off
Exact stop (G09)		G09 IP_;
Change of offset value by program (G10)		G10 P_R_;
Cutter compensation B (G39 – G42) Cutter compensation C (G40 – G42)		$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G41\\G42\end{matrix}\right\}$ H_ ; H : Tool offset G40 : Cancel
Tool length offset A (G43, G44, G49)		$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ Z_ H_ ; $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H_ ; H : Tool offset G49 : Cancel
Tool length offset B (G43, G44, G49)		$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G43\\G44\end{matrix}\right\}\left\{\begin{matrix}Z\_\\Y\_\\X\_\end{matrix}\right\}$ H_ ; $\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H_ ; H : Tool offset G49 : Cancel
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 ;

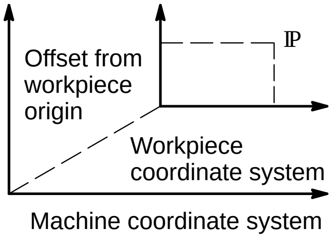
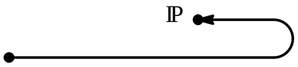
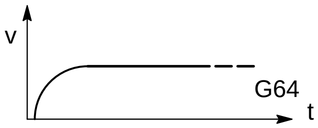
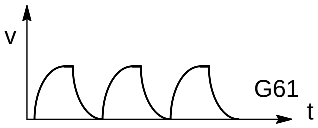
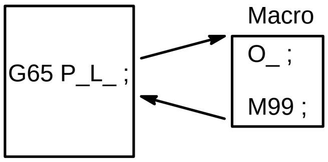
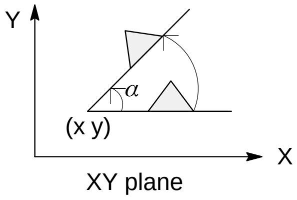
(3/6)

Functions	Illustration	Tape format
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_ ;
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)		$\begin{Bmatrix} G17 \\ G18 \\ G19 \end{Bmatrix} \begin{Bmatrix} G41 \\ G42 \end{Bmatrix}$ 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

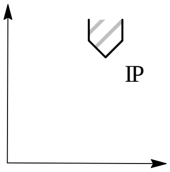
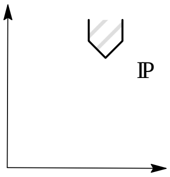
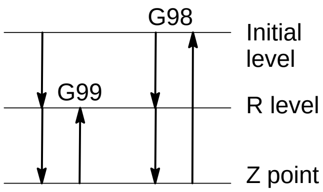
(4/6)

Functions	Illustration	Tape format
Tool length compensation A (G43, G44, G49)		$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ Z_ H_ ; $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H_ ; H : Tool offset number G49 : Cancel
Tool length compensation B (G43, G44, G49)		$\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G43\\G44\end{matrix}\right\}\left\{\begin{matrix}Z\_\\Y\_\\X\_\end{matrix}\right\}$ H_ ; $\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ H_ ; H : Tool offset number G49 : Cancel
Tool length compensation C (G43, G44, G49)		$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ α_H_ ; α : Any address of a single axis H : Tool offset number G49 : Cancel
Tool offset (G45 to G48)		$\left\{\begin{matrix}G45\\G46\\G47\\G48\end{matrix}\right\}$ IP _ D_ ; D : Tool offset number
Scaling (G50, G51)		G51 X_Y_Z_ $\left\{\begin{matrix}P\_\\I\_J\_K\_\end{matrix}\right\}$; 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_ ;

(5/6)

Functions	Illustration	Tape format
Machine coordinate system selection (G53)		G53 IP_ ;
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 $\underline{\alpha}$; 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 change (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	△	
–			○		○	◦	○		○	–		○				◦				Minus sign		
:			○	○	○	◦		○		___						◦				Colon (address O)		
/	○		○		○	◦	○	○	○	/			○	○		◦			○	Optional block skip		
.			○		○	◦	○	○		.		○	○		○	◦		○	○	Period (decimal point)		
#	○		○			◦		○	○	Parameter (No. 6012)						◦				Sharp		
$			○			◦	○			___						◦				Dollar sign	×	×
&	○		○			◦	○	○		&					○	◦	○	○		Ampersand	△	○
’			○			◦	○	○	○	___						◦				Apostrophe	△	△
*	○		○		○	◦		○		Parameter (No. 6010)						◦				Asterisk	△	
,	○		○		○	◦	○			,			○	○	○	◦		○	○	Comma		
;	○		○	○	○	◦		○	○	___						◦				Semicolon	×	×
<			○	○	○	◦	○									◦				Left angle bracket	△	△

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.

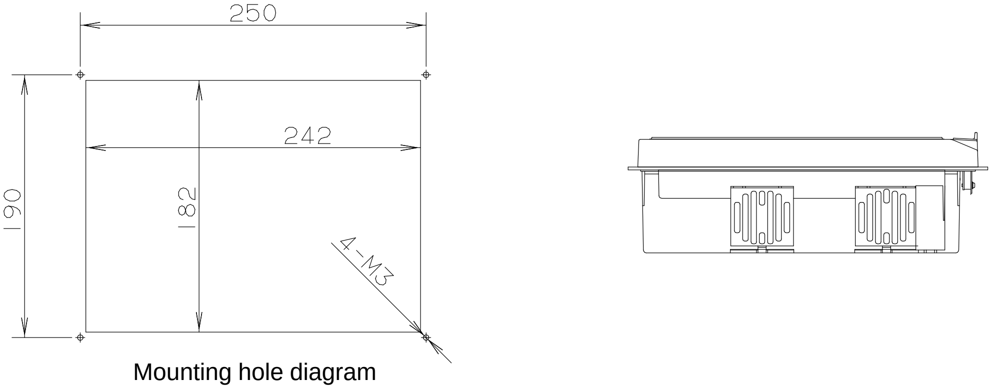
3 Any code having an invalid parity will cause a TH alarm. Within a comment, however, such a code is ignored and will not cause a TH alarm.

4 With EIA code, the code with all eight holes punched has special meaning. It is ignored and does not cause any parity alarm.

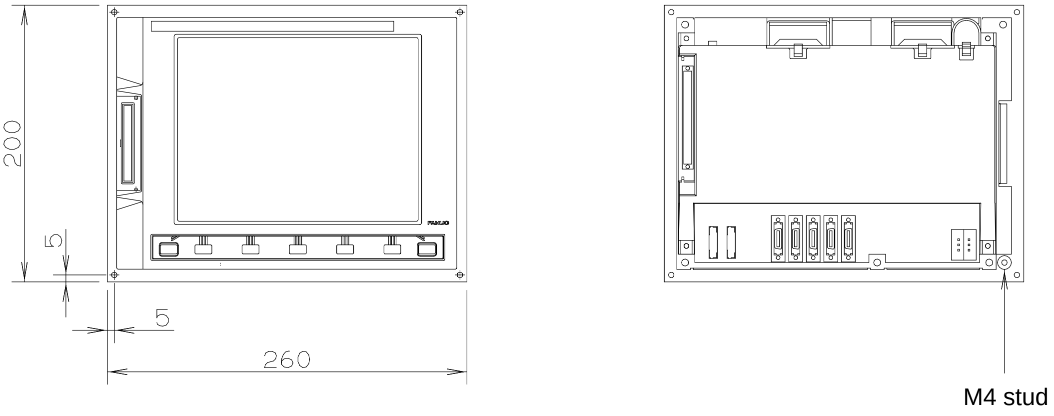
D EXTERNAL DIMENSIONS BASIC UNIT

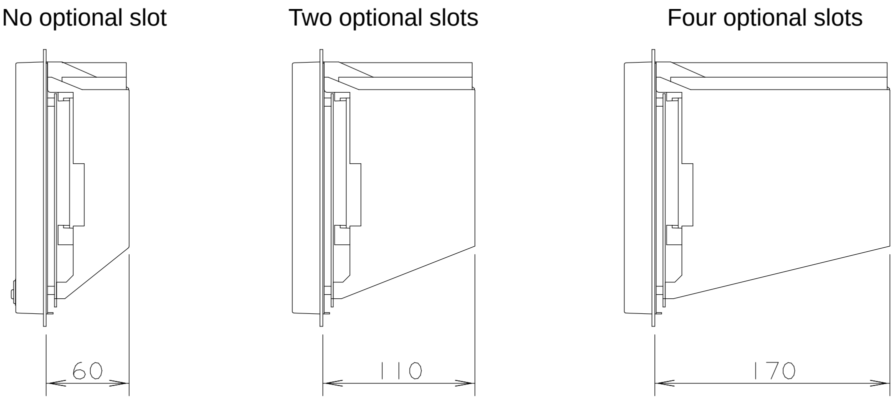
Fig. 1 EXTERNAL DIMENSIONS OF CNC CONTROL UNIT WITH 7.2″/8.4″ LCD

At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.



Mounting hole diagram





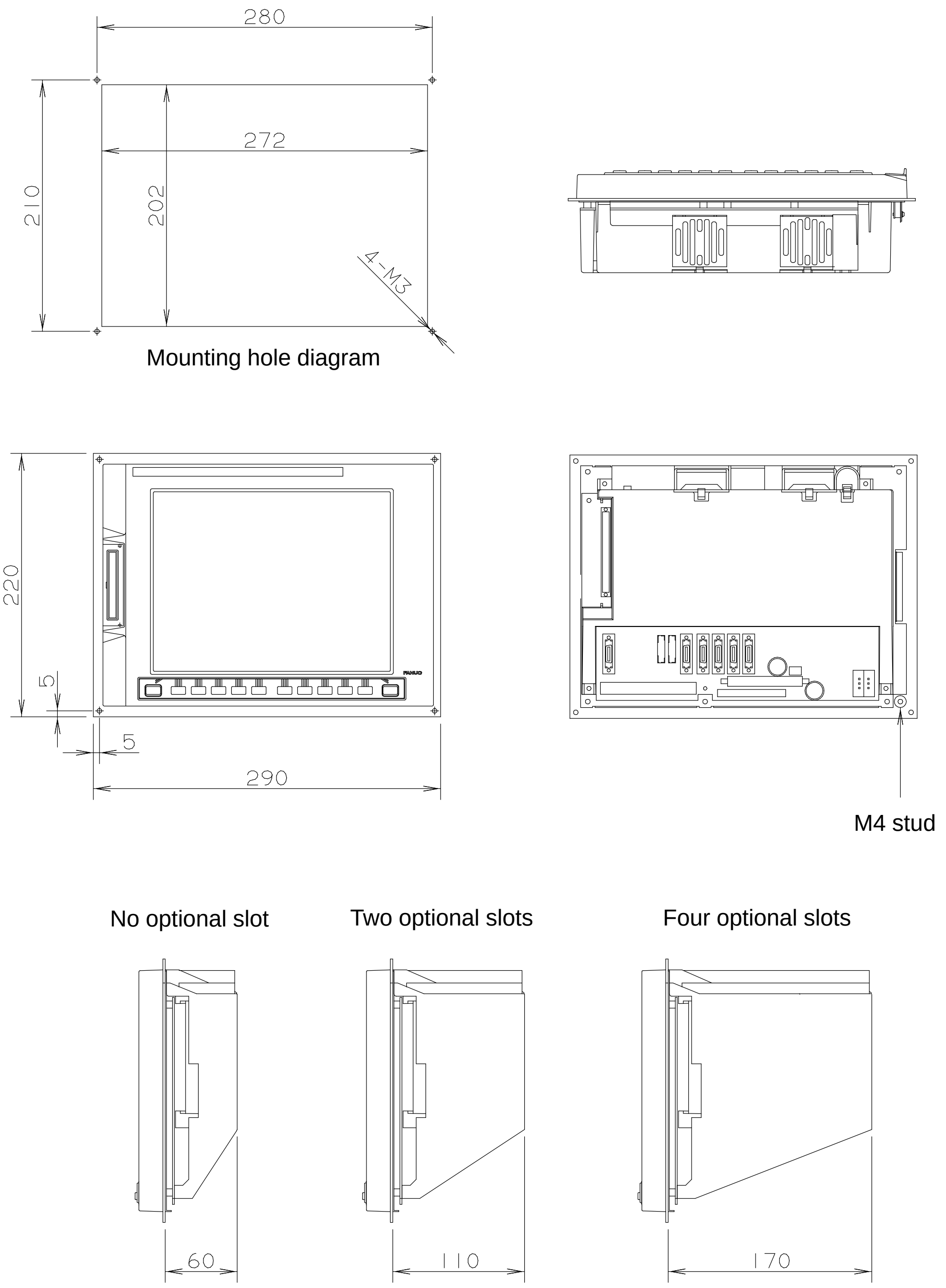
No optional slot

Two optional slots

Four optional slots

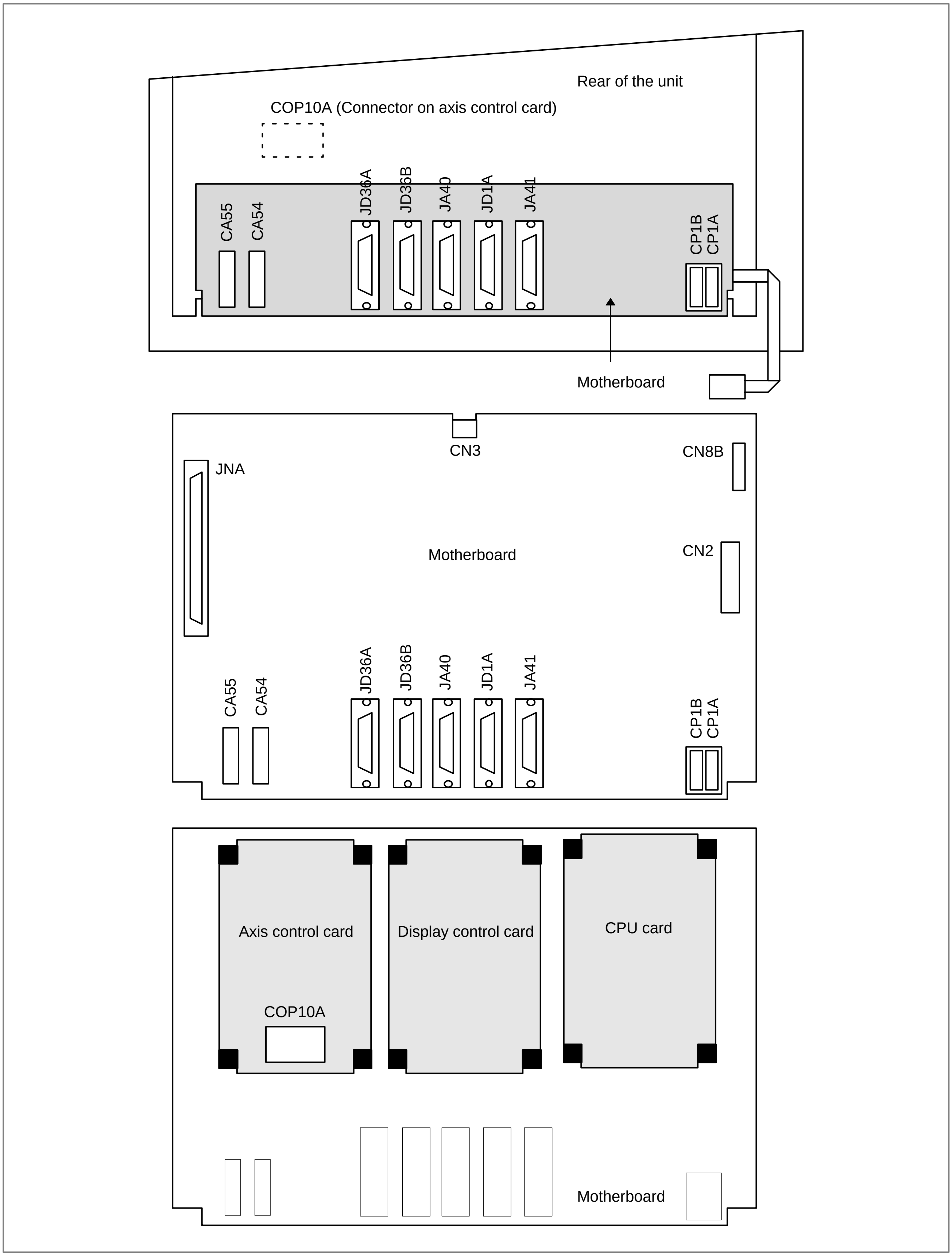
Fig. 2 EXTERNAL DIMENSIONS OF CNC CONTROL UNIT WITH 9.5″/10.4″ LCD

At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

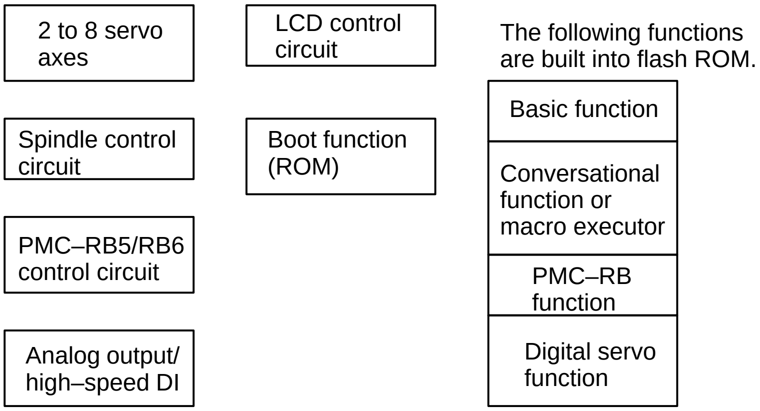


E PRINT BOARD

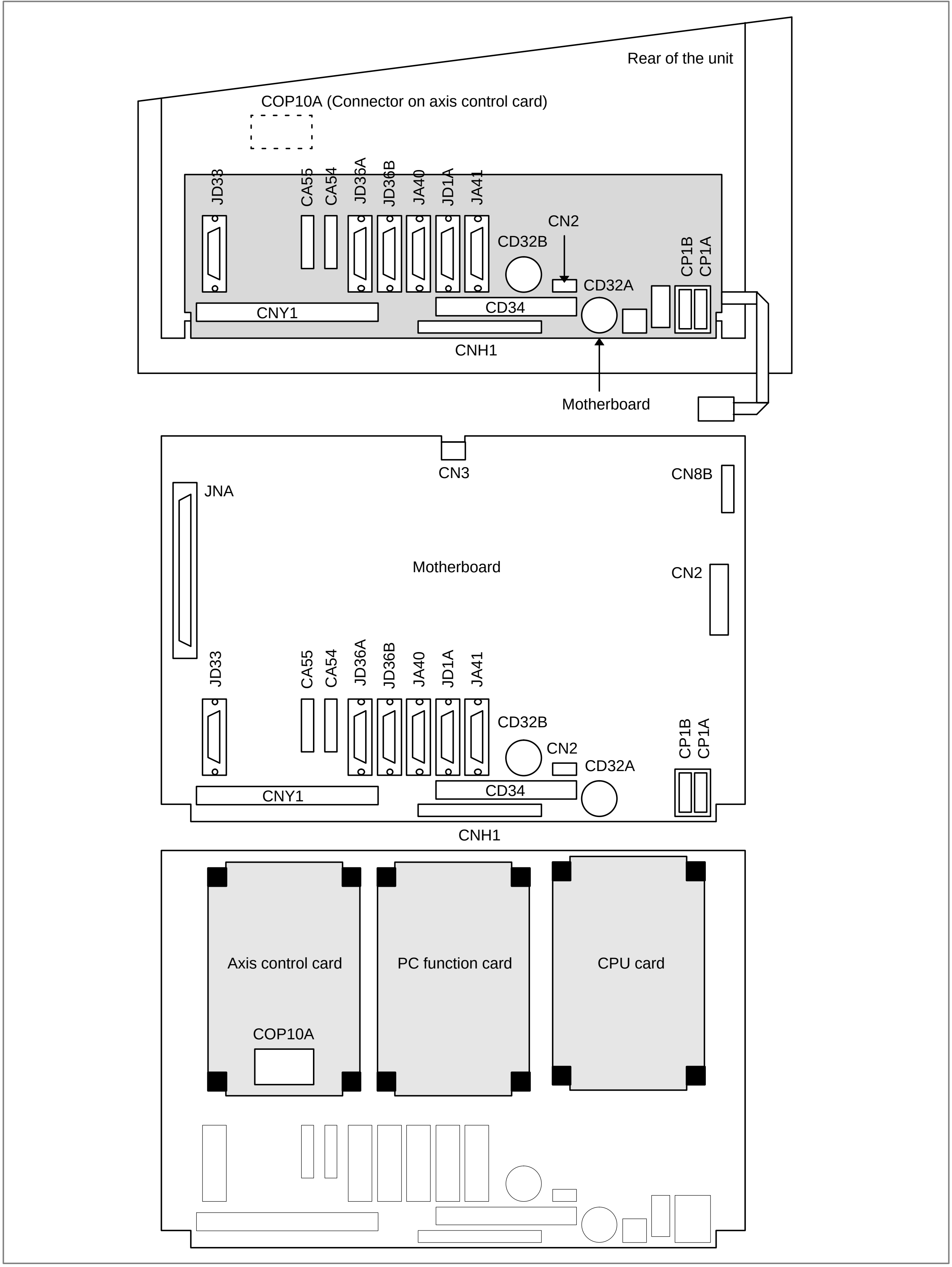
Motherboard (without PC functions)



Connector name	Function
COP10A	Servo amplifier (FSSB)
CA55	MDI
CA54	Servo check
JD36A	RS–232C serial port
JD36B	RS–232C serial port
JA40	Analog output/high–speed DI
JD1A	Serial I/O Link
JA41	Serial spindle/position coder
CP1B	DC24V–OUT
CP1A	DC24V–IN
JNA	F–BUS interface
CN8B	Video signal interface
CN2	PCMCIA interface
CN3	Inverter PCB interface



Motherboard (with PC functions)



Connector name	Function
JD33	RS–232C on PC side
COP10A	Servo amplifier (FSSB)
CA55	MDI
CA54	Servo check
JD36A	RS–232C serial port
JD36B	RS–232C serial port
JA40	Analog output/high–speed DI
JD1A	Serial I/O Link
JA41	Serial spindle/position coder
CP1B	DC24V–OUT
CP1A	DC24V–IN
CNY1	PC expansion
CD34	FDD signal
CNH1	HDD
CN2	FDD power
CD32A	Keyboard
CD32B	Mouse
JNA	F–BUS interface
CN8B	Video signal interface
CN2	PCMCIA interface
CN3	Inverter PCB interface

2 to 8 servo axes

Spindle control circuit

PMC–RB5/RB6 control circuit

Analog output/ high–speed DI

PC function

Boot function (ROM)

The following functions are built into flash ROM.

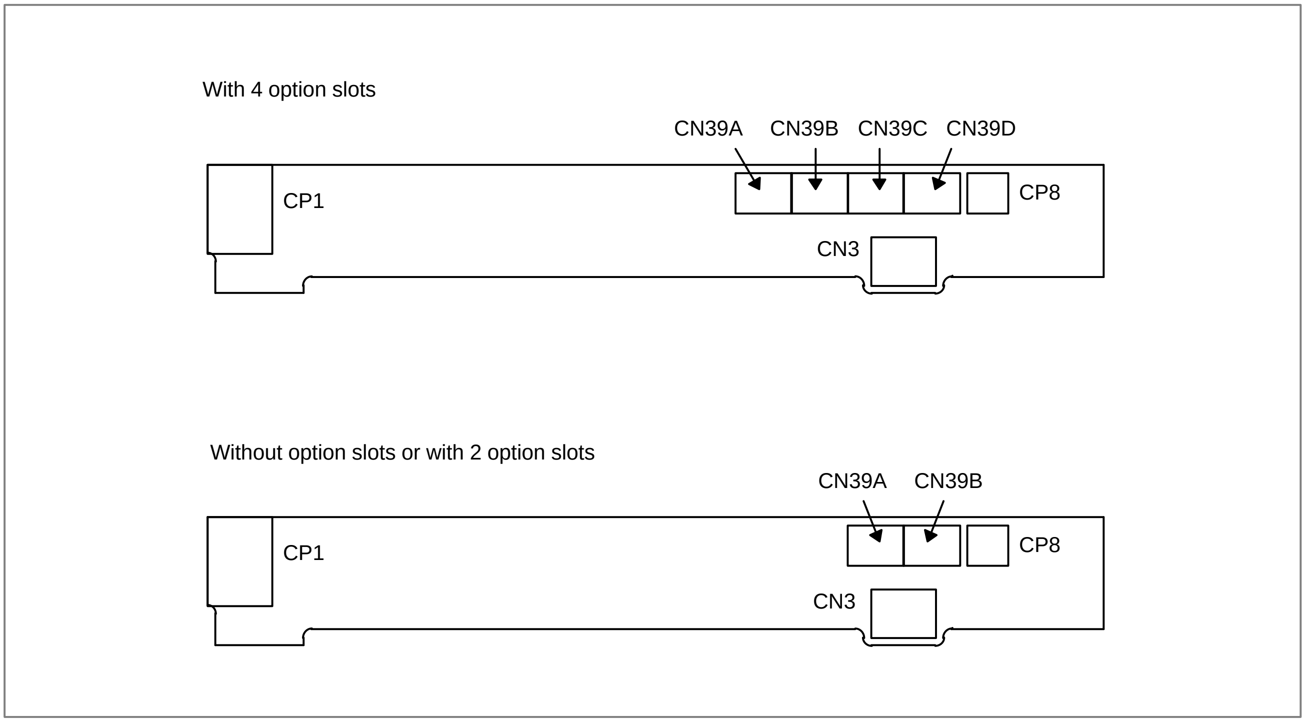
Basic function

Conversational function or macro executor

PMC–RB function

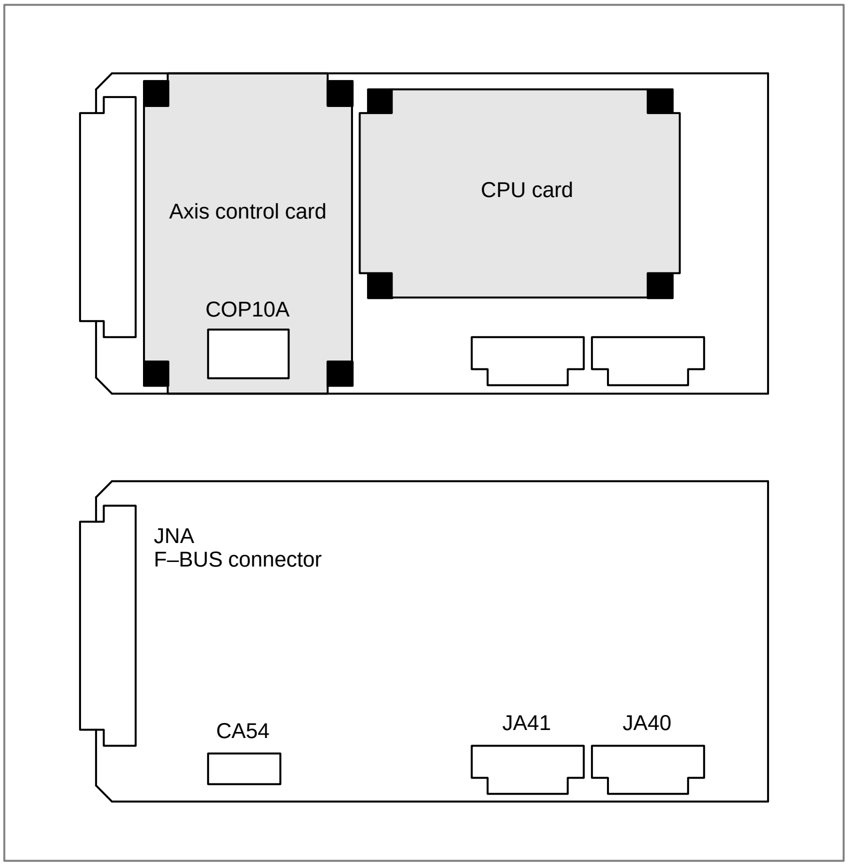
Digital servo function

Inverter PCB



Connector name	Function
CN39A	Fan power
CN39B	
CN39C	
CN39D	
CP8	Battery
CP1	LCD backlight power
CN3	Inverter PCB power

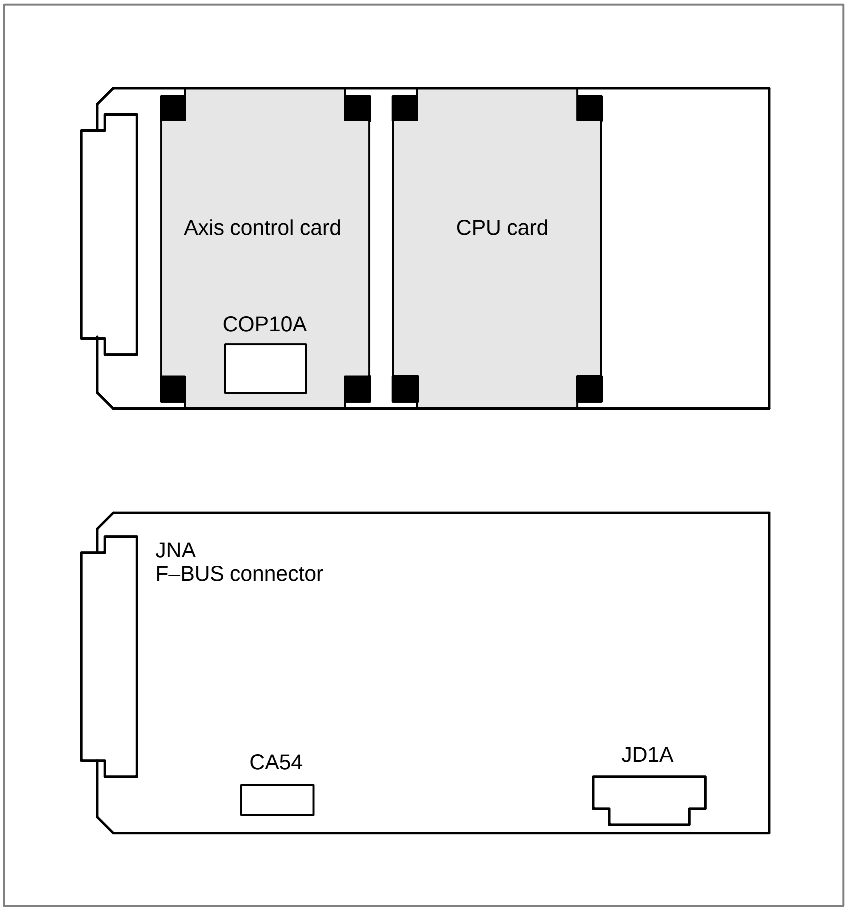
Sub–CPU board



Connector name	Function
COP10A	Servo amplifier (FSSB)
CA54	Servo check
JA41	Serial spindle/position coder
JA40	Analog output

Sub–CPU for 2–path control	2 to 8 servo axes	Spindle control circuit	Analog output

Loader control board



Connector name	Function
COP10A	Servo amplifier (FSSB)
CA54	Servo check
JD1A	Serial I/O Link

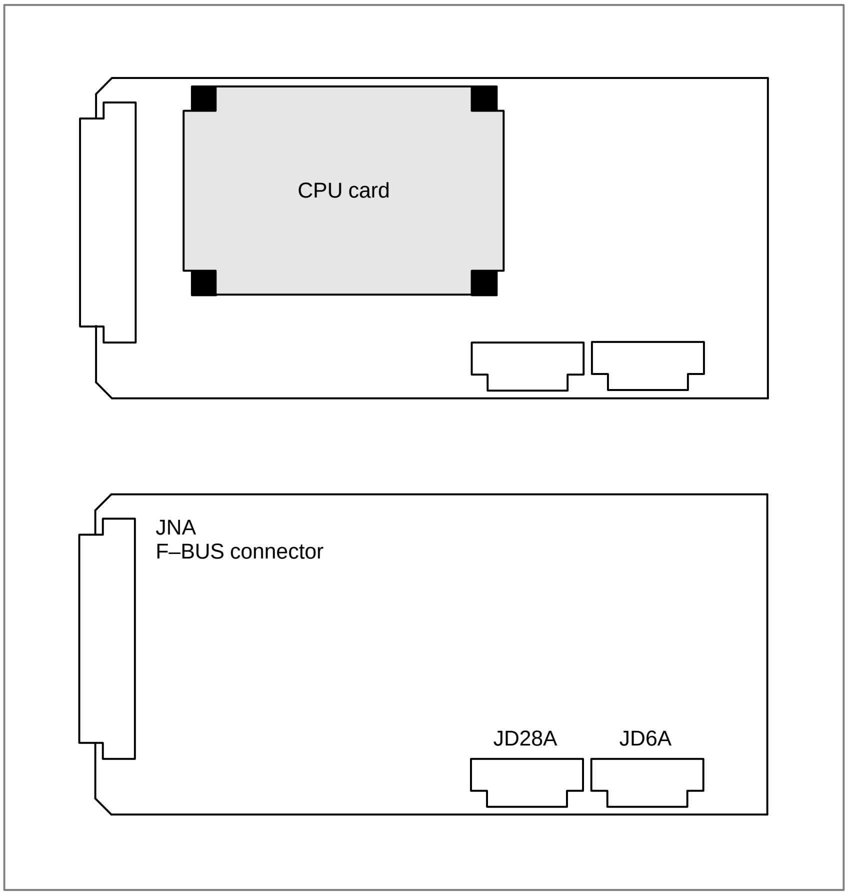
2 or 4 servo axes

Main memory for loader control

PMC control circuit

Loader control function

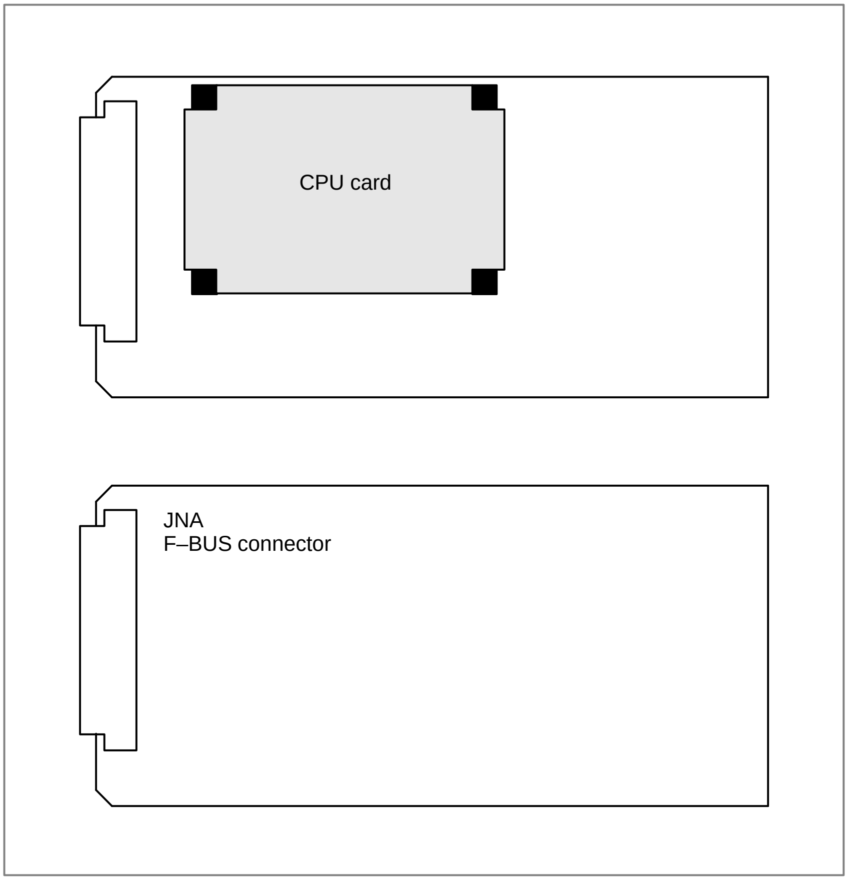
Serial communication board (remote buffer/ DNC1/DNC2/HDLC)



Connector name	Function
JD28A	RS–232C serial port
JD6A	RS–422 serial port

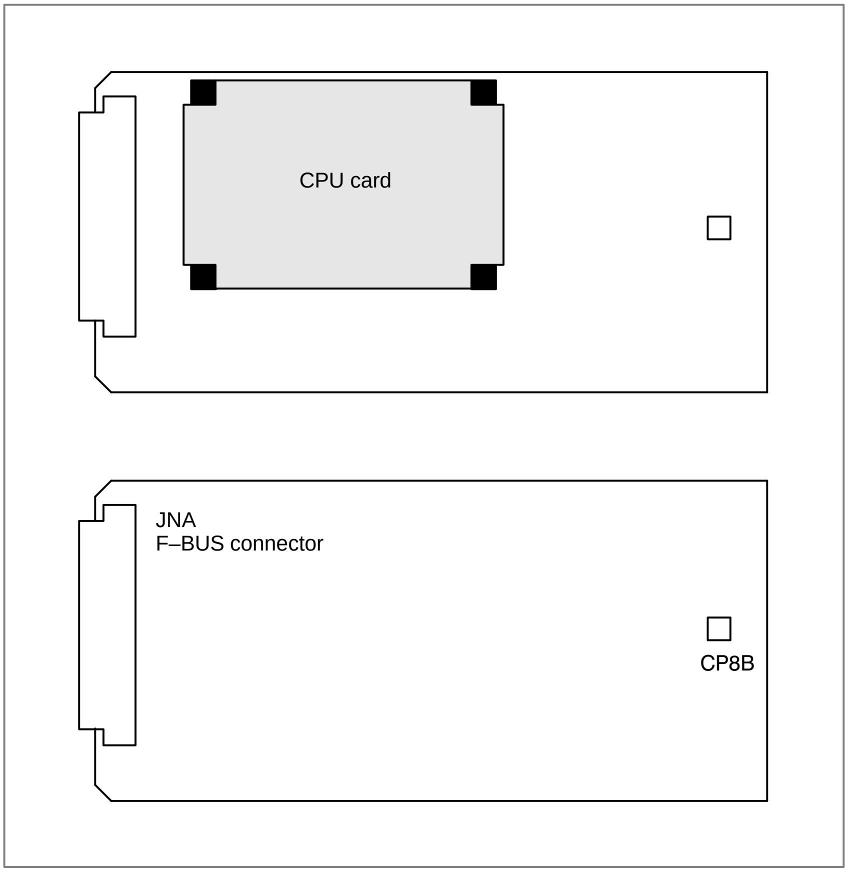
Communication function

C board



C function for PMC

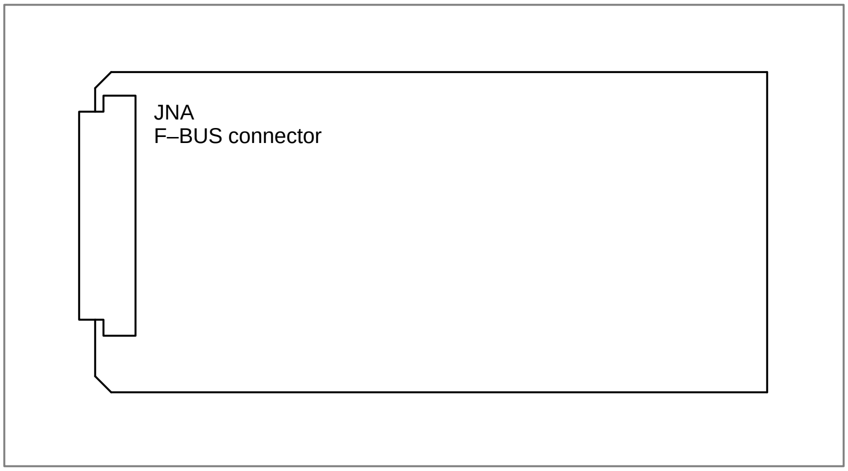
CAP–II board



Connector name	Function
Connector name	Function
CP8B	For SRAM backup battery

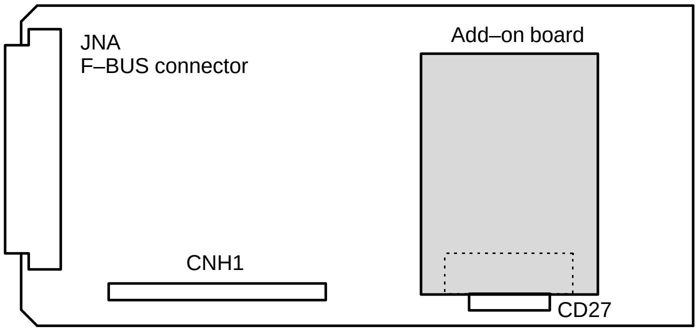
CAP II function

RISC board



High–precision contour control function

Data server board

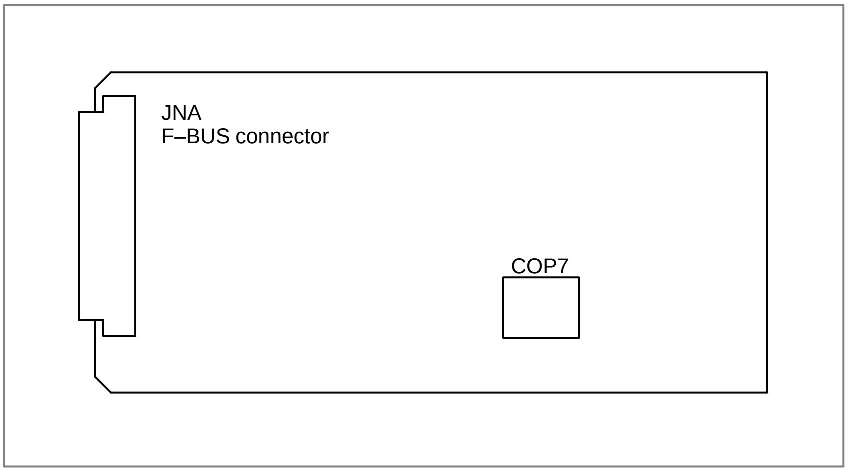


CD27 is provided on the add–on board.

Connector name	Function
CNH1	IDE hard disk interface
CD27	Ethernet interface

Data server function

HSSB interface board

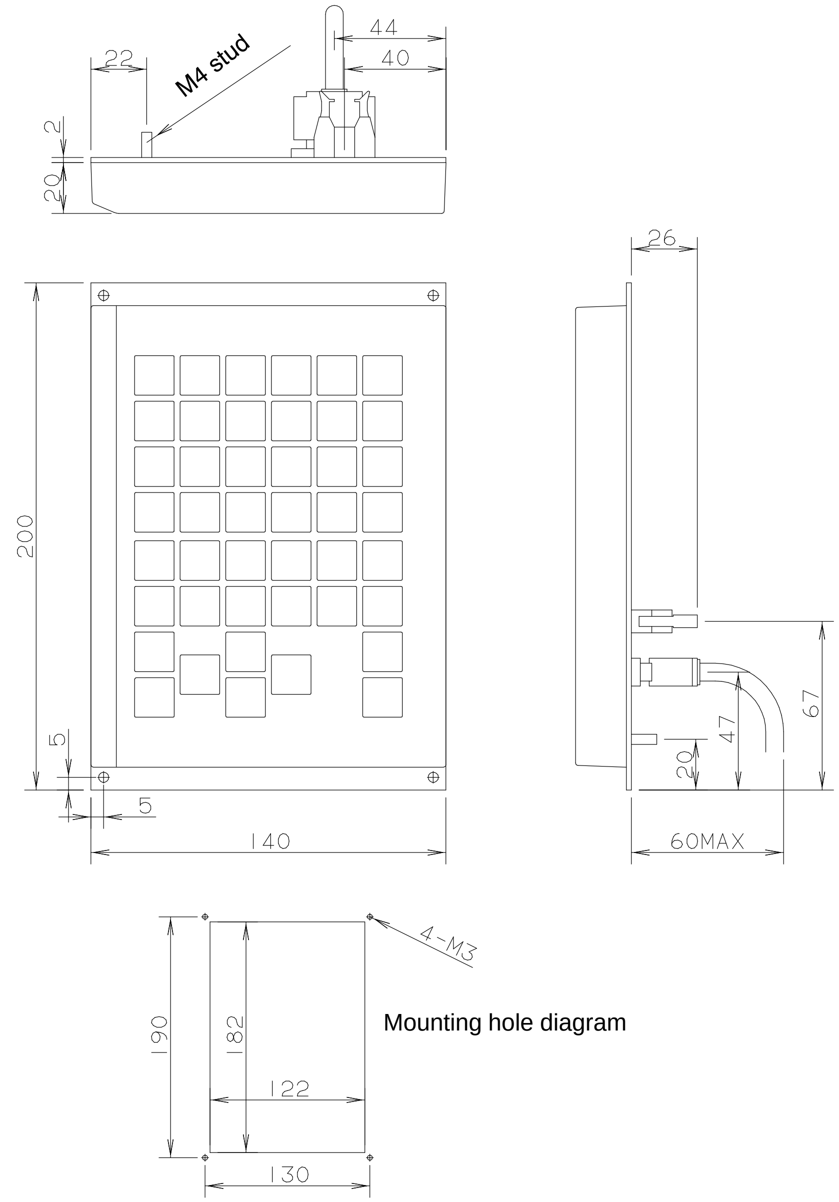


Connector name	Function
COP7	High–speed serial bus interface

High–speed serial bus interface

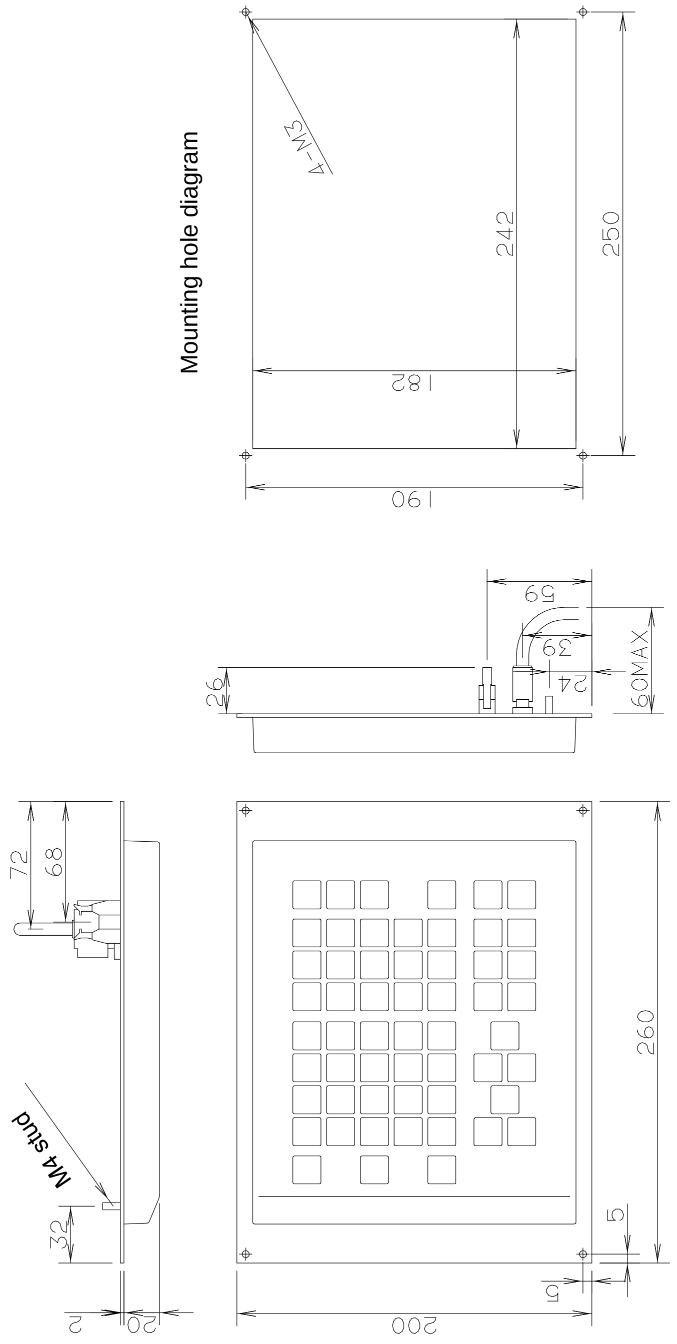
F EXTERNAL DIMENSIONS MDI UNIT

Fig. 1 EXTERNAL DIMENSIONS OF SEPARATE–TYPE SMALL MDI UNIT



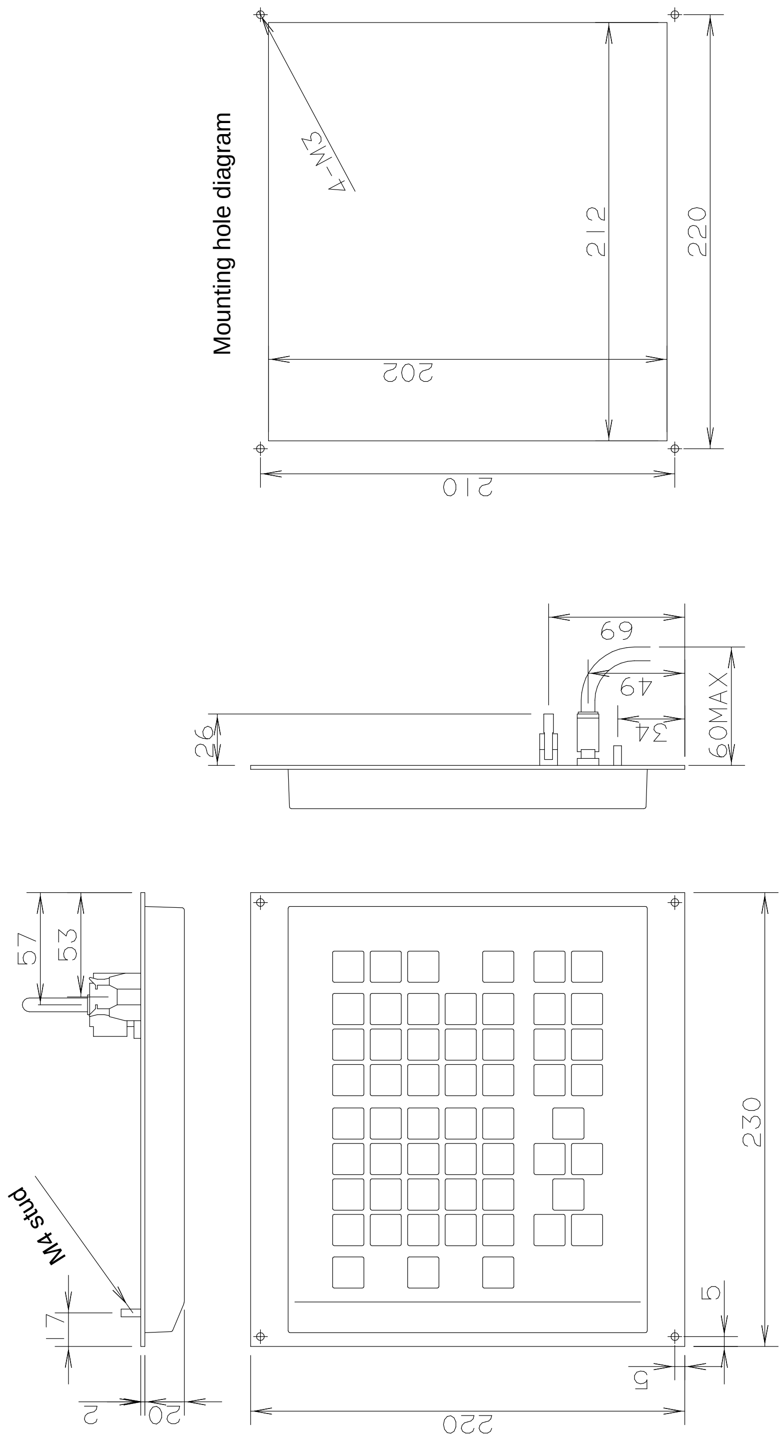
At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 2 EXTERNAL DIMENSIONS OF SEPARATE–TYPE STANDARD MDI UNIT



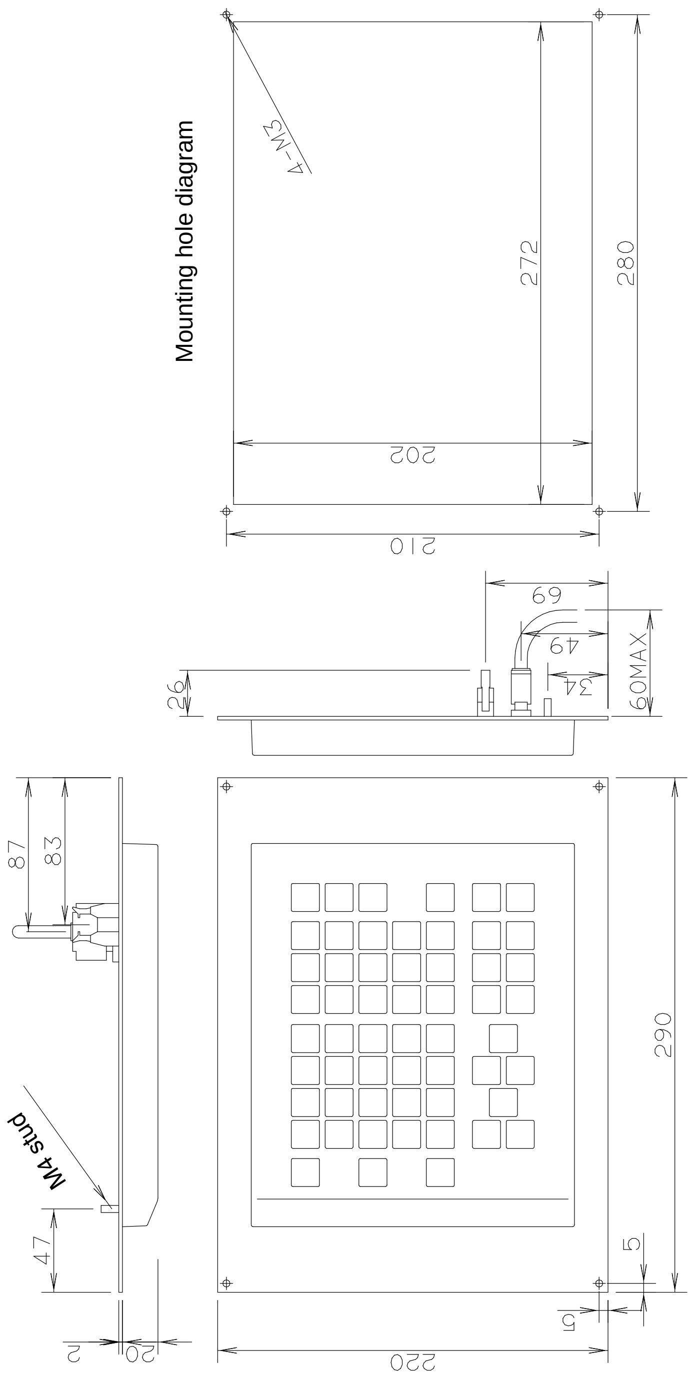
At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 3 EXTERNAL DIMENSIONS OF SEPARATE–TYPE STANDARD MDI UNIT (HORIZONTAL TYPE)



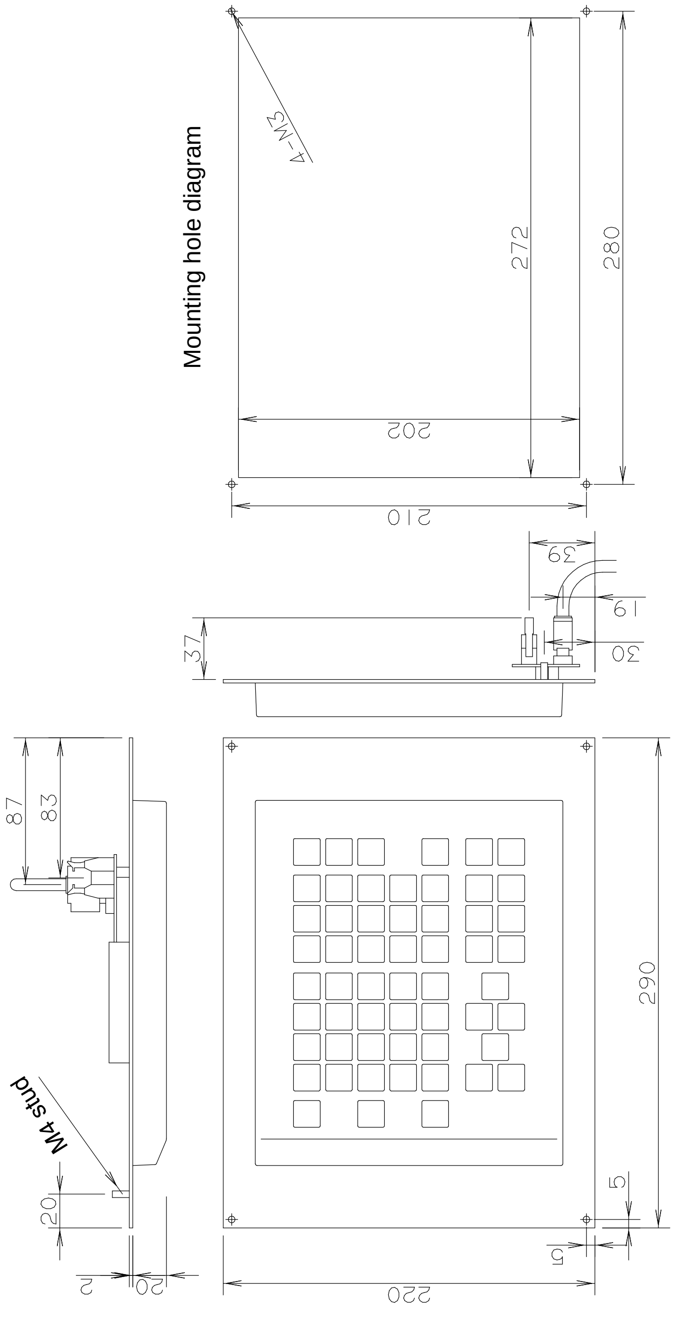
At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 4 EXTERNAL DIMENSIONS OF SEPARATE–TYPE STANDARD MDI UNIT (VERTICAL TYPE)



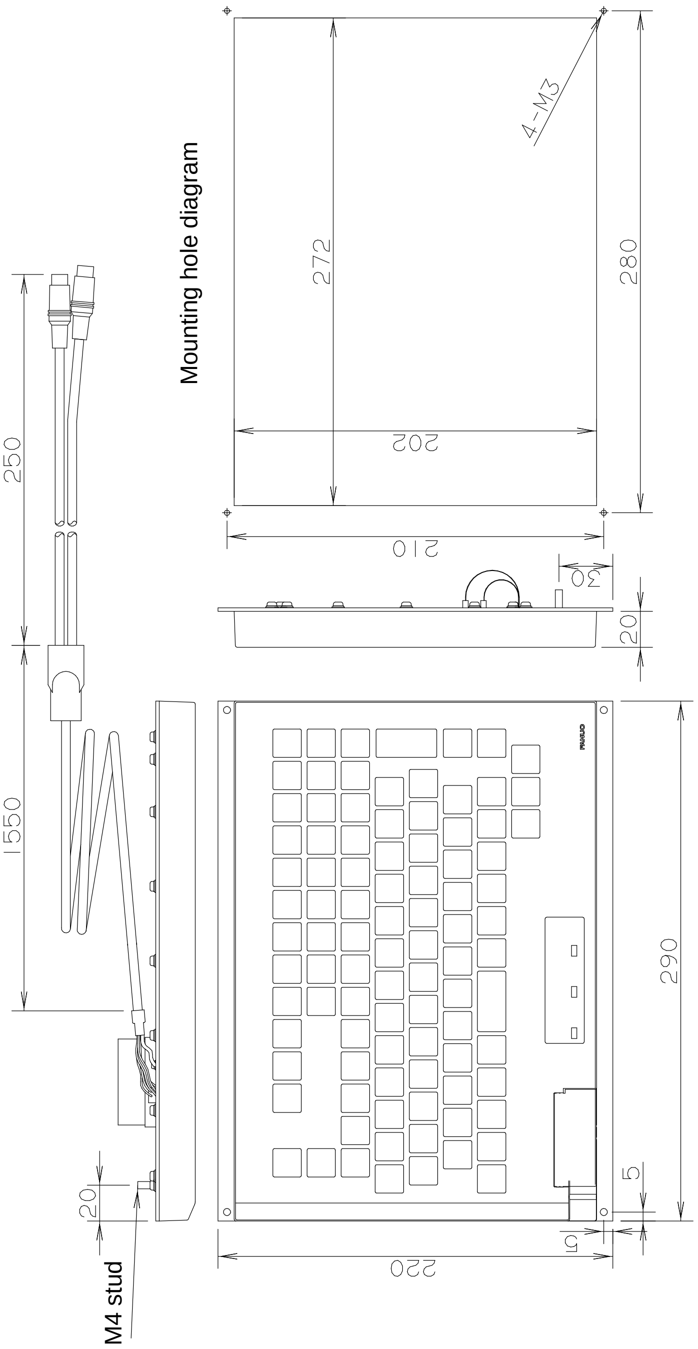
At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 5 EXTERNAL DIMENSIONS OF SEPARATE–TYPE STANDARD MDI UNIT (VERTICAL TYPE) FOR 160*i*/180*i*



At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

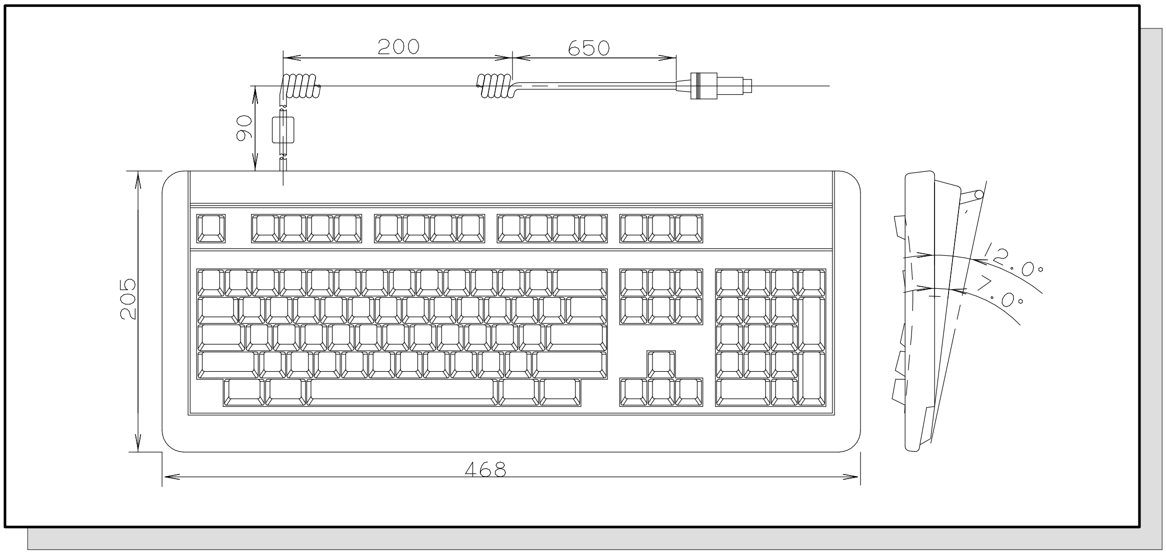
Fig. 6 EXTERNAL DIMENSIONS OF FA FULL KEYBOARD



At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 7(a) EXTERNAL DIMENSIONS OF 101–TYPE FULL KEYBOARD (ENGLISH)

Specification No.: A86L–0001–0210

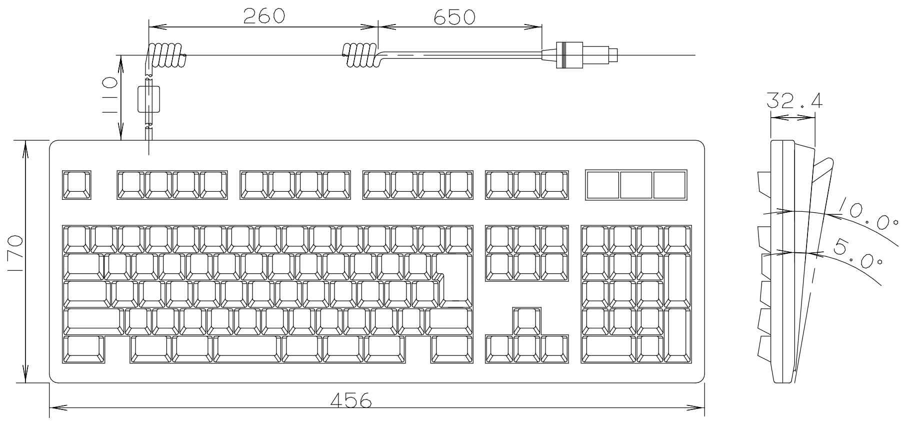


NOTE

This keyboard is not dust–proof. It should be used for program development only. It can be used at temperatures of between 0 and 40°C.

Fig. 7(b) EXTERNAL DIMENSIONS OF 106–TYPE FULL KEYBOARD (JAPANESE)

Specification No.: A86L–0001–0211

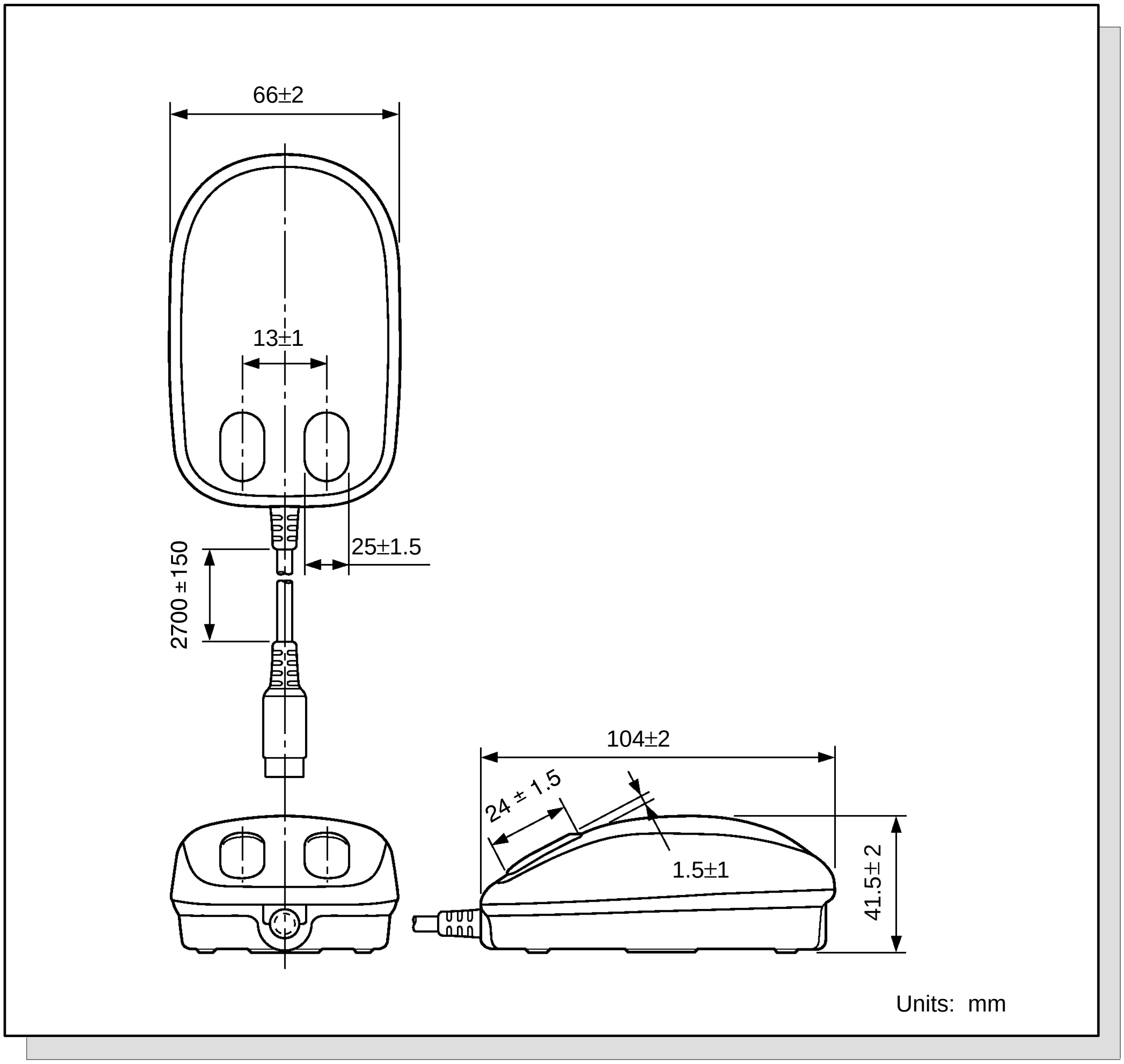


NOTE

This keyboard is not dust–proof. It should be used for program development only. It can be used at temperatures of between 0 and 40°C.

Fig. 8 EXTERNAL DIMENSIONS OF MOUSE

Specification No.: A86L–0001–0212



NOTE

This mouse is not dust–proof. It should be used for program development only. It can be used at temperatures of between 0 and 40°C. The mouse is fitted with a 2.7–m cable.

G EXTERNAL DIMENSIONS OF EACH UNIT

Fig. 1 EXTERNAL DIMENSIONS OF HIGH–SPEED SERIAL BUS INTERFACE BOARD TYPE 2 (FOR PC)

Specification No.: A20B–8100–0100

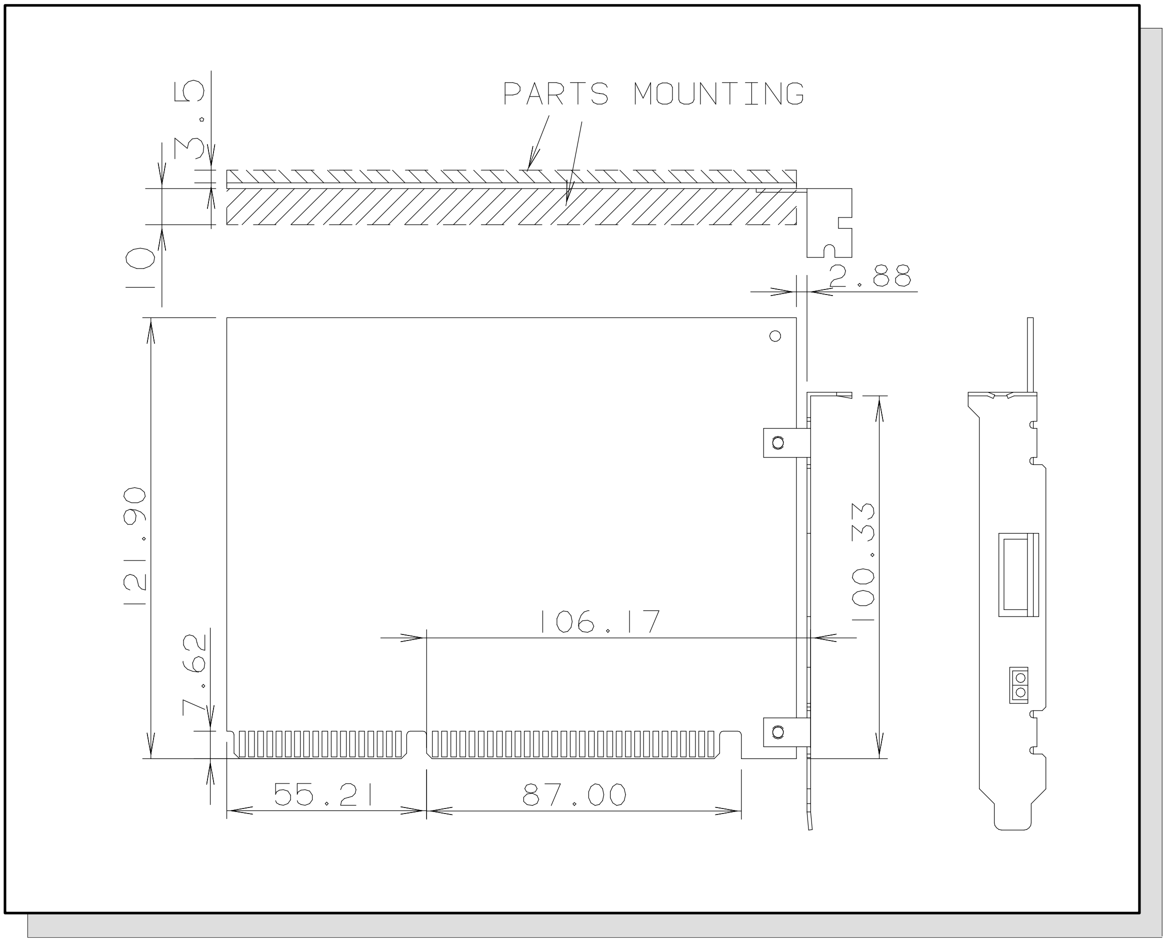
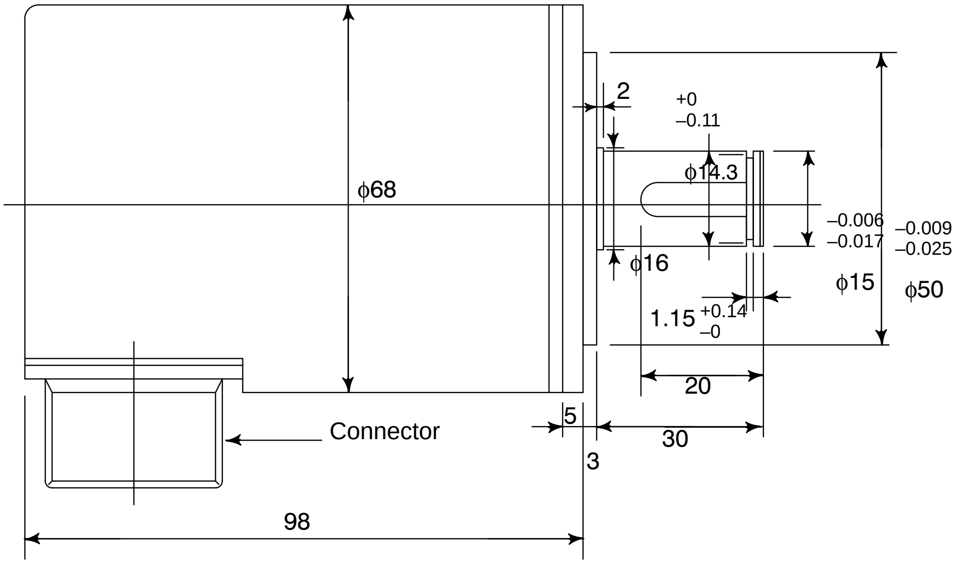
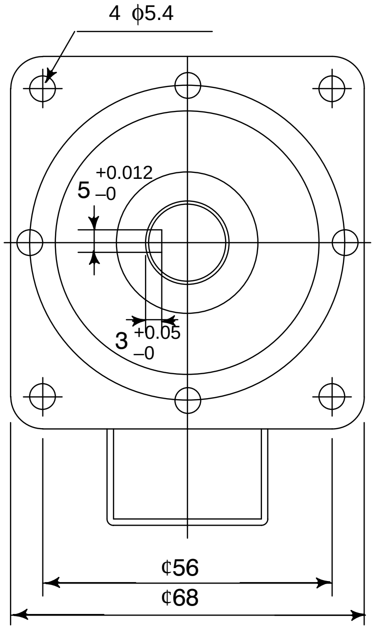


Fig. 2 POSITION CODER

Specification : A86L–0026–0001#102 (Max. 4000rpm)
A86L–0026–0001#002 (Max. 6000rpm)





Note : Mechanical specifications of the position coder are as follows :

(1) Input axis inertia
1.0_10^{-3}kg · cm · sec^{2} or less
(2) Input axis starting torque
100g · cm or less
(3) Input axis permissible loads
(4) Weight : 1kg or less

	Radial	Thrust
Operation	1kg or less	1kg or less
Idle	20kg or less	10kg or less

Attach a pulley directly to the position coder shaft and drive the timing belt. Confirm that the loads conform with the above allowable value.

Fig. 3 MANUAL PULSE GENERATOR

Specification : A860–0202–T001

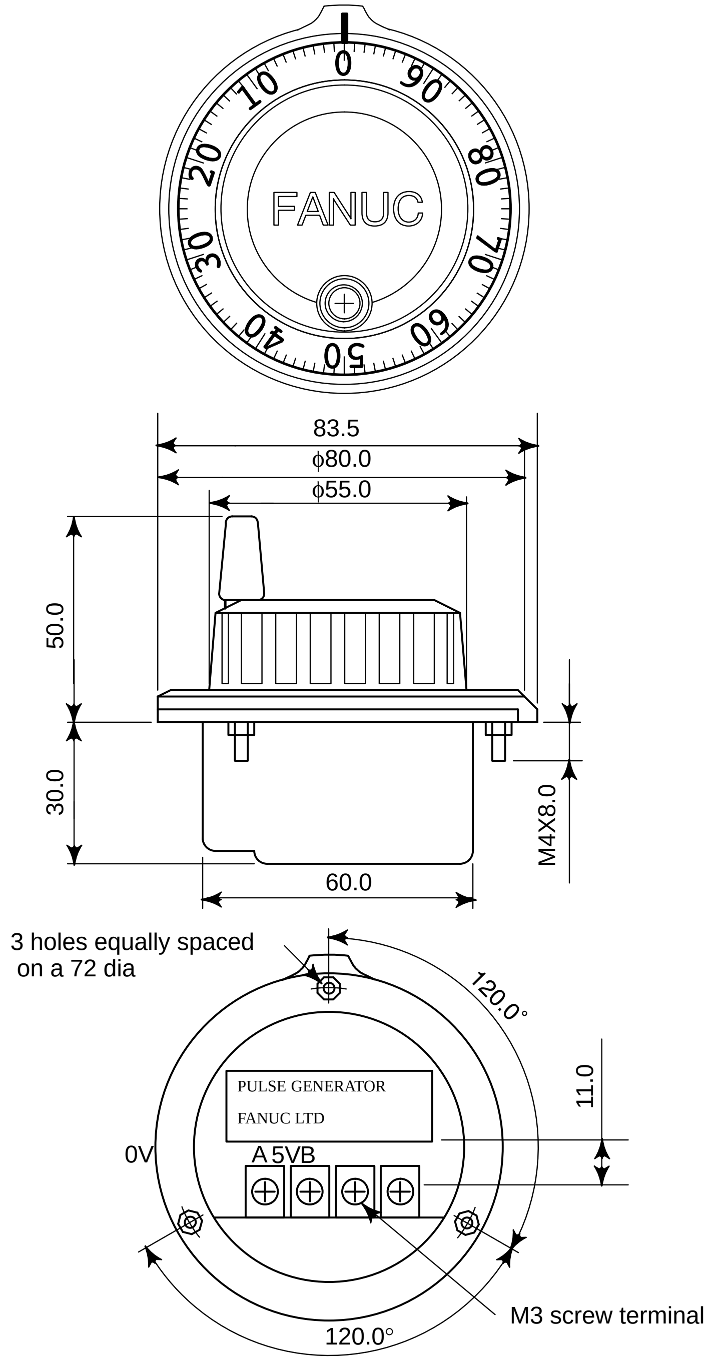
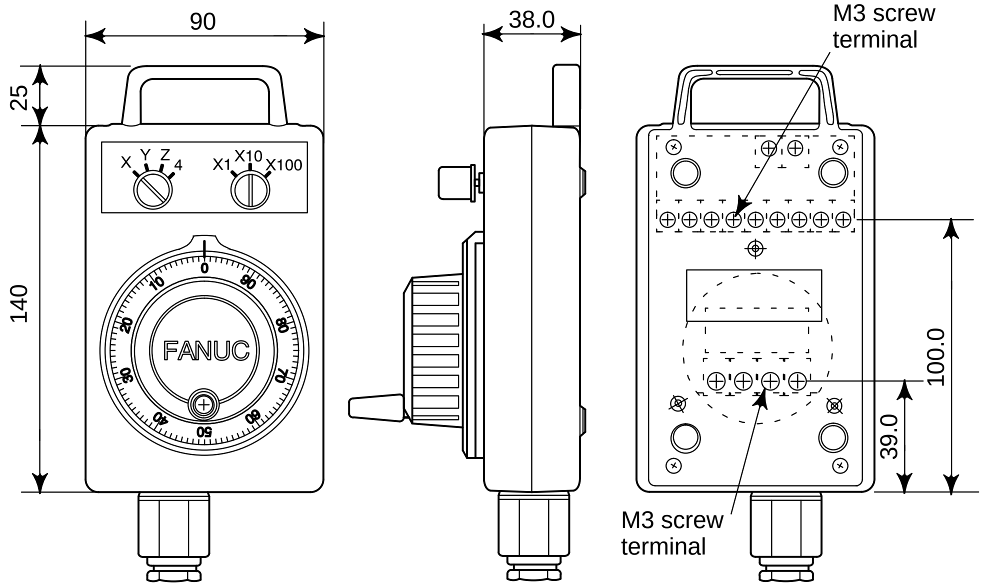


Fig. 4 PENDANT TYPE MANUAL PULSE GENERATOR

Specification : A860–0202–T004 to T015

A860–0202–T004 to T009



A860–0202–T010 to T015

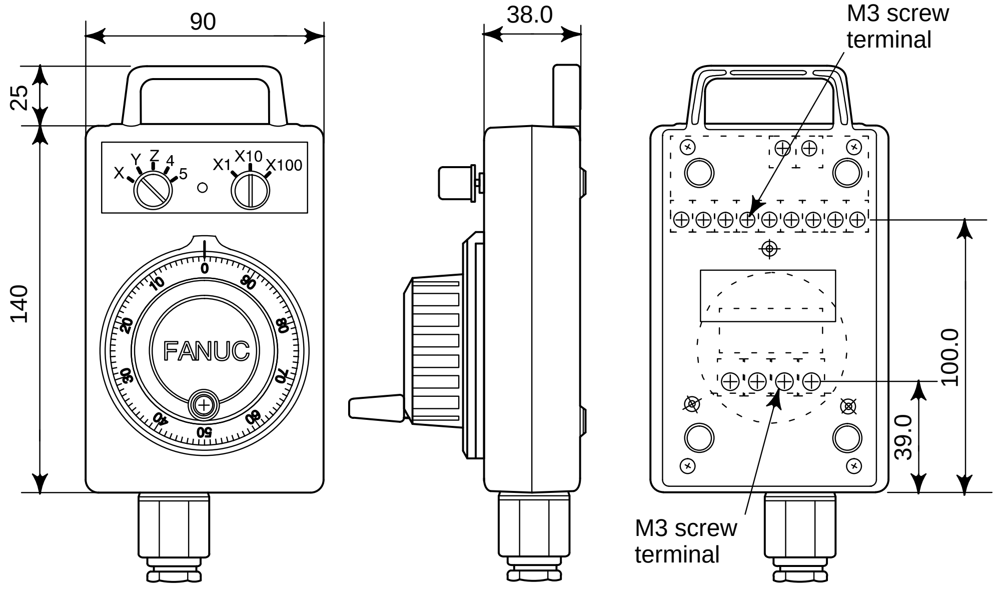


Fig. 5 EXTERNAL DIMENSIONS OF SEPARATE DETECTOR INTERFACE UNIT

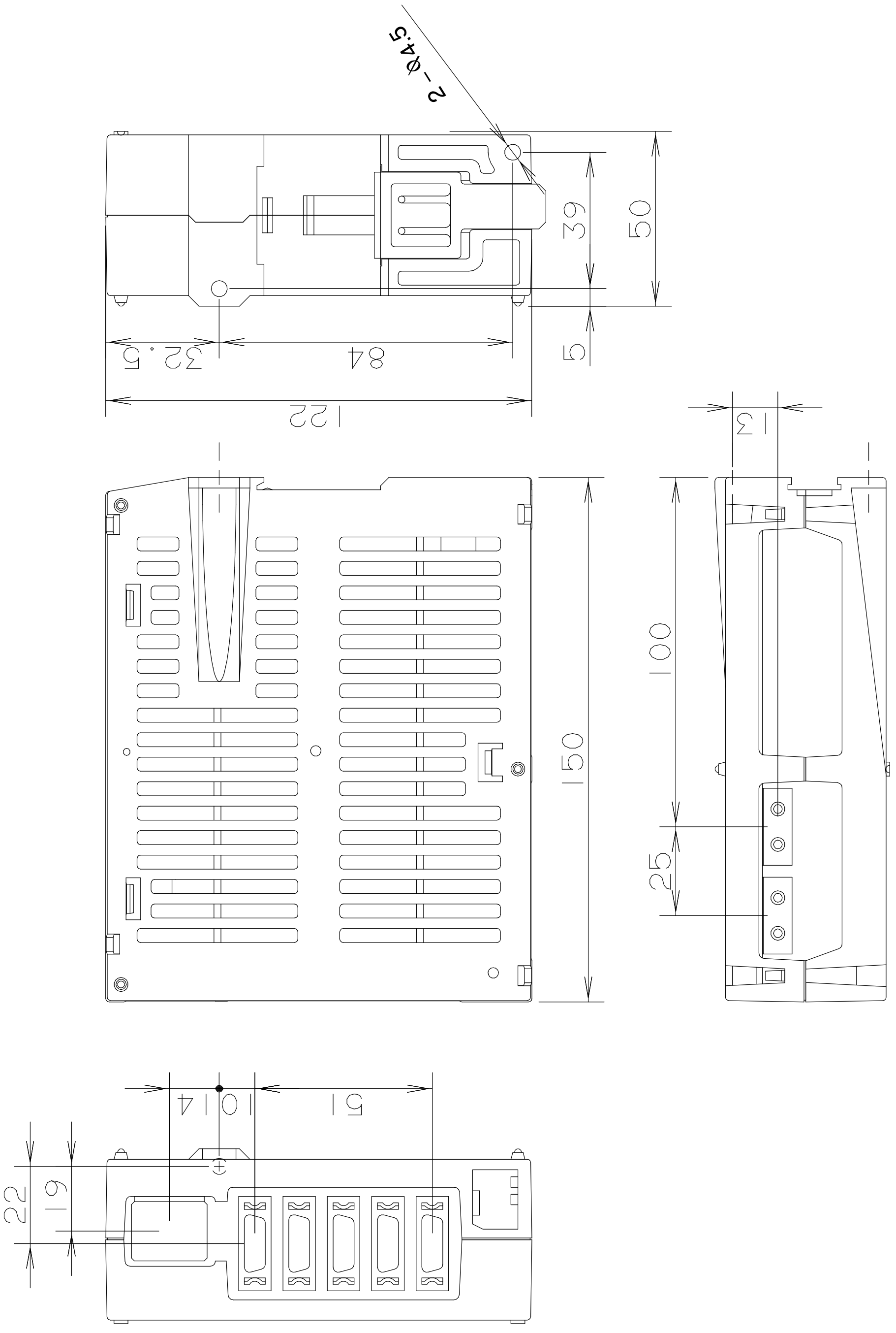


Fig. 6 BATTERY CASE FOR SEPARATE TYPE ABSOLUTE PULSE CODER

Specification : A06B–6050–K060

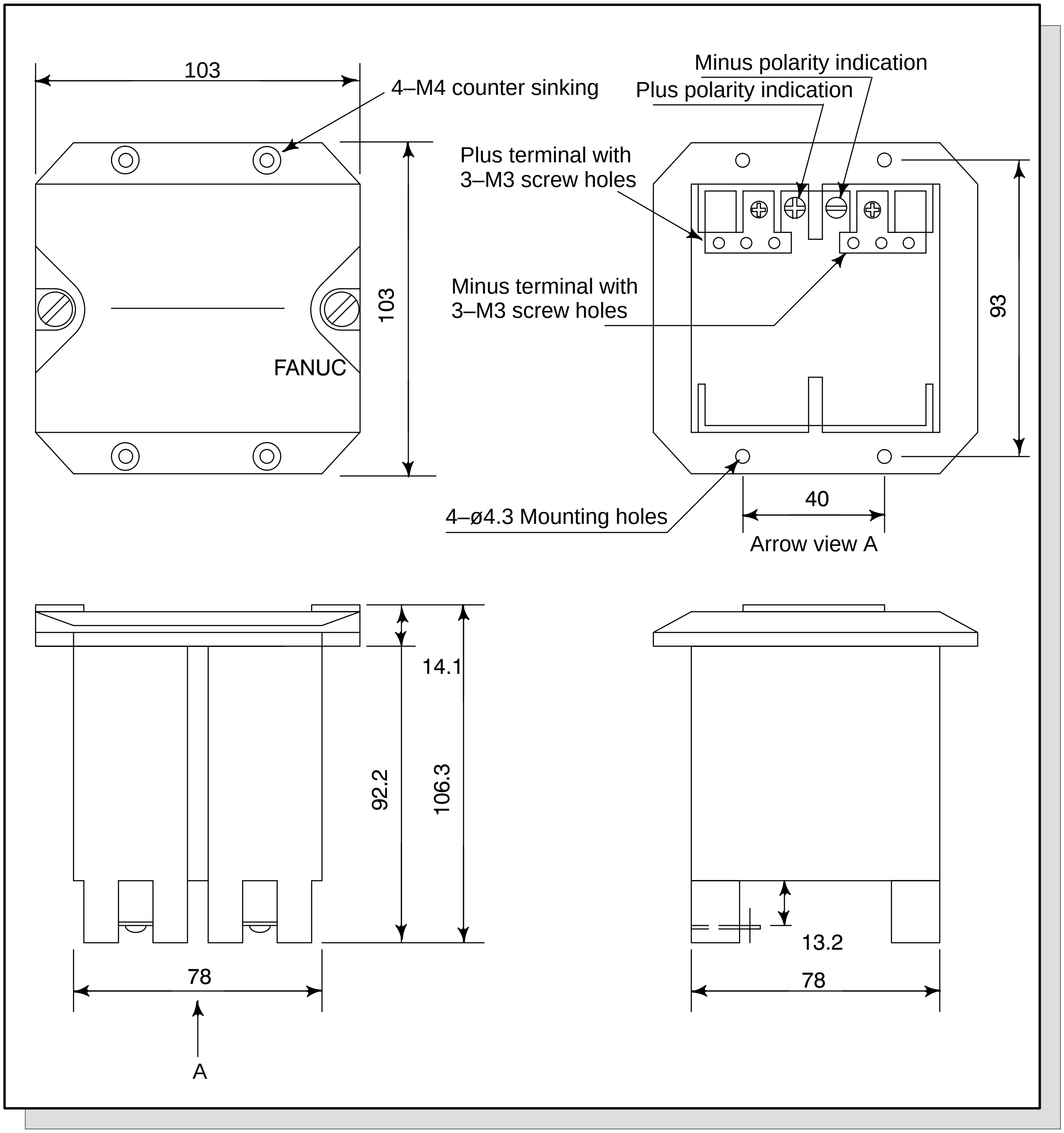


Fig. 7 EXTERNAL DIMENSIONS OF TAP

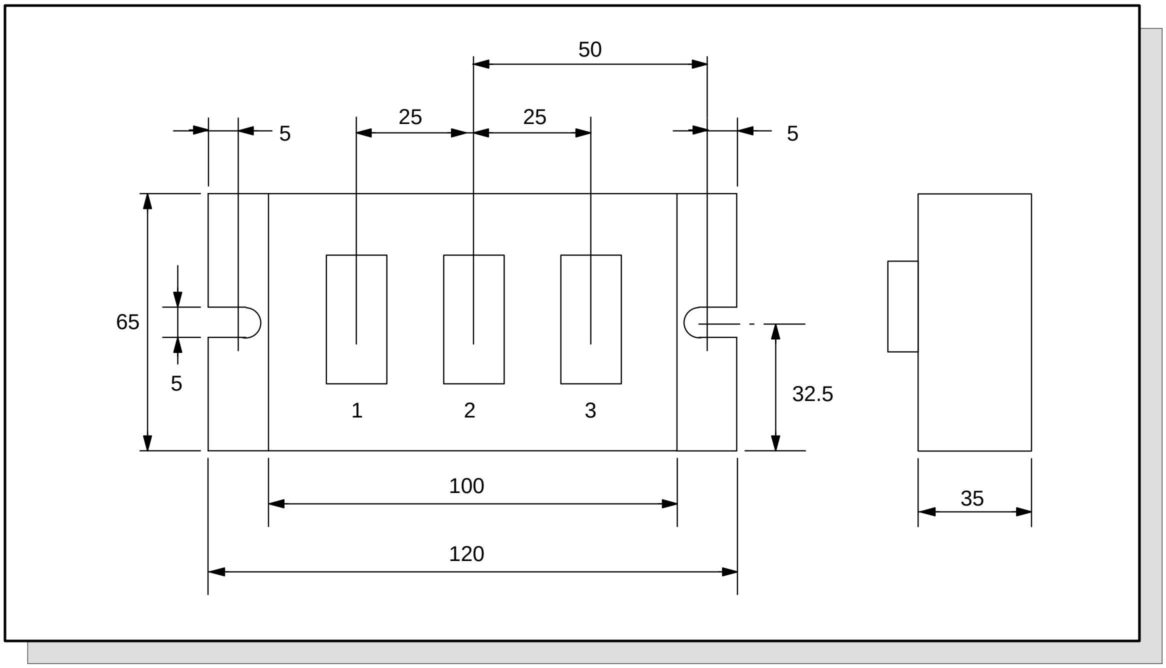


Fig. 8 EXTERNAL DIMENSIONS OF TERMINAL RESISTANCE UNIT

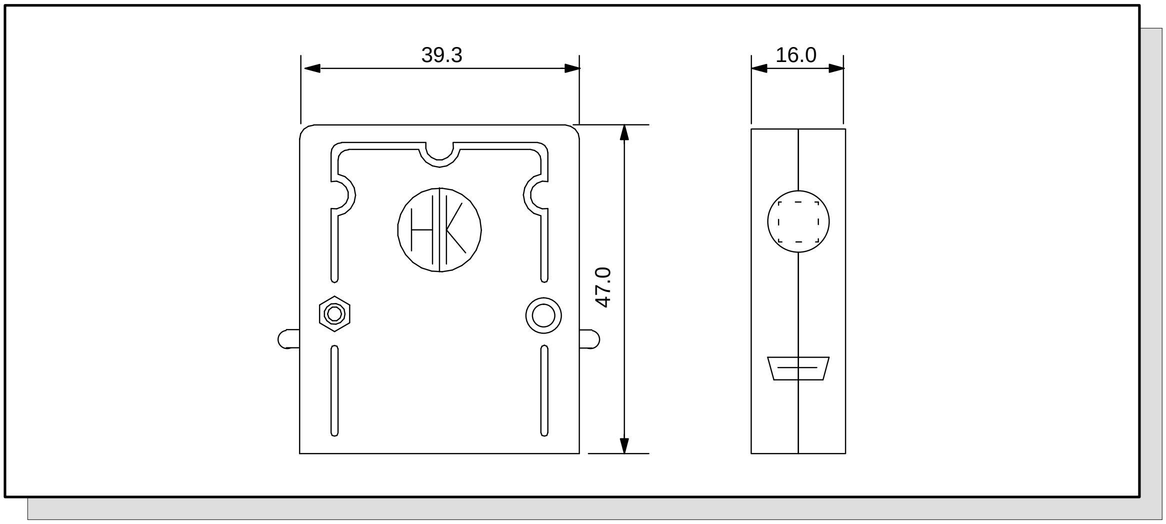
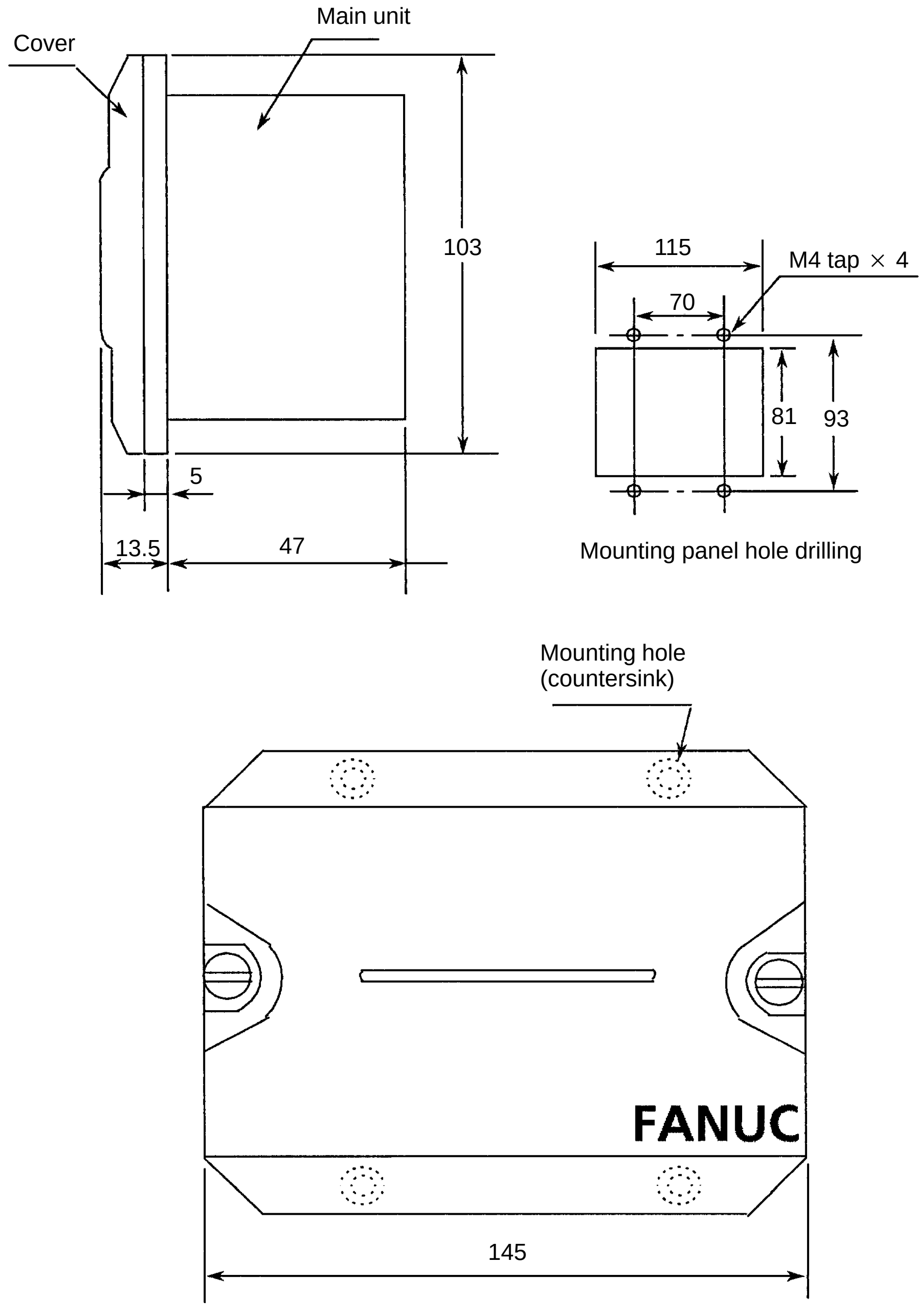
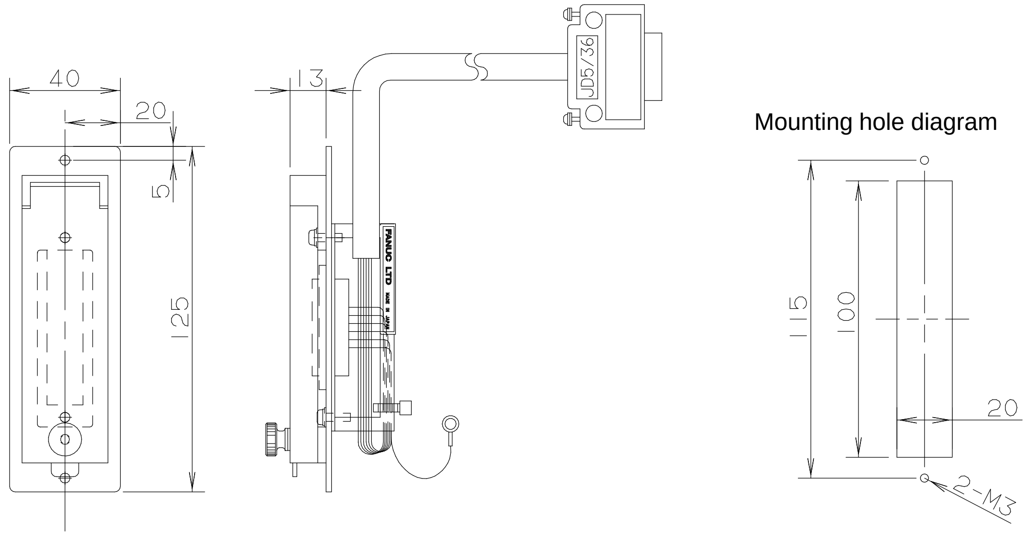


Fig. 9 EXTERNAL DIMENSIONS OF EXTERNAL CNC BATTERY UNIT



The battery unit is fitted with a 14–m battery cable.

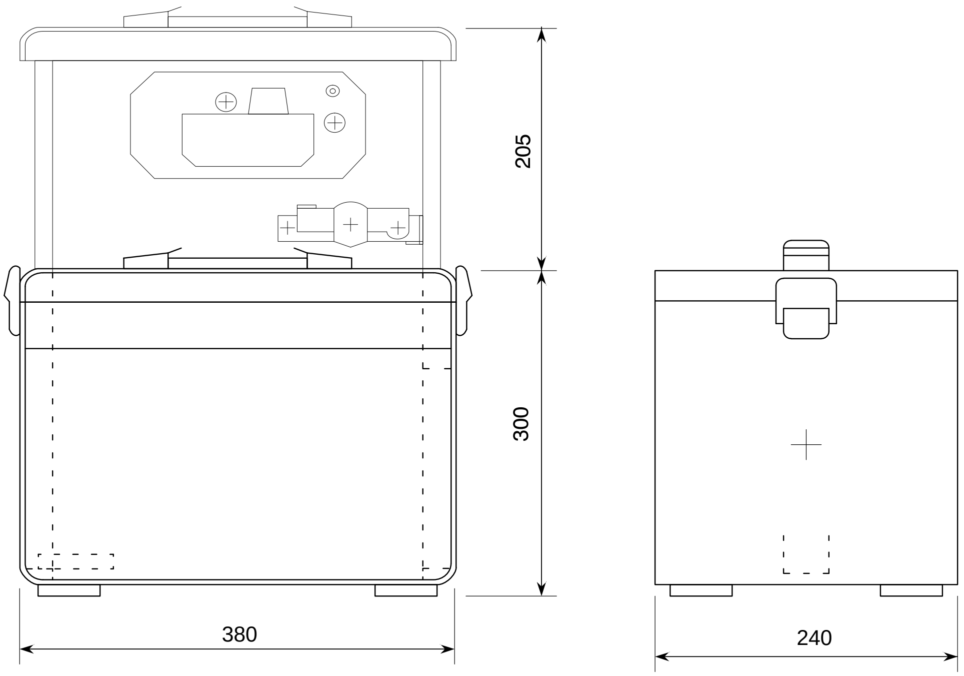
Fig. 10 EXTERNAL DIMENSIONS OF PUNCH PANEL (NARROW TYPE)



At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

Fig. 11 PORTABLE TAPE READER WITHOUT REELS

Specification : A13B–0074–B001



Paint : Munsell No. 5GY3.5/0.5 leather tone finish
Weight : Applox. 15kg

Fig. 12 PORTABLE TAPE READER WITH REELS

Specification : A13B–0087–B001

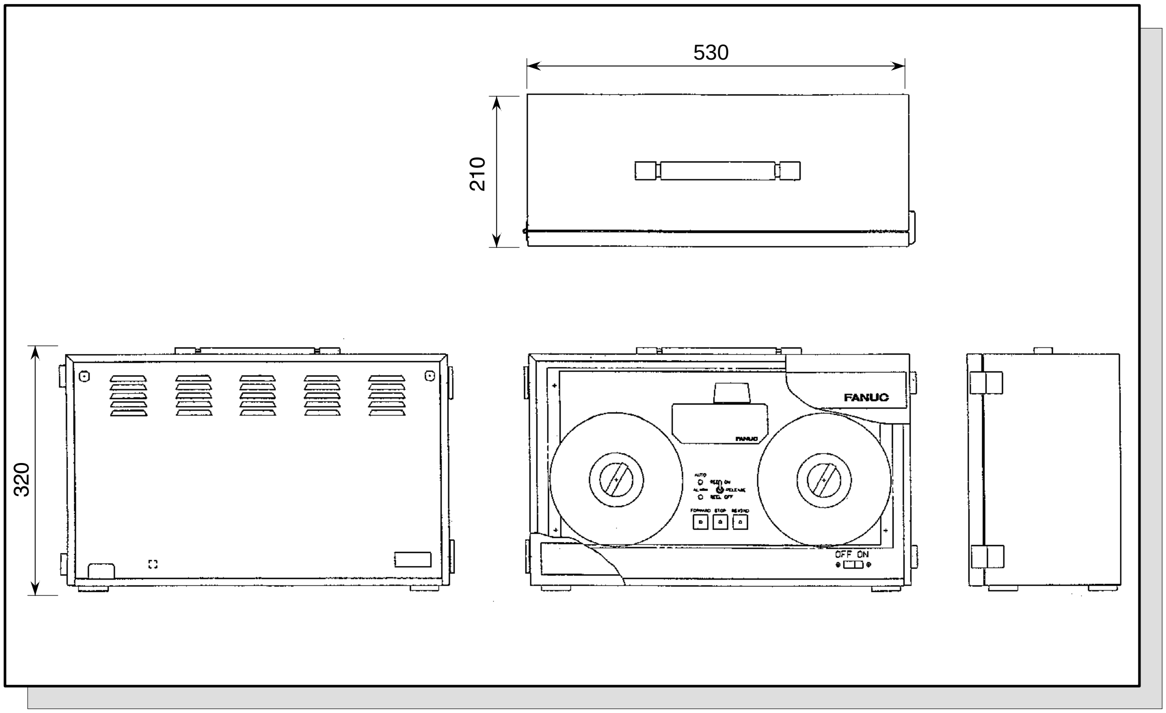
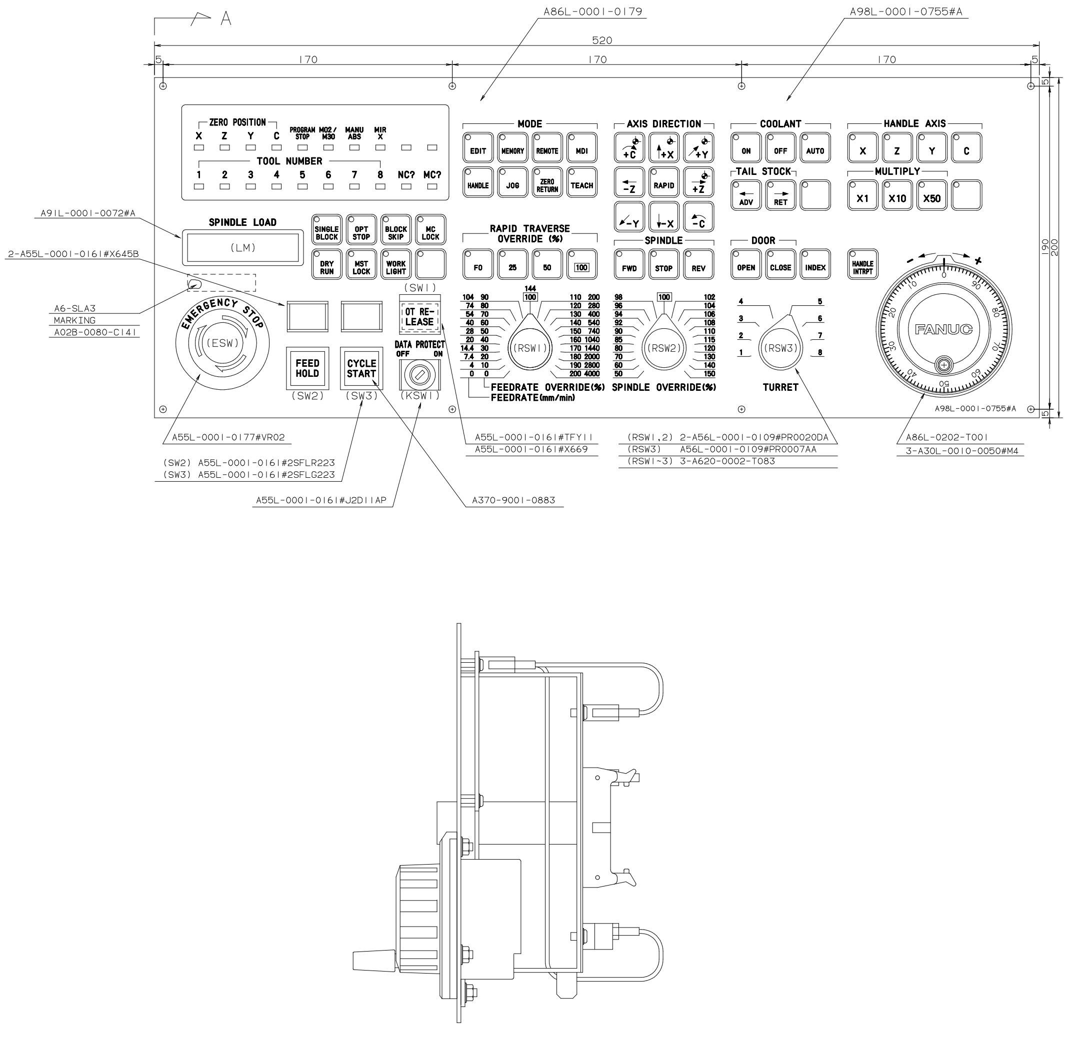


Fig. 13 STANDARD MACHINE OPERATOR’S PANEL

Specification : A02B–0080–C141 (T series)
A02B–0080–C142 (M series)



Revision Record

FANUC Series 16*i*/18*i*/21*i*/160*i*/180*i*/210*i*–MODEL A DESCRIPTIONS (B–63002EN)

Edition	Date	Contents	Edition	Date	Contents
01	Apr.,'97	—			