FANUC Series 0*i*-MODEL D
FANUC Series 0*i* Mate-MODEL D

DESCRIPTIONS

B-64302EN/01

The products in this manual are controlled based on Japan's "Foreign Exchange and Foreign Trade Law". The export from Japan may be subject to an export license by the government of Japan.
Further, re-export to another country may be subject to the license of the government of the country from where the product is re-exported. Furthermore, the product may also be controlled by re-export regulations of the United States government.
Should you wish to export or re-export these products, please contact FANUC for advice.

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

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

DEFINITION OF WARNING, CAUTION, AND NOTE

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

⚠ **WARNING**

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

⚠ **CAUTION**

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

NOTE

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

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

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.

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.

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 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 (T series) or normal-direction (perpendicular) control (M series), 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 **Interference check for each path (T series)**
An interference check for each path (T series) 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 (M series)**
Note that programmed operations vary considerably when a programmable mirror image (M series) 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.

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 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.

⚠ **WARNING**

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 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.

4 **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.

5 **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.

6 **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.

7 **Software operator's panel**
Using the software operator's panel, 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.

8 **RESET key**
Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI panel problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

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.

⚠ **WARNING**

10 **Feed hold, override, and single block**
The feed hold, feedrate override, and single block functions can be disabled using custom macro system variables #3003 and #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/tool nose radius compensation in MDI mode**
Pay careful attention to a tool path specified by a command in MDI mode, because cutter/ tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter compensation mode (M series) or tool nose radius compensation mode (T series), 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.

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 Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) 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 Section “Method of replacing battery” in the OPERATOR’S MANUAL (Common to Lathe System/Machining Center System) for details of the battery replacement procedure.

⚠ WARNING

3 **Fuse replacement**

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

APPENDIX

I. GENERAL

1 GENERAL

This manual describes the models indicated in the table below.
In the text, the abbreviations indicated below may be used.

Model name	Abbreviation		
FANUC Series 0*i* -TD	0*i* -TD	Series 0*i* -D	0*i* -D
FANUC Series 0*i* -MD	0*i* -MD		
FANUC Series 0*i* Mate -TD	0*i* Mate -TD	Series 0*i* Mate -D	0*i* Mate -D
FANUC Series 0*i* Mate -MD	0*i* Mate -MD		

NOTE

1. For explanation purposes, these models may be classified as shown below:
   - T series: 0*i* -TD / 0*i* Mate -TD
   - M series: 0*i* -MD / 0*i* Mate -MD
2. The 0*i* -D / 0*i* Mate -D requires setting of parameters to enable part of basic functions. For the parameters to be set, see Section 4.51, "PARAMETERS OF 0*i* -D / 0*i* Mate -D BASIC FUNCTIONS" in the PARAMETER MANUAL (B-64310EN).

Related manuals of Series 0*i* -D,Series 0*i* Mate -D

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

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-64302EN	*
CONNECTION MANUAL (HARDWARE)	B-64303EN	
CONNECTION MANUAL (FUNCTION)	B-64303EN-1	
OPERATOR'S MANUAL (Common to Lathe System/Machining Center System)	B-64304EN	
OPERATOR'S MANUAL (For Lathe System)	B-64304EN-1	
OPERATOR'S MANUAL (For Machining Center System)	B-64304EN-2	
MAINTENANCE MANUAL	B-64305EN	
PARAMETER MANUAL	B-64310EN	
START-UP MANUAL	B-64304EN-3	
Programming		
Macro Compiler / Macro Executor PROGRAMMING MANUAL	B-64303EN-2	
Macro Compiler OPERATOR'S MANUAL	B-64304EN-5	
C Language PROGRAMMING MANUAL	B-64303EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-64393EN	
Network		
PROFIBUS-DP Board OPERATOR'S MANUAL	B-64404EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64414EN	
Operation guidance function		
MANUAL GUIDE *i* (Common to Lathe System/Machining Center System) OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* (For Machining Center System) OPERATOR'S MANUAL	B-63874EN-2	
MANUAL GUIDE *i* (Set-up Guidance Functions) OPERATOR'S MANUAL	B-63874EN-1	
MANUAL GUIDE 0*i* OPERATOR'S MANUAL	B-64434EN	

Manual name	Specification number	
TURN MATE *i* OPERATOR'S MANUAL	B-64254EN	

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

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

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR β*i* series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR β*i* series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR α*i*s series FANUC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i* series FANUC LINEAR MOTOR L*i*S series FANUC SYNCHRONOUS BUILT-IN SERVO MOTOR D*i*S series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR α*i*/β*i* series, BUILT-IN SPINDLE MOTOR B*i* series PARAMETER MANUAL	B-65280EN

This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

Special symbols

This manual uses the following symbols:

- M

Indicates the description that are valid only for the M series.
The term "M series" used in the text means "machining center system type".
In a general description of the method of machining, an M series operation is identified by a phase such as "for milling machining".

- T

Indicates the description that are valid only for the T series.
The term "T series" in the text means "lathe system type".

In a general description of the method of machining, a T series operation is identified by a phrase such as "for lathe cutting".

-

Indicates the end of a description of a control type.
When a control type mark mentioned above is not followed by this mark, the description of the control type is assumed to continue until the next item or paragraph begins. In this case, the next item or paragraph provides a description common to the control types.

- IP

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (Used for descriptions of command formats)

- ;

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.
(Used for descriptions of command formats)

II. NC FUNCTION

1 CONTROLLED AXIS

Chapter 1, "CONTROLLED AXIS", consists of the following sections:

1.1 NUMBER OF MAXIMUM CONTROLLED AXES

The number of maximum controlled axes is the number of machine controlled axes.
The number of Cs and PMC axes is included in the number of machine controlled axes.
The number of maximum available controlled axes, which depends on the model, control type, or option configuration, is shown blow.

Item	**0*i* -MD**	**0*i* -TD**	
		1-path system	**2-path system**
Max. controlled axes	5	4	8 (Total of 2-path)

Item	**0*i* Mate -MD**	**0*i* Mate -TD**
Max. controlled axes	4	3

1.2 NUMBER OF MACHINE GROUPS

If multiple paths are used, several paths can be formed into a group. By doing so, the group can share data, and if an alarm is issued with a path, the other path(s) in the group can be stopped. A group of those paths is referred to as a machine group.
The maximum number of available machine groups for the 0*i*-D/0*i* Mate-D is 1.

Item	**0*i* -MD**	**0*i* -TD**	
		1-path system	**2-path system**
Machine groups	1	1	1

Item	**0*i* Mate -MD**	**0*i* Mate -TD**
Machine groups	1	1

1.3 NUMBER OF CONTROLLED PATHS

A path represents a group of axes that are controlled by the same NC program.
The maximum number of available controlled paths is shown below.

Item	**0*i* -MD**	**0*i* -TD**
Control paths	1	2

Item	**0*i* Mate -MD**	**0*i* Mate -TD**
Control paths	1	1

1.3.1 2-path Control (T Series)

The 2-path control function for using two paths performs two types of machining at the same time independently. This function is suited to a lathe or automatic lathe that uses two tool posts to perform cutting at the same time.
The inter-path waiting function, inter-path interference check function, balanced cutting, synchronous/composite/superimposed control, inter-path spindle control, and inter-path shared memory are provided as functions specific to 2-path control.

1.4 NUMBER OF CONTROLLED AXES / NUMBER OF CONTROLLED SPINDLE AXES

Structure of the number of controlled axes and the number of controlled spindles depend on the model and the control type as shown below.

Item	**0*i* -MD**	**0*i* -TD**	
		1-path system	**2-path system**
Maximum controlled axes (each path)	Max. 5 axes (Including Cs axes and PMC axes)	Max. 4 axes (Including Cs axes and PMC axes)	Max. 5 axes (Including Cs axes and PMC axes)
Simultaneously controlled axes (each path)	Max. 4 axes	Max. 4 axes	Max. 4 axes
Axis control by PMC	Max. 4 axes at a time (Not available on Cs axis)	Max. 4 axes at a time (Not available on Cs axis)	Max. 4 axes at a time (Not available on Cs axis)
Designation of Spindle axes (each path/total)	2 axes	2 axes	Max. 2 axes / 3 axes
Cs contouring control (each path/total)	1 axis	1 axis	Max. 1 axis / 2 axes

Item	**0*i* Mate -MD**	**0*i* Mate -TD**
Maximum controlled axes	Max. 4 axes	Max. 3 axes (Including Cs axes)
Simultaneously controlled axes	Max. 3 axes	Max. 3 axes
Axis control by PMC	-	-
Designation of Spindle axes	1 axis	1 axis
Cs contouring control	-	Max. 1 axis

1.5 AXIS CONTROL BY PMC

The PMC can directly control any given axis, independent of the CNC. By specifying an amount of travel, feedrate, and so forth from the PMC, a movement can be made along an axis independently of other axes operated under CNC control. This enables the control of turrets, pallets, index tables and other peripheral devices using any given axes of the CNC.

The following operations can be directly controlled from the PMC:

(1) Rapid traverse with a travel distance specified
(2) Cutting feed with a travel distance specified : Feed per minute
(3) Cutting feed with a travel distance specified : Feed per revolution
(4) Cutting feed with a travel distance specified : Feed per minute
(5) Dwell
(6) Continuous feed
(7) Reference position return
(8) 1st to 4th reference position return
(9) External pulse synchronization - Position coder
(10) External pulse synchronization - 1st to 3rd manual handle (3rd manual handle only for the M series)
(11) Feedrate control
(12) Torque control command
(13) Auxiliary function, Auxiliary function 2, Auxiliary function 3
(14) Selection of the machine coordinate system

1.6 Cs CONTOURING CONTROL

The Cs contouring control function positions the serial spindle using the spindle motor in conjunction with a dedicated detector mounted on the spindle.
The Cs contouring control function is higher in precision than spindle positioning (T series), and enables positioning with other servo axes. Namely, the Cs contouring control function enables linear interpolation between the spindle and servo axes.

The speed of the serial spindle is controlled by the spindle speed control function, while the spindle positioning is controlled by the Cs contouring control function ("spindle contouring control"). Spindle speed control rotates the spindle using the velocity command, while the spindle contour control rotates the spindle using the move command.
Switching between spindle speed control and Cs contouring control is performed by the signal from the PMC.
In the Cs contouring control mode, the Cs contouring control axis can be operated either manually or automatically, in the same way as normal servo axes.

1.7 NAMES OF AXES

Axis names can be assigned to axes controlled by the CNC (including PMC controlled axes). An axis name can be freely selected from 'A', 'B', 'C', 'U', 'V', 'W', 'X', 'Y', and 'Z'.

NOTE

1. The same name must not be set for multiple axes. (However, an axis name used in a path may be used in another path.)
2. When G code system A is used with a T series, U, V, and W must not be used as axis names. Only when G code system B or C is used, U, V, and W can be used as axis names.
3. When the second auxiliary function is provided, if the address for specifying the second auxiliary function is used as an axis name, the second auxiliary function is disabled.
4. When address C or A is used in chamfering/corner rounding or direct drawing dimension programming for the T series, address C or A cannot be used as an axis name.
5. When a multiple repetitive cycle (T series) is used, only the X-, Y-, and Z-axes can be used as the target axis address.

1.8 SYNCHRONOUS / COMPOSITE CONTROL (T SERIES)

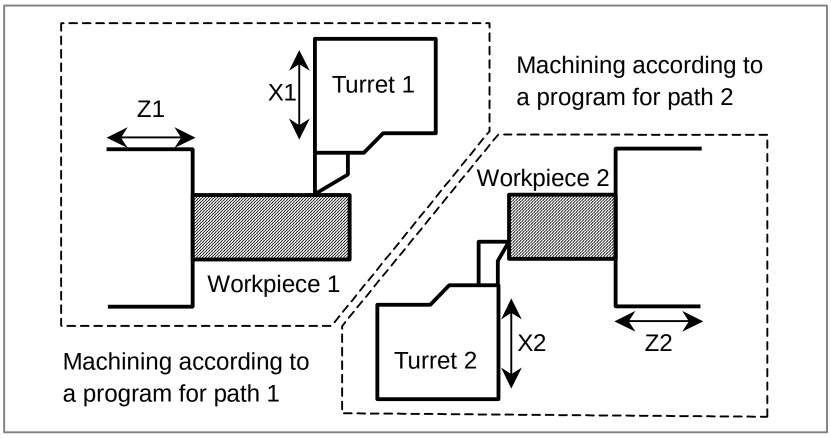
T

In 2-path control, movements are usually made on the axes of a path according to a move command for the path (independent control in each path). However, the synchronous/composite control function enables an arbitrary axis of one path to be synchronized with an arbitrary axis of another path (synchronous control).
Moreover, a move command for an arbitrary axis of one path and a move command for an arbitrary axis of another path can be exchanged with each other to make a movement on each axis (composite control).

Explanation

- Independent control in each path

Movements on the axes (X1, Z1, and so on) of path 1 are made according to a move command for path 1, and movements on the axes (X2, Z2, and so on) of path 2 are made according to a move command for path 2.



- Synchronous control

By applying a move command for an axis (master axis) to a different arbitrary axis (slave axis), the movements on the two axes can be synchronized with each other. Whether to synchronize the movement on a slave axis with the move command for the master axis or make a movement on a slave according to the command for the slave can be chosen using the signal (synchronous control selection signal) from the PMC or a program command.

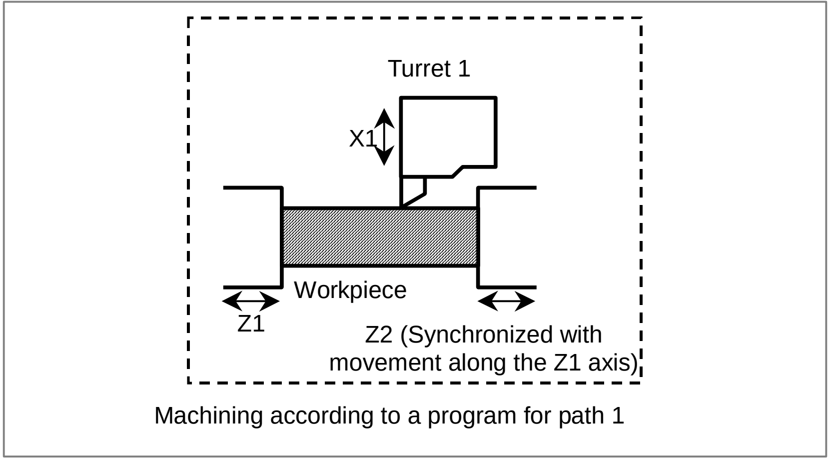
⚠ CAUTION

1 Synchronization mentioned above means that a move command for the master axis is also specified for a slave axis at the same time. Synchronization loss compensation, which detects the positional deviation between the master axis and slave axis and compensates for the deviation, is not performed. However, the positional deviation is detected at all times, and if the positional deviation exceeds a certain parameter-set value, the movement on each axis is stopped with an alarm.

2 The master axis and slave axis may belong to the same path, or the master axis may belong to one axis and the slave axis may belong to another. Moreover, multiple slave axes can be specified for one master axis.

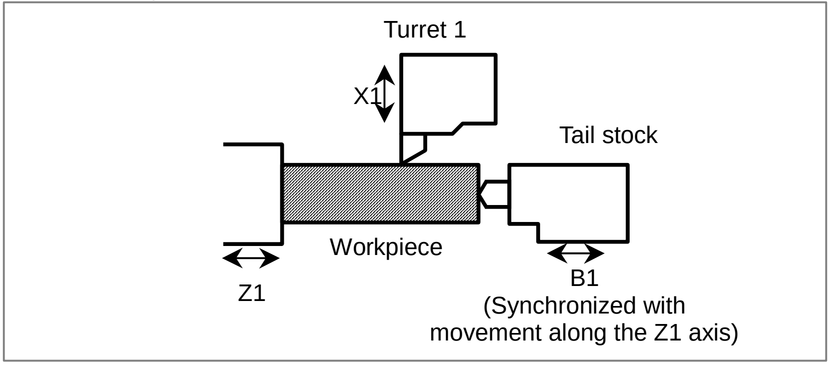
Example 1)

The Z2 axis of path 2 is synchronized with the Z1 axis of path 1.



Example 2)

The B1 axis of path 1 is synchronized with the Z1 axis of path 1.



- Composite control

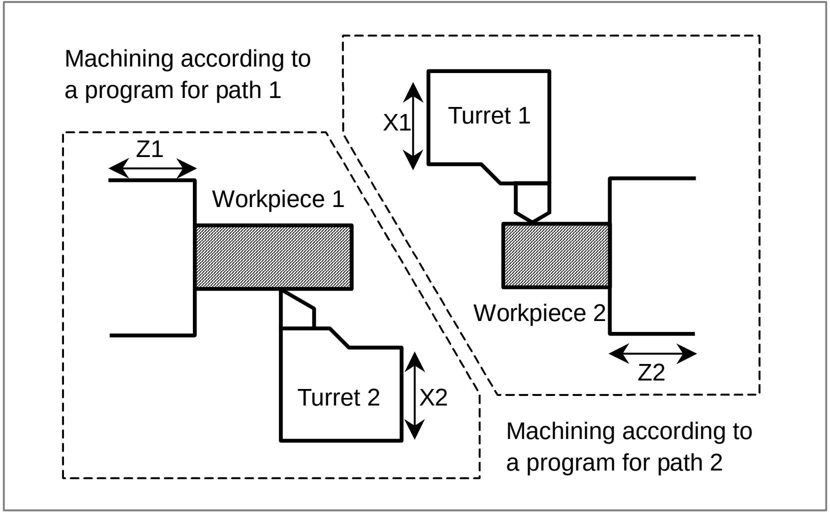
A move command for an arbitrary axis of one path and a move command for an arbitrary axis of another path can be exchanged with each other to make a movement on each axis.

Example)

A move command for the X1 axis of path 1 and a command for the X2 axis of path 2 are exchanged with each other.

The program for path 1 makes movements along the X2 axis and Z1 axis.

The program for path 2 makes movements along the X1 axis and Z2 axis.



1.9 SUPERIMPOSED CONTROL (T SERIES)

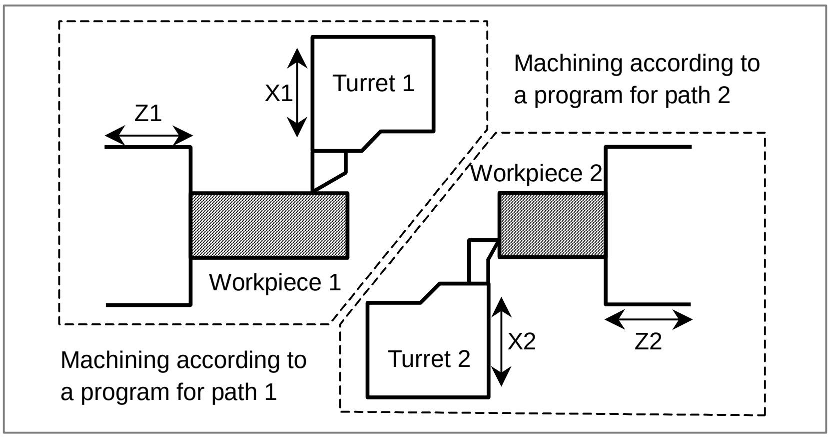
T

In 2-path control, usually, movements are made on the axes of path 1 according to a move command for path 1, and movements are made on the axes of path 2 according to a move command for path 2 (independent control in each path). However, the superimposed control function enables the travel distance on an arbitrary axis of one path to be superimposed on the travel distance on an arbitrary axis of another path.

Explanation

- Independent control in each path

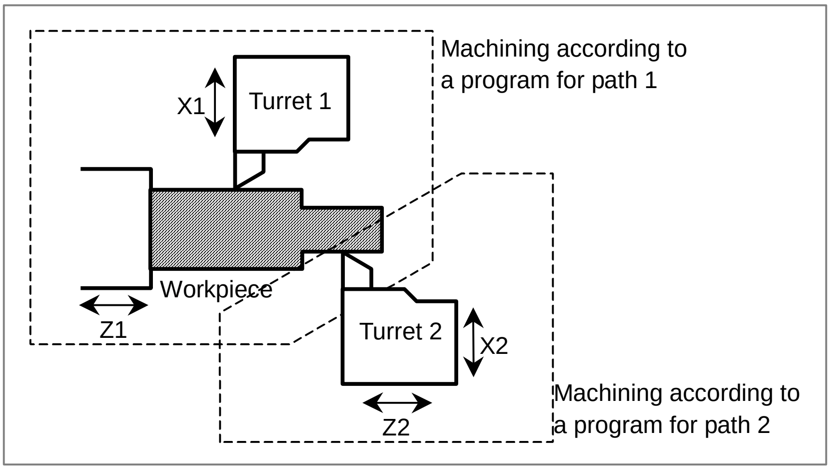
Movements on the axes (X1, Z1, and so on) of path 1 are made according to a move command for path 1, and movements on the axes (X2, Z2, and so on) of path 2 are made according to a move command for path 2.



- Superimposed Control

To the travel distance on an axis (slave axis) for which an ordinary move command is executed, the travel distance on the axis (master axis) of another path is added. Superimposed control resembles synchronous control. In superimposed control, however, a movement on the slave axis can be specified with a command for the path to which the slave axis belongs.
The master axis and slave axis may belong to the same path, or the master axis may belong to one axis and the slave axis may belong to another. Moreover, multiple slave axes can be specified for one master axis. By parameter setting, the move directions on the master axis and slave axis can be reversed from each other.
Example)

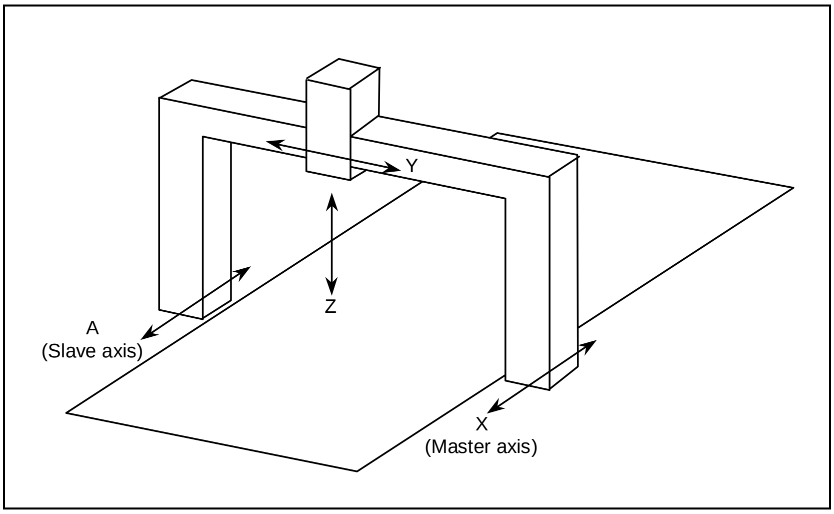
A move command for the Z1 axis of path 1 is superimposed on the travel distance on the Z2 axis of path 2.



1.10 AXIS SYNCHRONOUS CONTROL

When a movement is made along one axis by using two servo motors as in the case of a large gantry machine, a command for one axis can drive the two motors by synchronizing one motor with the other. Moreover, when a synchronous error exceeding a set value occurs, a synchronous error check can be made to issue an alarm and stop a movement along the axis.

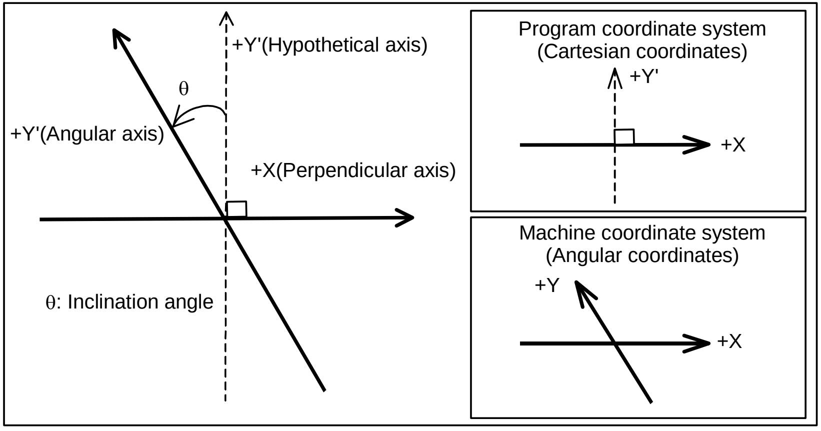
An axis used as the reference for axis synchronous control is referred to as a master axis (M-axis), and an axis along which a movement is made in synchronism with the master axis is referred to as a slave axis (S-axis).



The synchronization function can be used to perform automatic compensation that eliminates deviation of machine coordinates during release of emergency stop.
Moreover, an external signal can be used to turn synchronization on and off.

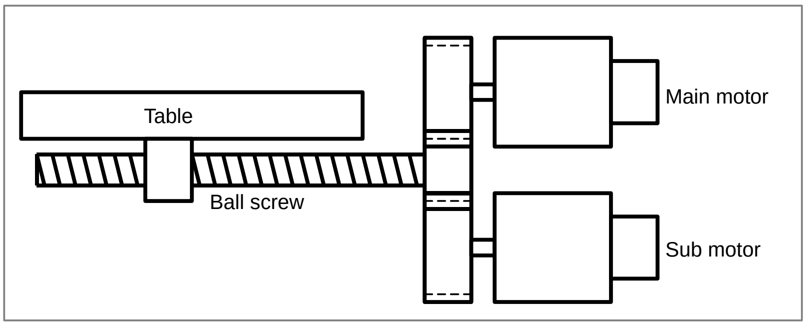
1.11 ANGULAR AXIS CONTROL

When the angular axis installed makes an angle other than 90° with the perpendicular axis, the angular axis control function controls the distance traveled along each axis according to the inclination angle as in the case where the angular axis makes 90° with the perpendicular axis.
Arbitrary axes can be specified as a set of an angular axis and perpendicular axis by parameter setting.
The actual distance traveled is controlled according to an inclination angle. However, a program, when created, assumes that the angular axis and perpendicular axis intersect at right angles.



1.12 TANDEM CONTROL

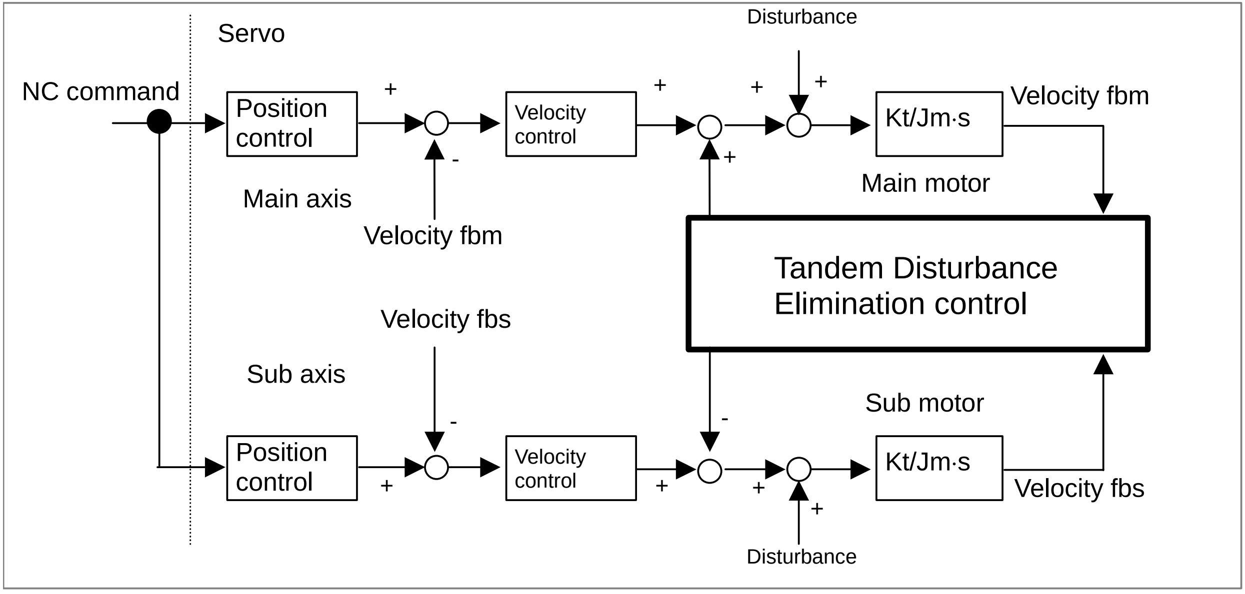
If a single motor cannot produce sufficient torque to move a large table, for example, this function allows two motors to be used. By means of this function, two motors can be used to perform movement along a single axis.
Positioning is carried out only for the master axis. The slave axis is used only to produce a torque. By means of this function, double the amount of torque can be obtained.



The CNC generally processes the two axes of tandem control as a single axis. In the management of servo parameters and the monitoring of servo alarms, however, the two axes are handled individually.

1.13 TANDEM DISTURBANCE ELIMINATION CONTROL

This function suppresses vibration caused by interference between the main axis and sub-axis in position tandem control (feed axis synchronization).



1.14 TORQUE CONTROL

For a PMC controlled axis, continuous feed based on torque control is performed.
Control on a PMC controlled axis can be switched from position control to torque control, so that the servo motor outputs torque as specified by the NC.

1.15 POLE POSITION DETECTION FUNCTION

When a synchronous built-in servo motor is driven, the magnetic pole position of the motor is detected.

NOTE

1. This function cannot be used with a vertical axis to which force is applied at all times.
2. This function cannot be used with an axis when the axis is completely locked.

1.16 CONTROL AXIS DETACH

These signals release the specified control axes from control by the CNC. When attachments are used (such as a detachable rotary table), these signals are selected according to whether the attachments are mounted. When multiple rotary tables are used in turn, the tables must use motors of the same model.

1.17 INCREMENT SYSTEM

Three types of increment systems are available as indicated in the table below, and can be chosen from by parameter setting.

Table 1.17 (a) Increment system

Name of increment system	Least input increment		Least command increment	
IS-A	0.01	mm	0.01	mm
	0.001	inch	0.001	inch
	0.01	deg	0.01	deg
IS-B	0.001	mm	0.001	mm
	0.0001	inch	0.0001	inch
	0.001	deg	0.001	deg
IS-C	0.0001	mm	0.0001	mm
	0.00001	inch	0.00001	inch
	0.0001	deg	0.0001	deg

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the parameter INM.

For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.

By parameter setting, a least input increment 10 times greater than a least command increment can be set as indicated in the table below.

Table 1.17 (b) Increment system

Name of increment system	Least input increment		Least command increment	
IS-B	0.01	mm	0.001	mm
	0.001	inch	0.0001	inch
	0.01	deg	0.001	deg
IS-C	0.001	mm	0.0001	mm
	0.0001	inch	0.00001	inch
	0.001	deg	0.0001	deg

NOTE

When the increment system is IS-A or pocket calculator type decimal point programming is used, a least input increment 10 times greater than a least command increment cannot be set.

1.18 FLEXIBLE FEED GEAR

The detection multiply (DMR) can be extended to set DMR=n/m by using two parameters n and m.

1.19 DUAL POSITION FEEDBACK

In general, a machine with a large backlash may operate stably with a semi-closed loop but may vibrate with a closed loop. This function exercises control so that such a machine can operate stably with a closed loop as in the case of a semi-closed loop.

The block diagram of dual position feedback control is shown below.

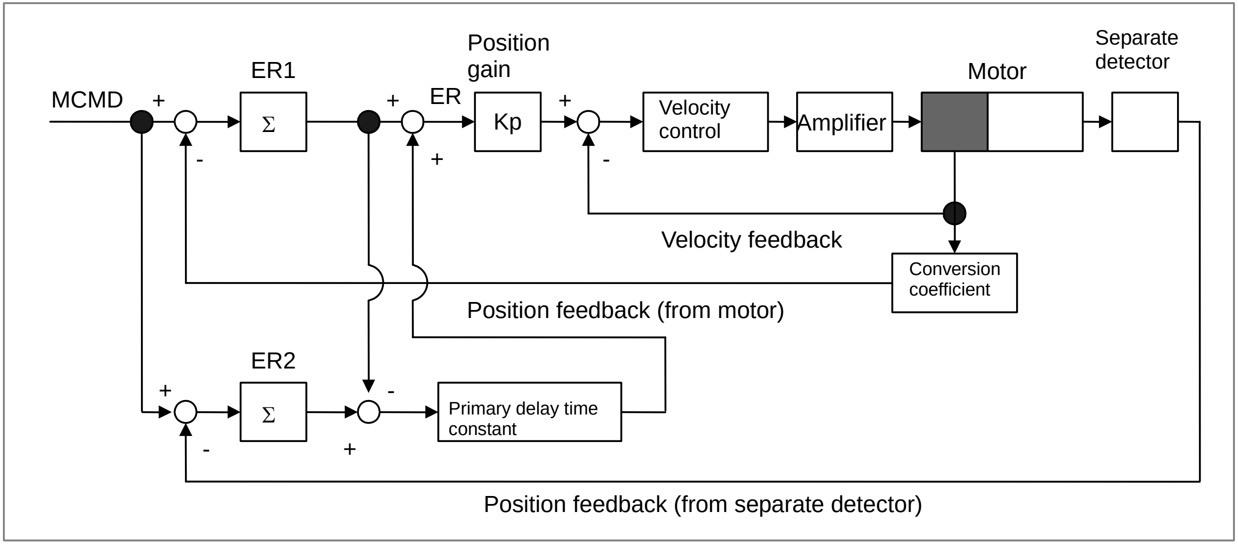


Fig. 1.19 (a) Block diagram of dual position feedback control

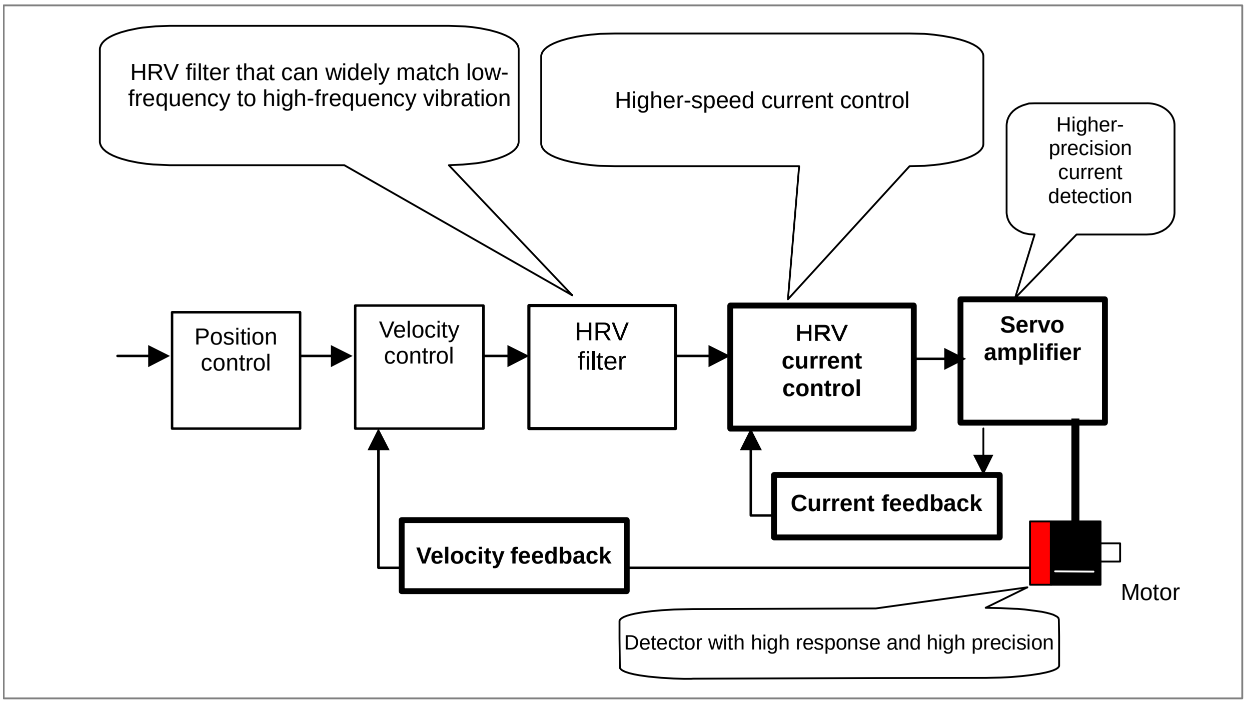
1.20 HRV CONTROL

HRV control is a digital servo current control method, and the HRV control system includes servo HRV2 and servo HRV3. By employing these control methods, even higher speed, higher precision, and higher machining speed can be achieved.

- Servo HRV control system

Servo HRV control
- Servo HRV2 control
- Servo HRV3 control

HRV control has three features:
(1) A disturbance elimination filter for eliminating low-frequency vibration from a low-rigidity machine has been developed.
(2) Smoother feed is made possible by a higher-precision servo amplifier and detector.
(3) By employing high-speed DSP, a current control cycle higher than the conventional one is made achievable with the standard servo system.



1.21 INCH/METRIC CONVERSION

Either inch or metric input (least input increment) can be selected by G code.

Format

G20 ;	Inch input
G21 ;	Metric input

Explanation

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. Do not specify this G code in the middle of a program.

Moreover, inch/metric conversion is possible in setting data setting.

After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system. The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Workpiece origin offset value
- Tool compensation value
- Unit of scale for manual pulse generator
- Movement distance in incremental feed
- Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

1.22 INTERLOCK

1.22.1 Start Lock

This function disables movement along axes during automatic operation (memory operation, DNC operation, or MDI operation).

1.22.2 All-axis Interlock

Feed on all axes can be disabled. If all-axis interlock is applied during movement, a gradual stop occurs. When the all-axes interlock signal is canceled, movement restarts.

1.22.3 Each-axis Interlock

Feed on a specified axis can be disabled, independent of other axes. If each-axis interlock is applied to an axis during cutting feed, a gradual stop occurs on all axes of the movable machine section.
When the interlock signal is canceled, movement restarts.

1.22.4 Each-axis Direction Interlock

For each axis, axial movement can be disabled in a specified axis direction only. If each-axis interlock is applied to an axis during cutting feed, a gradual stop occurs on all axes of the movable machine section. When the interlock signal is canceled, movement restarts.

1.22.5 Block Start Interlock

During automatic operation, the start of the next block can be disabled. A block whose execution has already been started continues to be executed up to the end of the block.
When block start interlock is canceled, the execution of the next block starts.

1.22.6 Cutting Block Start Interlock

During automatic operation, the start of a block including a move command other than a command for positioning can be disabled.
When cutting block start interlock is canceled, the execution of the next block starts.
If spindle rotation is specified or the spindle speed is changed, the next cutting block can be executed at the desired spindle speed by applying cutting block start interlock until the spindle reaches the desired speed.

1.23 MACHINE LOCK

1.23.1 All-axis Machine Lock

The change of the position display can be monitored without moving the machine.
When all-axis machine lock signal is set to 1, output pulses (move commands) to the servo motors are stopped in manual or automatic operation. The commands are distributed, however, updating the absolute and relative coordinates. The operator can therefore check if the commands are correct by monitoring the position display.
Machine lock during operation can be enabled even in the middle of block execution.

1.23.2 Each-axis Machine Lock

With the each-axis machine lock signal, machine lock can be applied to each axis.

1.24 EMERGENCY STOP

An emergency stop stops all commands and instantly stops the machine. Connect the emergency stop signal to both of the control unit side and servo unit side.

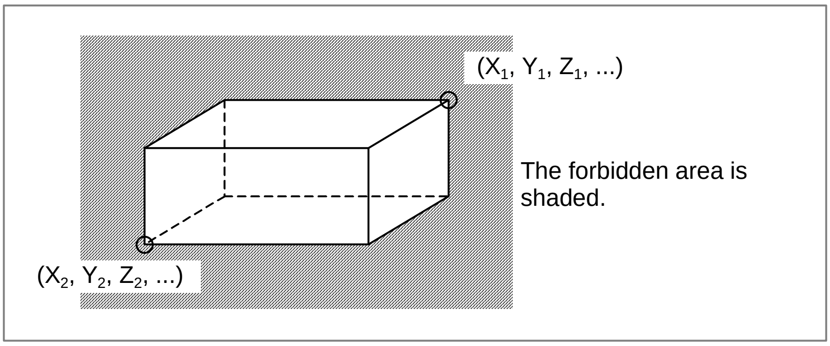
When an emergency stop is applied, servo system activation is canceled, and the servo ready signal is turned off. However, the travel distance of the machine during that time is reflected in the current position, so that the position data is not lost (follow-up). If the position detection system is normal, operation can be restarted after emergency stop cancellation without performing a reference position return operation again.

1.25 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL alarm is displayed.
An overtravel signal is provided for each direction on each axis.

1.26 STORED STROKE CHECK 1

A machine movable range is set with coordinates in the machine coordinate system in parameters. If the machine attempts to move beyond the range, it is decelerated and stopped and an alarm is displayed. This function is enabled after manual reference position return is performed after power-on.
It can be used instead of an overtravel limit switch (hardware component).
When both functions are used, both are valid.
Unlike an overtravel limit switch, this function checks whether the position at which the machine is stopped after decelerated from the current position is beyond the range.
The stroke check 1 release signal common to all axes can be set to 1 so that the control unit does not make stroke check 1.



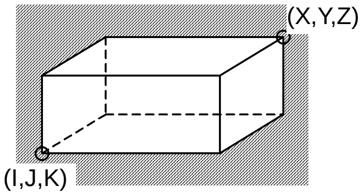
1.27 STROKE LIMIT EXTERNAL SETTING (M SERIES)

M

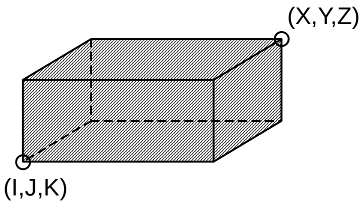
When a tool is changed, the tool tip is aligned with the end of the limit area and signals are input. This operation sets the machine position (machine coordinates) at that time as the limit position in stored stroke check parameters. A setting signal is provided for each direction of each axis.

1.28 STORED STROKE CHECK 2 (G22, G23)

For stored stroke check 2, the outside or inside of the area specified by parameters or a program is defined as the forbidden area. As a limit position, specify a distance from the origin of the machine coordinate system. This function is enabled after manual reference position return is performed at power-on. When the limits are specified in a program, they can be set for the X-, Y-, and Z-axes. For this reason, the forbidden area can be changed according to the workpiece. Whether to define the inside or outside of the specified area as the forbidden area is determined by setting the corresponding parameter.



When the outside of the specified area is defined as the forbidden area.



When the inside of the specified area is defined as the forbidden area.

Format

G22 X_ Y_ Z_ I_ J_ K_ ; Stored stroke check 2 on

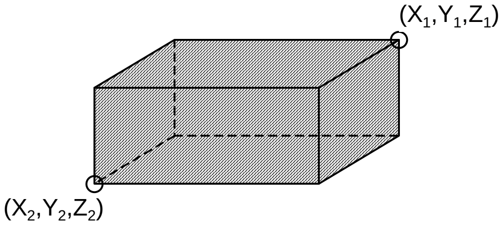
X, Y, Z : Coordinates in the + direction of stored stroke check 2
I, J, K : Coordinates in the - direction of stored stroke check 2

The address is X, Y, Z, I, J, or K. X and I, Y and J, and Z and K specify a forbidden area for the X-axis, Y-axis, and Z-axis of the basic three axes, respectively. If an address is omitted, a stroke check is make according to the parameter settings.

G23 ; Stored stroke check 2 off

1.29 STORED STROKE CHECK 3

The inside of a parameter-set area is a forbidden area.



When the inside of the specified area is defined as the forbidden area.

1.30 STROKE LIMIT CHECK BEFORE MOVE

During automatic operation, before the movement specified by a given block is started, whether the tool enters the inhibited area defined by stored stroke check 1, 2, or 3 is checked by determining the coordinate of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the inhibited area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

1.31 CHUCK AND TAIL STOCK BARRIER (T SERIES)

T

The chuck and tail stock barrier function prevents damage to the machine by checking whether the tool tip interferes with either the chuck or tail stock.

Specify an area into which the tool may not enter (entry-prohibition area). This is done using the special setting screen, according to the shapes of the chuck and tail stock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.

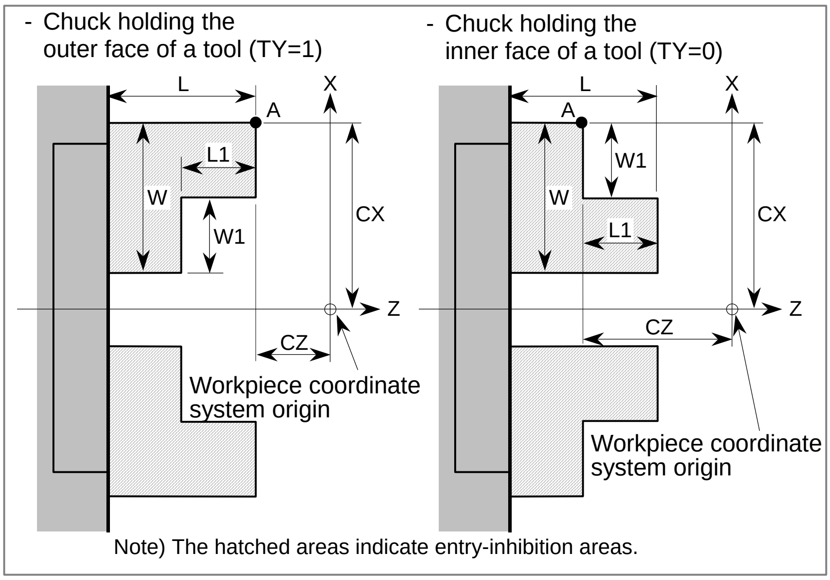
The tool can be removed from the prohibited area only by retracting it in the direction from which the tool entered the area.

This function can be enabled or disabled by G22 (stored stroke check 2 on), G23 (stored stroke check 2 off), and a machine-side signal.

G code	Tail stock barrier signal	Tail stock barrier	Chuck barrier
G22	0	Valid	Valid
	1	Invalid	Valid
G23	0	Invalid	Invalid
	1	Invalid	Invalid

Explanation

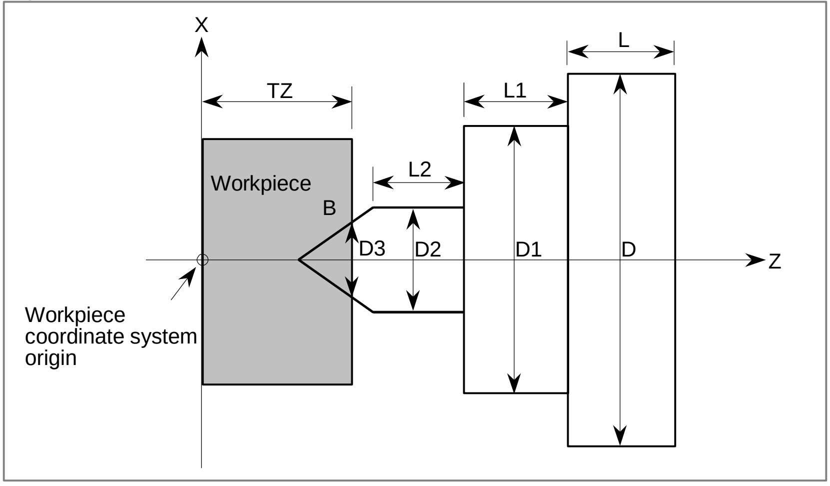
- Chuck figure definition



Note) The hatched areas indicate entry-inhibition areas.

Symbol	Description
TY	Chuck-shape selection (0: Holding the inner face of a tool, 1: Holding the outer face of a tool)
CX	Chuck position (along X-axis)
CZ	Chuck position (along Z-axis)
L	Length of chuck jaws
W	Depth of chuck jaws (radius)
L1	Holding length of chuck jaws
W1	Holding depth of chuck jaws (radius)

- Tail stock figure definition



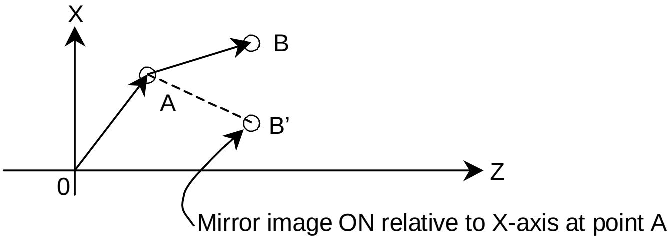
Symbol	Description
TZ	Tail stock position (along the Z-axis)
L	Tail stock length
D	Tail stock diameter
L1	Tail stock length (1)
D1	Tail stock diameter (1)
L2	Tail stock length (2)
D2	Tail stock diameter (2)
D3	Tail stock hole diameter (3)

1.32 CHECKING THE STORED STROKE LIMIT DURING THE TIME FROM POWER–ON TO THE REFERENCE POSITION ESTABLISHMENT

This function stores the machine coordinates present immediately before the power is turned off. Therefore, immediately after the power is turned on again, this function can restore the approximate machine coordinates and enables the function for checking the stored stroke limit during the time from power–on to the reference position establishment.
Even before the reference position is established by manual reference position return, the stored stroke limit check can be performed using approximate machine coordinates.

1.33 MIRROR IMAGE

Mirror image can be applied to each axis, either by signals or by parameters (setting input is acceptable). All movement directions are reversed during automatic operation along axes to which a mirror image is applied.



Mirror image (Example for lathe system)

However, the following directions are not reversed:

- Direction of movement of the machine coordinate system selection (G53), direction of manual operation, and direction of movement, from the intermediate position to the reference position during automatic reference position return (for the M series and T series)
- Approach direction for single direction positioning (G60) and shift direction for boring cycles (G76 and G87) (for the M series only)

Signals indicate whether mirror image is applied to each axis. System variable contains the same information.

1.34 FOLLOW-UP

If the machine moves in the state in which position control on controlled axes is disabled (during servo-off, emergency stop, or servo alarm), feedback pulses are accumulated in the error counter. The CNC reflects the machine movement corresponding to the error count in the current position managed by the CNC. This operation is referred to as follow-up. When follow-up is performed, the current position managed by the CNC does not shift from the actual machine position.

So, operation can be restarted after emergency stop cancellation or servo alarm cancellation without performing a reference position return operation again.

You can select whether to perform follow-up for axes when the servo is turned off.

Follow-up is always performed during emergency stop or a servo alarm.

1.35 SERVO OFF / MECHANICAL HANDLE FEED

Place the controlled axes in the servo off state, stop the current to the servo motor, which disables position control. However, the position detection feature functions continuously, so the current position is not lost.

These signals are used to prevent the servo motors from overloading when the tools on the axes are mechanically clamped under certain machining conditions on the machine, or to move the machine by driving the motors by mechanical handles.

1.36 CHAMFERING ON/OFF (T SERIES)

T

In the threading cycle (G76), which is a multiple repetitive cycle for turning, and in the threading cycle (G92), which is a canned cycle, threading can be selected with the chamfering signal.

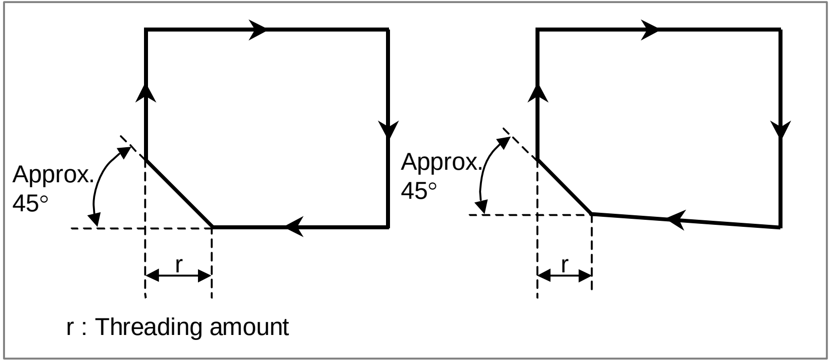


Fig. 1.36 (a) Straight threading and taper threading

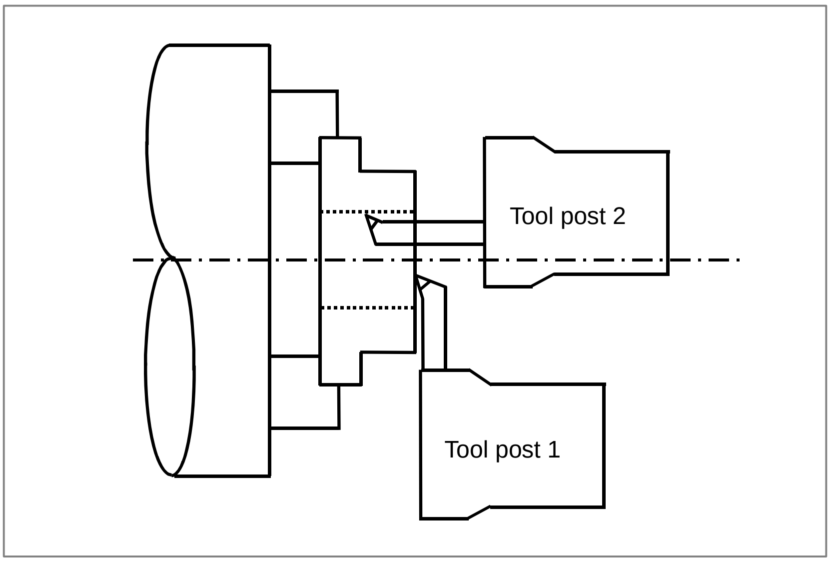
1.37 INTERFERENCE CHECK FOR EACH PATH (T SERIES)

T

When tool posts on two paths machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur.

If such a command that causes tool posts of paths to interfere with each other is specified, this function gradually stops the tool posts before the tool posts actually interfere with each other.

The contours and shapes of the tool posts on individual paths are checked to determine whether or not an interference occurs.



To make a path interference check, data including the relationships between the tool posts on individual paths and interference forbidden areas (that is, tool shapes) needs to be set.

Based on the interference forbidden areas of the tool currently selected on the tool post of each path and tool posts, an inter-path interference check determines whether the tools and tool posts interfere with each other, by checking whether those forbidden areas overlap each other as a result of movement of each tool post. If an interference occurs, the interfering tool posts gradually stop with an alarm.

1.38 UNEXPECTED DISTURBANCE TORQUE DETECTION FUNCTION

Machine collision, defective, and damaged cutters cause a large load torque on the servo and spindle motors, compared with normal rapid traverse or cutting feed. This function detects the disturbance torque on the motors and sends this value as an estimated load torque to the PMC. If the detected disturbance torque value is abnormally great compared with the value specified in the parameter, the function stops the servo motor as early as possible or reverses the motor by an appropriate value specified in a parameter, in order to minimize possible damage to the machine.

The unexpected disturbance torque detection function is further divided as follows:

(1) Estimated disturbance torque output function
    The CNC is always calculating the estimated disturbance torque for the motor (excluding acceleration/deceleration torque). The estimated disturbance torque output function enables the PMC to read the calculated torque using the window function.

(2) Unexpected disturbance torque detection alarm function
    This function stops motors or reverses them by an amount specified in a parameter, causing the CNC to output an alarm, whenever the disturbance torque is greater than the value specified in a parameter. (The function to reverse motors is effective only for servo motors.)

(3) Unexpected disturbance torque detection group function
This function stops the motor immediately if the load torque obtained by the estimated disturbance torque output function is greater than the parameter setting when servo axes are classified into arbitrary groups. This function stops all axes (including the axes for which the group number is set to 0) classified into the same group by the parameter sets them to the interlock state. Depending on the parameter setting, this function reverses the motors by an amount specified in the parameter and sets all axes in the same group to the interlock state.

NOTE
Either the unexpected disturbance torque detection alarm function or the unexpected disturbance torque detection group function is selected by parameter setting. Both functions cannot be used at the same time.

Unexpected disturbance torque detection can also be disabled only for specific axes by using parameter for the unexpected disturbance torque detection function and unexpected disturbance torque detection ignore signals. (This function is effective only for servo motors.)

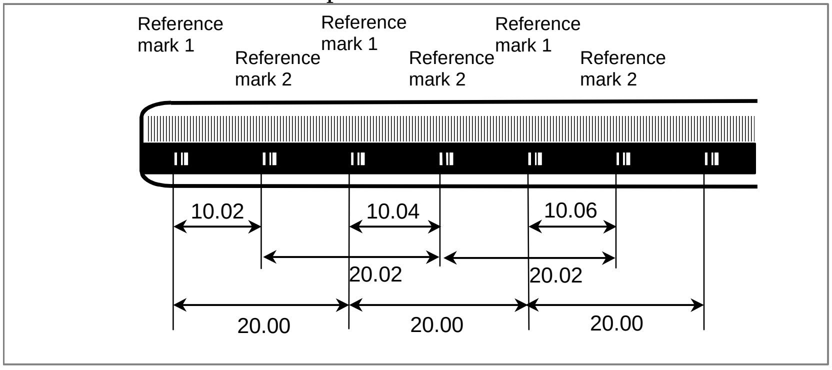
1.39 POSITION SWITCH

Position switch signals can be output to the PMC while the machine coordinates along a controlled axes are within a parameter-specified ranges.
Using parameters, specify arbitrary controlled axes and machine coordinate operating ranges for which position switch signals are output.
Up to 10 position switch signals can be output.
Parameter can be set to use up to 16 position switch signals.

1.40 LINEAR SCALE WITH ABSOLUTE ADDRESS REFERENCE MARK

1.40.1 Linear Scale Interface with Absolute Address Reference Mark

With linear scale with absolute address reference mark, an absolute position can be identified if the interval of reference marks is known, because the intervals of two reference marks (one-rotation signals) differ from each other by a certain distance. This CNC measures one-rotation signal intervals by making a slight movement on an axis to calculate an absolute position. So, a reference position can be established without making a movement to the reference position on the axis.



1.40.2 Linear Scale with Absolute Address Reference Mark Expansion

When a G00 command is specified or a move command based on jog feed is specified, this function enables a reference mark interval measurement to be made automatically in order to establish a reference position.

1.41 LINEAR SCALE WITH DISTANCE-CODED REFERENCE MARKS (SERIAL)

By using High-resolution serial output circuit for the linear scale with distance-coded reference marks (serial), the CNC measures the interval of referenced mark by axis moving of short distance and determines the absolute position.
This function enables high-speed high-precision detection by using High-resolution serial output circuit.
It is available that using maximum stroke 30 meters length.

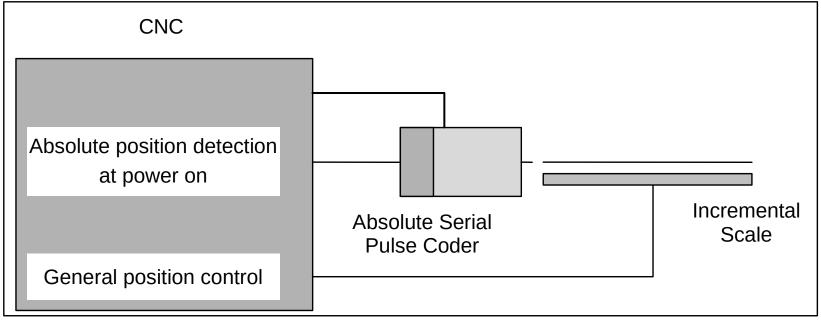
1.42 ABSOLUTE POSITION DETECTION

An absolute position detector (absolute pulse coder) is an incremental pulse coder with an absolute counter. It detects the absolute position based on the value of the absolute counter. For an axis on which an absolute position detector is mounted, no reference position return is required at power-on because the machine position is always stored with batteries if the power to the CNC is turned off.
When the machine position has been brought into correspondence with the absolute position detector, the current position is read from the absolute counter at CNC power on and the machine and workpiece coordinate systems are automatically set using the value. In this case, you can immediately start automatic operation.
Restrictions described in the OPERATOR'S MANUAL and others that include those listed below are removed:

- "Reference position return is required after power-on."
- "The CNC can be used after reference position return is performed after power-on."

1.43 TEMPORARY ABSOLUTE COORDINATE SETTING

In the full closed system with an inner absolute position pulse coder (serial pulse coder) and an incremental scale, the position is set by using absolute position data from the inner absolute position pulse coder at the power on sequence. After that, the position is controlled with incremental data from the incremental scale. The position just after power on sequence is rough, and the manual reference position return is required to get the accurate position.



The system with the Temporary Absolute Coordinate Setting

With this function, the position at the power on is rough, but the following functions are available before the reference position return.

- Stroke limit check
- Position switch

1.44 FUNCTION OF DECELERATION STOP IN CASE OF POWER FAILURE

If a power failure occurs during an axial movement, this function stops the movement by decreasing the speed on each axis at a rate specified in parameter. This function prevents the machine from being damaged by an overrun.

1.45 CORRESPONDENCE OF ROTARY SCALE WITHOUT ROTARY DATA

This manual describes how to deal with an absolute position detector (absolute pulse coder) or a rotary scale with distance-coded reference marks (serial), when the rotary scale without rotary data is used, such as Heidenhain rotary scale RCN223, RCN723, RCN220, or Futaba rotary scale FRR902L3DB.

2 OPERATION

Chapter 2, "OPERATION", consists of the following sections:

2.1 OPERATION MODE

2.1.1 Automatic Operation (Memory Operation)

Program registered in the memory can be executed.

2.1.2 MDI Operation

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

2.1.3 DNC Operation

A program can be executed while being read from the input device connected with the reader/punch interface.

2.1.4 DNC Operation with Memory Card

A program can be executed while being read from the memory card.

2.1.5 Schedule Operation

Programs can be executed by specifying their program file numbers on the memory card in the sequence in which they are to be executed and the number of times that they are to be executed.

2.2 PROGRAM SEARCH

By operating the MDI panel, a program to be executed can be selected from the programs stored in the program memory.

2.3 SEQUENCE NUMBER SEARCH

By operating the MDI panel, a block can be selected according to a sequence number in the currently selected program in the program memory.
When a program is to be executed starting with a block in the middle of the program, the sequence number of the block is to be specified to search for the sequence number.

2.4 SEQUENCE NUMBER COMPARISON AND STOP

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed. By setting operation, the operator can set a sequence number through the MDI panel. This function is useful for checking a program, because the program can be stopped at a desired position without modifying the program.

2.5 PROGRAM RESTART

When the tool is broken during automatic operation, or when a machining operation interrupted by a holiday is to be restarted after the holiday, you can restart machining from a desired block by specifying the sequence number of the block or the number of blocks from the beginning of the program to the block where machining is to restart.
This function can also be used as a high-speed program check function.
P type : Restart after a tool is broken down
Q type : Restart after the power is turned off (after a holiday, etc.) or after emergency stop is canceled

2.6 MALFUNCTION PREVENT FUNCTIONS

These functions monitor the CNC internal status and check that related data is within the allowable range. If an invalid state due to a deteriorated hardware component or noise is detected, these functions stop the machine with an alarm to prevent any malfunction.

The following malfunction prevention functions are available:
- Checking the maximum speed of the servo motor
- Checking the maximum acceleration of the servo motor
- Checking the maximum speed of the spindle motor
- Checking the stored stroke limit at the end point
- Monitoring execution of NC command analysis
- Monitoring execution of acceleration/deceleration after interpolation

2.7 WRONG OPERATION PREVENTION FUNCTION

An improper tool offset setting or an improper operation of the machine can result in the workpiece being cut inadequately or the tool being damaged. Also, if data is lost due to an operation mistake, it takes extra time to recover from the mistake.

The wrong operation prevention functions described below are meant to prevent the operator from performing any unintended operation.

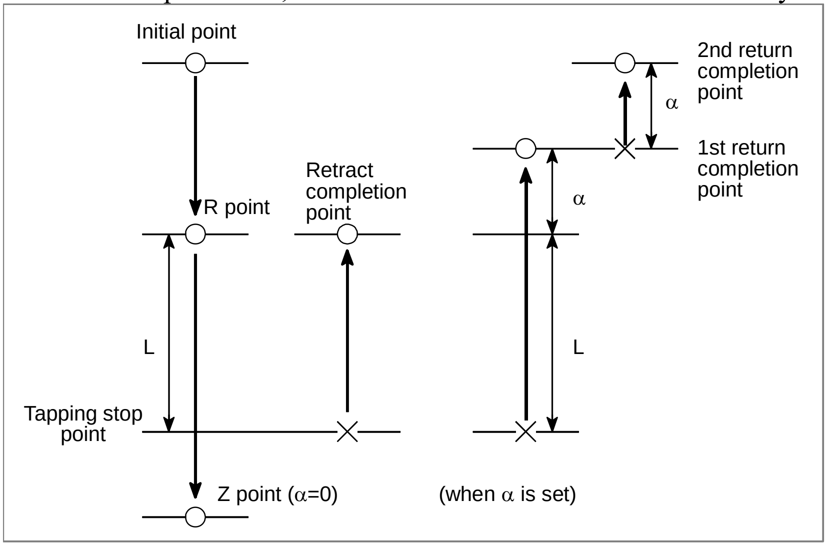
1 Functions that are used when data is set
  - Data check to verify that the offset data is within the valid setting range
  - Incremental input operation confirmation
  - Prohibition of the absolute input by the soft key to prevent any improper absolute or incremental input operation
  - Confirmation of any operation of deleting the program or all data
  - Confirmation of a data update during the data setting process

2 Functions that are used when the program is executed
  - Highlighting of updated modal information
  - Display of the executed block status prior to the program execution
  - Display of the axis status, such as the mirror image function enabled or the interlock function enabled
  - Check for starting from the middle of the program
  - Data check to verify that the offset data is within the effective setting range
  - Maximum incremental value check

2.8 RETRACTION FOR RIGID TAPPING (M SERIES)

M

When rigid tapping is stopped, either as a result of an emergency stop or 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 retraction signal is input, the tap is removed from the hole, based on the stored information. The tap is pulled toward the R point. When a retract value α is set in parameter, the retraction distance can be increased by α.



2.9 BUFFER REGISTER

The CNC contains a buffer register so that a pause in CNC command operation caused by program reading and preprocessing time can be suppressed.

2.10 DRY RUN

The tool is moved at a constant feedrate regardless of the feedrate specified in the program. This function is used, for example, to check the movement of the tool without a workpiece.
Dry run is valid only for automatic operation.

2.11 SINGLE BLOCK

When the single block signal is set to 1 during automatic operation, the CNC enters the automatic operation stop state after executing the current block. In subsequent automatic operation, the CNC enters the automatic operation stop state after executing each block in the program. When the single block signal is set to 0, normal automatic operation is restored.
Single block operation is valid only for automatic operation.

2.12 JOG FEED

- Jog feed

Each axis can be moved in the + or - direction for the time the button is pressed. Feedrate is the parameter set speed with override of:
0 to 655.34%, 0.01% step.
The parameter set feedrate can be set to each axis.

- Manual rapid traverse

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.

2.13 MANUAL REFERENCE POSITION RETURN

Positioning to the reference position can be done by manual operation.
With jog feed mode, manual reference position return signals, and signal for selecting manual reference position return axis on, the tool the machine is turned on, it decelerates, and when it is turned off again, it stops at the first grid point.
Upon completion of manual reference position return operation, the reference position return completion signal is sent.
By performing manual reference position return, the machine coordinate system and the workpiece coordinate system is established.
A grid method of manual reference position return is available.
The grid method can shift the reference position with the grid shift function.

2.14 REFERENCE POSITION SETTING WITHOUT DOG

This function moves the machine to around the reference position set for each axis in the jog 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.

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.

- Procedure for setting the reference position

(1) Feed the tool, along the axis for which the reference position is to be set, by jog feed in the reference position return direction. Stop the tool near the reference position, but do not exceed the reference position.
(2) Enter manual reference position return mode, then set 1 for the feed axis direction selection signal (for the positive or negative direction) for the axis.
(3) The CNC positions the tool to the nearest grid line (based on one-rotation signals from the position detector) in the reference position return direction specified with parameter. The point at which the tool is positioned becomes the reference position.
(4) The CNC checks that the tool is positioned to within the in-position area, then sets the completion signal for reference position return and the reference position establishment signal to 1.

- Manual reference position return

When the feed axis and direction selection signal is set to 1 in manual reference position return mode after the reference position has been established, the tool is positioned to the reference position regardless of the direction specified by the feed axis and direction selection signal. The completion signal for reference position return is then set to 1.

2.15 REFERENCE POSITION SETTING WITH MECHANICAL STOPPER BY GRID METHOD

This function automates the procedure of butting the tool against a mechanical stopper on an axis to set a reference position. The purpose of this function is to eliminate the variations in reference position setting that arise depending on the operator, and to minimize work required to make fine adjustments after reference position setting.
Select the axis for which the reference position is to be set, then perform cycle start. Then, the following operations are performed automatically:

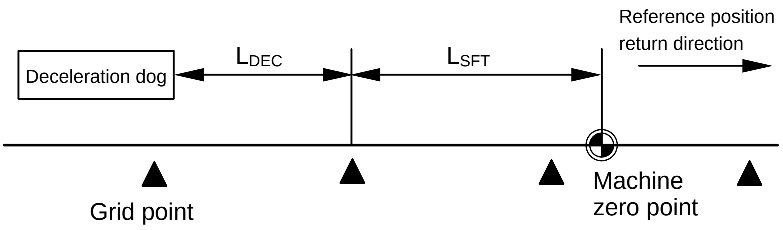
1. The torque (force) of the selected axis is reduced to make the butting feedrate constant, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
2. Again, the torque (force) of the selected axis is reduced, and the tool is butted against the mechanical stopper. Then, the tool is withdrawn a parameter-set distance from the mechanical stopper.
3. The withdrawal point on the axis is set as the reference position.

2.16 REFERENCE POSITION RETURN FEEDRATE SETTING

If a value is set with the parameter for reference position return feedrate setting, the manual and automatic reference position return feedrates and automatic rapid traverse rate before coordinate system establishment follow the parameter settings for each axis.
After a reference position return operation is completed and a reference position is established, the manual reference position return feedrate set in the parameter for each axis is used.

2.17 REFERENCE POSITION SHIFT

In reference position return operation based on the grid method, the reference position can be shifted without moving the deceleration dogs, by setting a reference position shift amount in a parameter.
This function eliminates the need for adjusting the deceleration dogs at the time of reference position adjustment, so that the time required for reference position adjustment can be reduced remarkably.



L_{DEC} : Distance from the turning-off of the deceleration dog to the first grid point (grid point when the reference position shift amount is 0)

L_{SFT} : Reference position shift amount

2.18 MANUAL HANDLE FEED

2.18.1 Manual Handle Feed (1 Unit)

By rotating the manual pulse generator on the machine operator's panel in handle mode, the axis feed corresponding to the amount of rotation can be performed. A desired axis can be selected using the handle axis selection switch.

2.18.2 Manual Handle Feed (2/3 Units)

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 can be done at a time.

2.18.3 Manual Handle Feed Magnification

The manual pulse generator generates 100 pulses per rotation. Move
amount per pulse can be specified from the following magnifications:
×1, ×10, ×m, ×m.
m and n are parameters set values of 1 to 2000.
Move distance is :
(Least command increment) × (magnification)
By parameter setting, the selected magnification can be multiplied by 10.

Table 2.18.3 (a) Travel increment in manual handle feed

Increment system	Travel increment	
	Metric input (mm)	Inch input (inch)
IS-A	0.01, 0.1, m/100, n/100	0.001, 0.01, m/1000, n/1000
IS-B	0.001, 0.01, m/1000, n/1000	0.0001, 0.001, m/10000, n/10000
IS-C	0.0001, 0.001, m/10000, n/10000	0.00001, 0.0001, m/100000, n/100000

2.19 MANUAL HANDLE INTERRUPTION

By rotating the manual pulse generator in the automatic operation mode (manual data input, DNC operation, or memory operation) or in the memory editing mode, handle feed can be superimposed on movement by automatic operation. A handle interruption axis is selected using the manual handle interruption axis selection signal.

The minimum unit of travel distance per scale division is the least input increment.

2.20 MANUAL INTERVENTION AND RETURN

When you perform a restart after stopping the axis movement by feed hold during automatic operation and intervening in the tool position manually for such purposes as to check the cutting surface, this function automatically returns the tool to the position where it was prior to manual intervention, before resuming automatic operation.

2.21 FANUC SERVO MOTOR β Series (I/O OPTION) MANUAL HANDLE INTERFACE

This function controls manual handle feed for β servo unit by using a manual pulse generator on the NC control side. Pulses from manual pulse generator are transferred from the NC control side to β servo unit through I/O Link. Still further, this function can control the magnification of pulses by changing the parameter. This function is available on the peripheral control interface. This function is the optional one.

2.22 INCREMENTAL FEED

Pressing a certain button on the machine operator's panel changes a specified travel distance in the + direction or - direction. The minimum unit of travel distance is the least input increment. It can be multiplied by a magnification of 10, 100, or 1000. By parameter setting, the selected magnification can be multiplied by 10. The table below lists travel increments.

Table 2.22 (a) Travel increments in incremental feed

Increment system	Travel increment	
	Metric input (mm)	Inch input (inch)
IS-A	0.01, 0.1, 1.0, 10.0	0.001, 0.01, 0.1, 1.0
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

⚠ **CAUTION**

For an axis with its diameter specified in incremental feed, the travel distance is based on the diameter value.

2.23 JOG AND HANDLE SIMULTANEOUS MODE

Usually, manual handle feed is enabled only when the operation mode is set to the manual handle feed mode. By parameter setting, however, manual handle feed can be performed even in the jog feed mode. Note, however, that jog feed and manual handle feed cannot be performed at the same time.

Manual handle feed can be performed only when jog feed is not being performed (when no movement is being made on the axis).

2.24 REFERENCE POSITION SIGNAL OUTPUT FUNCTION

If, after the establishment of the reference position on each axis, the coordinates in the machine coordinate system match the reference position, this function outputs a signal as the DO signal of the PMC.

If the coordinates in the machine coordinate system match the second to fourth reference positions, this function outputs a signal as the DO signal of the PMC, respectively.

2.25 MANUAL HANDLE RETRACE

In this function, the program can be executed both forward and backward with a manual handle (manual pulse generator) under automatic operation.
Therefore, errors of a program, interference, and so on can be checked easily by working a machine actually.

3 INTERPOLATION FUNCTION

Chapter 3, "INTERPOLATION FUNCTION", consists of the following sections:

3.1 NANO INTERPOLATION

As the unit of output to the servo system, the detection unit is usually used. However, this function enables output to the servo system to be performed using the detection unit multiplied by 1/1000 for improved precision in machining. This function remarkably improves surface roughness in particular.
However, positioning precision depends on machine conditions such as the resolution of the detector.

NOTE

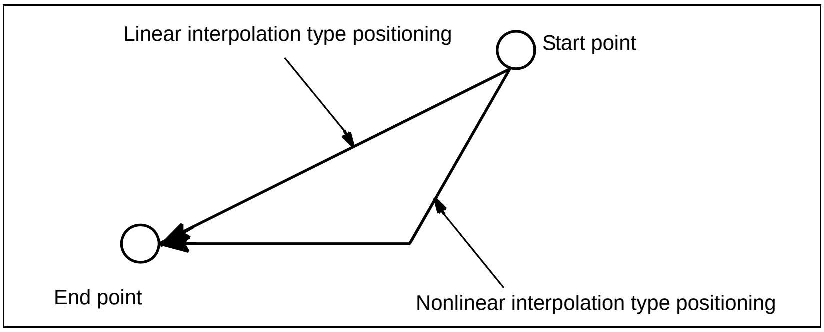
This function is most effective if the resolution of a detector is finer than the detection unit. Even if the resolution of a detector is the same as the detection unit (as in the case of a closed loop), this function is effective when the feed-forward function is used.

3.2 POSITIONING

This command moves a tool to the position in the workpiece coordinate system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

Either of the following tool paths can be selected according to parameter.

- Nonlinear interpolation type positioning
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- Linear interpolation type positioning.
  The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



In the positioning mode based on G00, the machine is accelerated up to a specified feedrate at the start of a block. At the end of the block, the machine is decelerated to a stop to make an in-position check (to see if the machine has arrived at a specified position).
By parameter setting, an in-position check for each block can be disabled.

Format

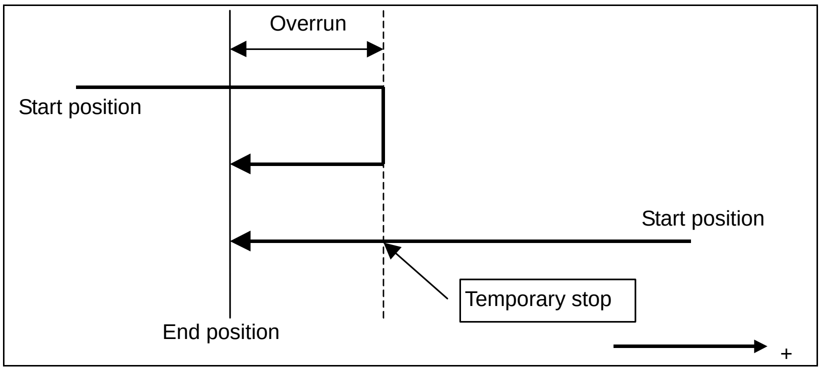
G00 IP_ ;

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.

3.3 SINGLE DIRECTION POSITIONING (M SERIES)

M

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



Direction positioning process

An overrun and a positioning direction are set by the parameter.
Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is a one-shot G code, can also be used as a model G code in group 01 by parameter setting. This setting can eliminate specifying a G60 command for every block. Other specifications are the same as those for an one-shot G60 command.

When an one-shot G code is specified in the single direction positioning mode, the one-shot G command is effective like G codes in group 01.

Format

G60 IP_ ;

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.

3.4 EXACT STOP MODE

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 until G62 (automatic corner override) (M series), G63 (tapping mode), or G64 (cutting mode), is commanded.

3.5 TAPPING MODE

When G63 is commanded, feedrate 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. This G63 is valid until G61 (exact stop mode), G62 (automatic corner override) (M series), or G64 (cutting mode) is commanded.

3.6 CUTTING MODE

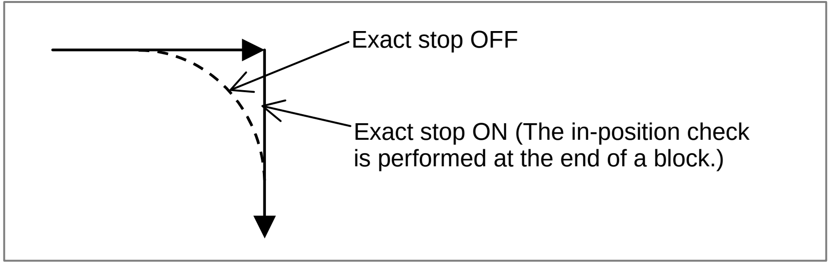
When G64 is commanded, deceleration at the end point of each block thereafter is not performed and cutting goes on to the next block. This command is valid until G61 (exact stop mode), G62 (automatic corner override) (M series), or G63 (tapping mode) is commanded. However, in G64 mode, feedrate is decelerated to zero and in-position check is performed in the following case;

- Positioning mode (G00, G60)
- Block with exact stop check (G09)
- Next block is a block without movement command

3.7 EXACT STOP

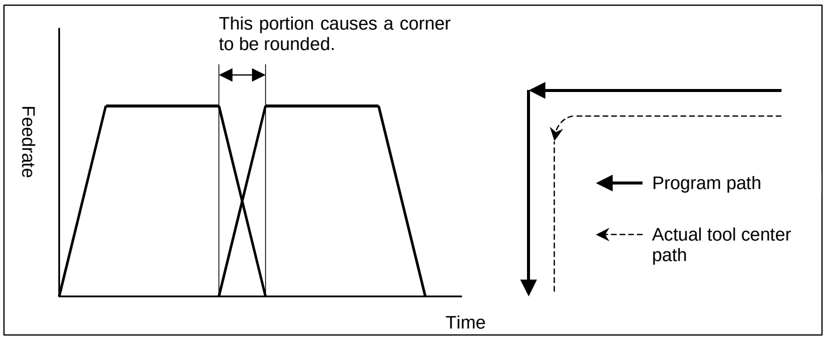
The feedrate of a block specifying continuous cutting feed can be controlled using NC commands as described below.

Move command in blocks commanded with G09 decelerates at the end point, and in–position check is performed. When the feed motor enters the effective area, the movement of the next block starts. This function is used when sharp edges are required for workpiece corners in cutting feed.

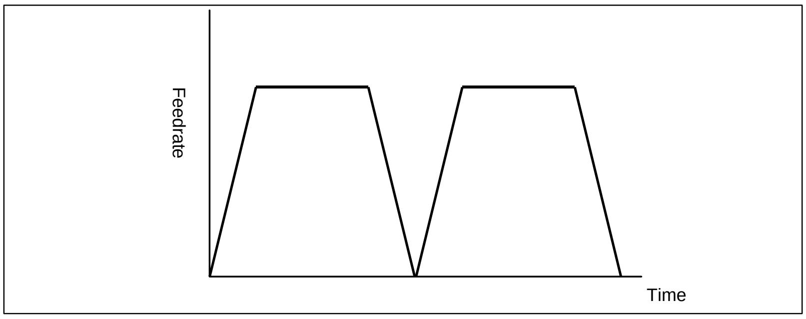


3.8 IN-POSITION CHECK SIGNAL

On general CNCs, the feedrate during cutting feed never becomes 0 between two successive blocks.
So, a corner may be rounded.

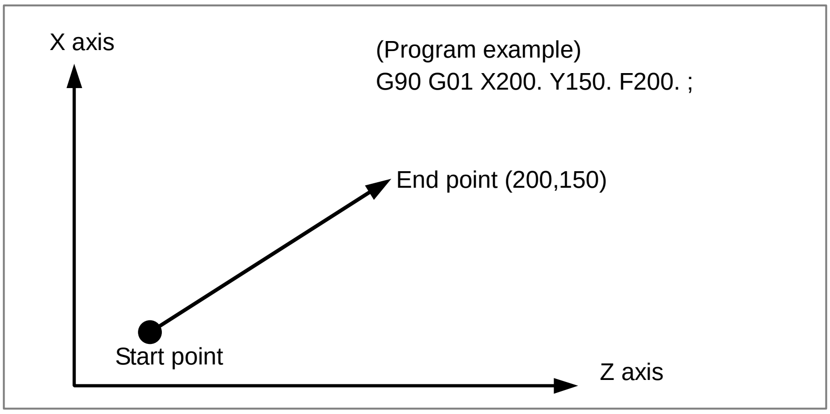


By using the in-position check signal, control can be exercised so that machining can proceed to the next block after checking that the acceleration/deceleration processing of the previous block is completed and the effective area has been entered.



3.9 LINEAR INTERPOLATION

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.



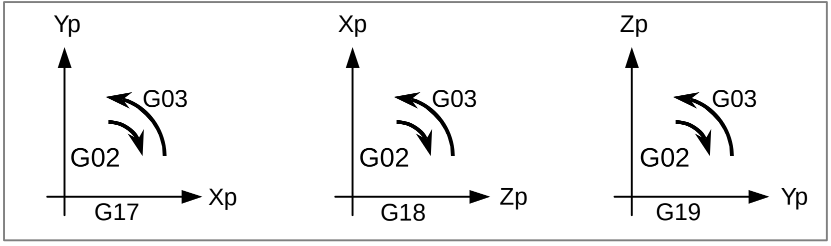
Format

G01 IP_ F_ ;

IP_ : For an absolute command, the coordinates of an end point, and for an incremental command, the distance the tool moves.
F_ : Speed of tool feed (Feedrate)

3.10 CIRCULAR INTERPOLATION

In the M series, circular interpolation of optional angle from 0° to 360° can be specified.
In the T series, circular interpolation of optional angle from 0° to 180° can be specified.
G02: Clockwise (CW) circular interpolation
G03: Counterclockwise (CCW) circular interpolation



Feedrate 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 where Xp: X axis or its parallel axis
G18: Zp-Xp plane Yp: Y axis or its parallel axis
G19: Yp-Zp plane 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 in the XpYp plane

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

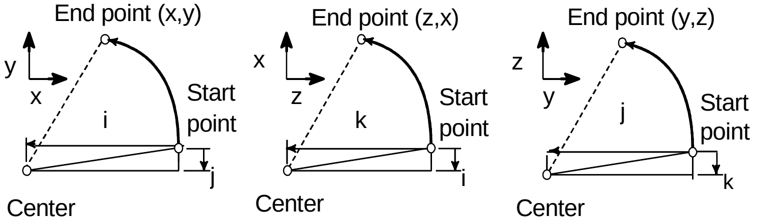
Arc in the ZpXp plane

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

Arc in the YpZp plane

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

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.)



3.11 DWELL

By specifying a dwell, the execution of the next block is delayed by the specified time. (Dwell per second)

By setting parameter in the feed per revolution mode, the execution of the next block is delayed until the rotation count of the spindle reaches the specified number. (Dwell per revolution)

Format

M

G04 X_; or **G04 P_ ;**

X_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

T

G04 X_ ; or **G04 U_ ;** or **G04 P_ ;**

X_ : Specify a time or spindle speed (decimal point permitted)
U_ : Specify a time or spindle speed (decimal point permitted)
P_ : Specify a time or spindle speed (decimal point not permitted)

3.12 POLAR COORDINATE INTERPOLATION (T SERIES)

T

Polar coordinate interpolation is a function that exercises contour control in converting a 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). This function is useful for grinding a cam shaft.

Format

G12.1; Starts polar coordinate interpolation mode (enables polar coordinate interpolation)
: Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (hypothetical axis).
G13.1 ; Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation)

Specify G12.1 and G13.1 in Separate Blocks.

Explanation

- Polar coordinate interpolation mode (G12.1)

The axes of polar coordinate interpolation (linear axis and rotary axis) should be specified in advance, with corresponding parameters. Specifying G12.1 places the system in the polar coordinate interpolation mode, and selects a plane (called the polar coordinate interpolation plane) formed by one linear axis and a hypothetical axis intersecting the linear axis at right angles. The linear axis is called the first axis of the plane, and the hypothetical axis is called the second axis of the plane. Polar coordinate interpolation is performed in this plane.

In the polar coordinate interpolation mode, both linear interpolation and circular interpolation can be specified by absolute or incremental programming.

Tool nose radius compensation can also be performed. The polar coordinate interpolation is performed for a path obtained after tool nose radius compensation.

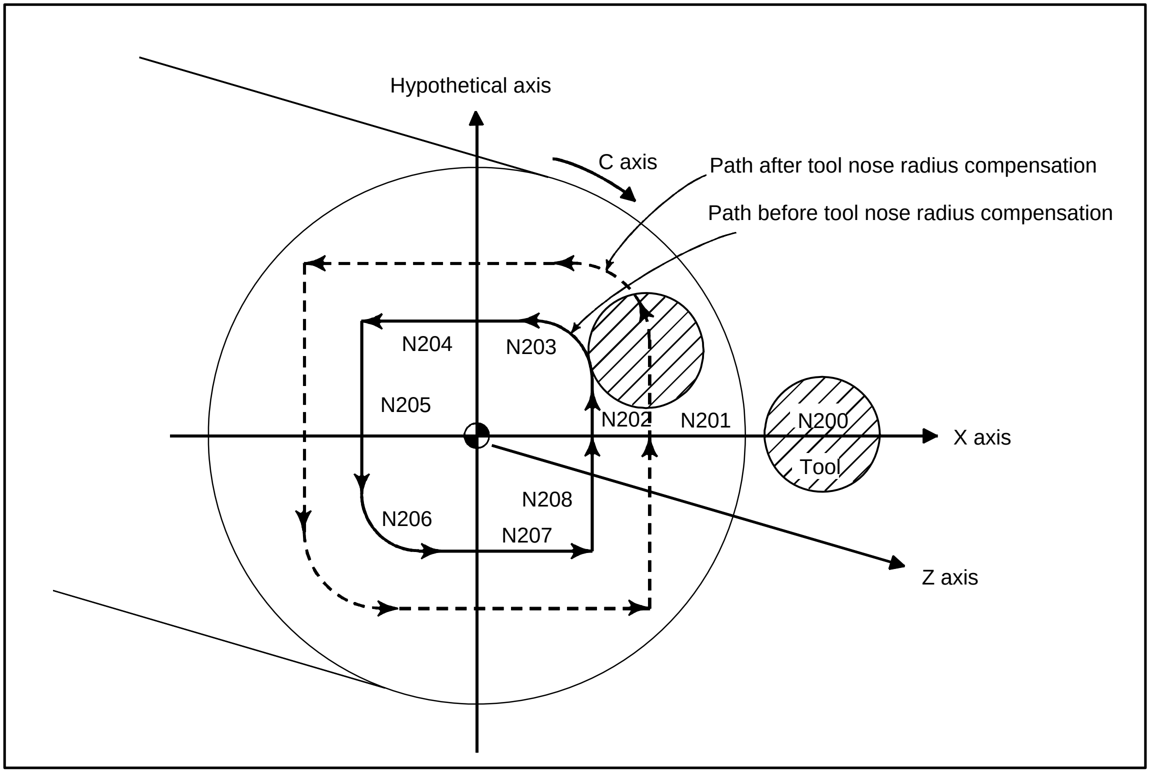
The tangential velocity in the polar coordinate interpolation plane (Cartesian coordinate system) is specified as the feedrate, using F.

- Polar coordinate interpolation cancel mode (G13.1)

Specifying G13.1 cancels the polar coordinate interpolation mode.

Example

- Polar coordinate interpolation in a Cartesian coordinate system consisting of the X axis (a linear axis) and a hypothetical axis



The X-axis is by diameter programming; the C-axis is by radius programming.

```
O0001 ;
   :
N010 T0101 ;
   :
N0100 G90 G00 X120.0 C0 Z__ ;        Positioning to start point
N0200 G12.1 ;                        Start of polar coordinate interpolation
N0201 G42 G01 X40.0 F__ ;          ┐
N0202 C10.0 ;                      │
N0203 G03 X20.0 C20.0 R10.0 ;      │
N0204 G01 X-40.0 ;                 │ Geometry program
N0205 C-10.0 ;                     ├ (program based on Cartesian coordinates on the plane
N0206 G03 X-20.0 C-20.0 I10.0 J0 ; │ of the X-axis and virtual axis)
N0207 G01 X40.0 ;                  │
N0208 C0 ;                         │
N0209 G40 X120.0 ;                 ┘
N0210 G13.1 ;                        Cancellation of polar coordinate interpolation
N0300 Z__ ;
N0400 X__C__ ;
   :
N0900 M30 ;
```

3.13 CYLINDRICAL INTERPOLATION

In cylindrical interpolation, the amount of movement of a rotary axis specified by angle is converted to the amount of movement on the circumference to allow linear interpolation and circular interpolation with another axis. Since programming is enabled with the cylinder side face expanded, programs such as a program for grooving cylindrical cams can be created very easily.

Format

G07.1 IP r ; Starts the cylindrical interpolation mode
(enables cylindrical interpolation).
:
G07.1 IP 0 ; The cylindrical interpolation mode is cancelled.

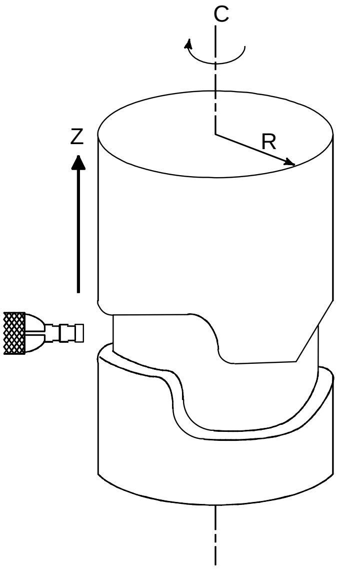
IP : An address for the rotary axis
r : The radius of the workpiece
Specify G07.1 IPr ; and G07.1 IP0; in separate blocks.

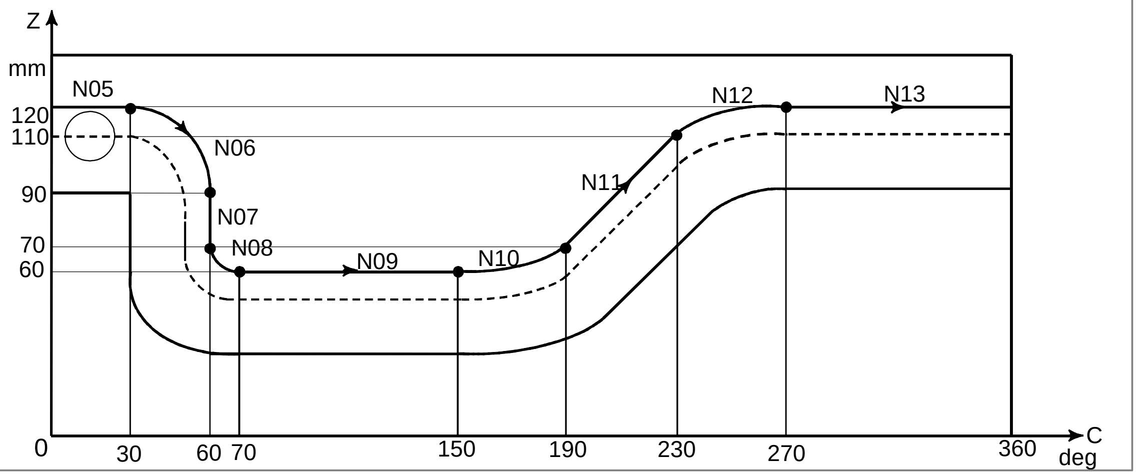
Example

Example of a cylindrical interpolation programs

```
O0001 (CYLINDRICAL INTERPOLATION);
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G18 Z0 C0 ;
N03 G07.1 C57299 ;
N04 G90 G01 G42 Z120.0 D01 F250 ;
N05 C30.0 ;
N06 G03 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G02 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G02 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G03 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```

Note) Sample program where the C axis is parallel to the X-axis.





3.14 HELICAL INTERPOLATION

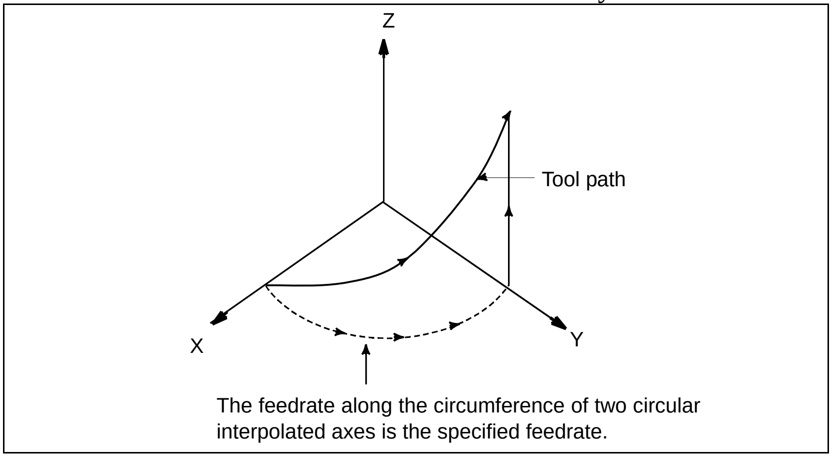
Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.
A tangential velocity of an arc in a specified plane or a tangential velocity about the linear axis can be specified as the feedrate, depending on the setting of parameter.

- When a feedrate along an arc is specified

Therefore, the feedrate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feedrate so the linear axis feedrate does not exceed any of the various limit values.



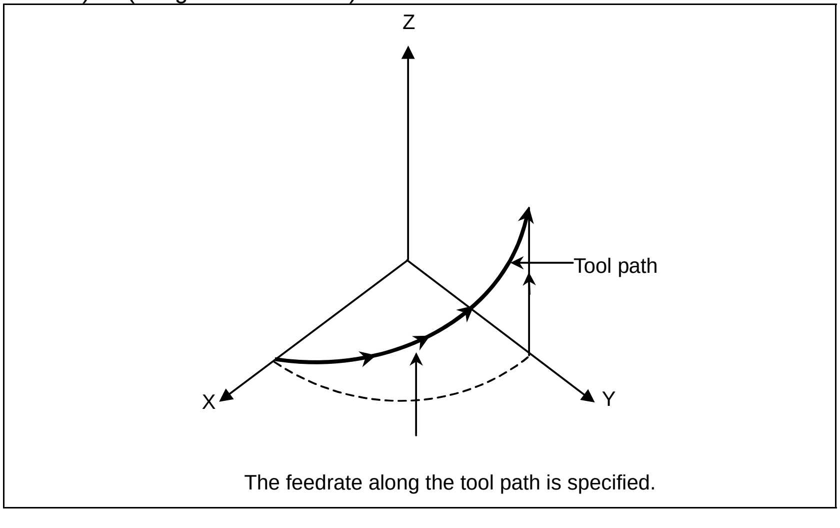
- When a feedrate along the tool path including a linear axis is specified

Therefore, the tangential velocity of the arc is expressed as follows:

$$F \times \frac{\text{Length of arc}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$

The velocity along the linear axis is expressed as follows:

$$F \times \frac{\text{Length of linear axis}}{\sqrt{(\text{Length of arc})^2 + (\text{Length of linear axis})^2}}$$



Format

Arc of XpYp plane

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

Arc of ZpXp plane

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

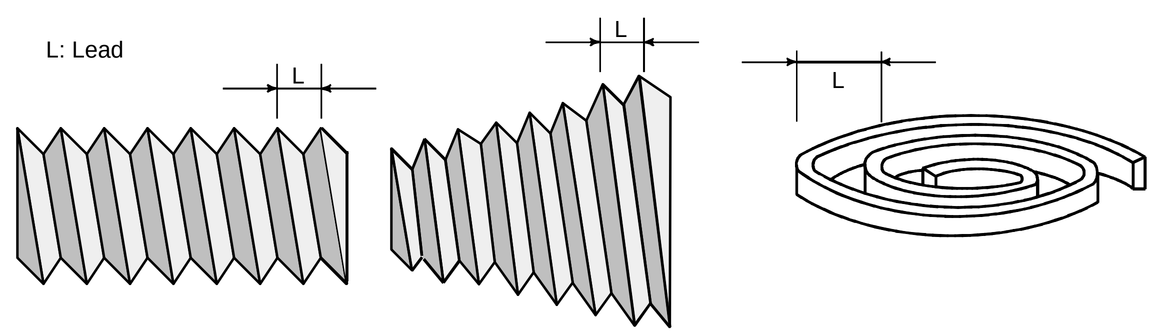
Arc of YpZp plane

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

α, β : Any one axis where circular interpolation is not applied. Up to two other axes can be specified.

3.15 THREAD CUTTING, SYNCHRONOUS CUTTING

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



Straight thread Taper thread Scroll thread

Format

G33 IP_ F_ ;

F_ : Lead along the long axis
(G32 when G code system A with T series)

Explanation

To form a single thread, threading is generally performed several times from rough machining to finish machining along the same path.

Threading starts in synchronism with a one-rotation signal from the position coder attached to the spindle. So, the tool path and cutting start point on the periphery of a workpiece remain unchanged, regardless of the number of threading operations performed. 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:

	Least command increment	Specifiable lead range
Metric input	0.01 mm	0.001 to 5000.0000 mm/rev
	0.001 mm	0.00001 to 500.00000 mm/rev
	0.0001 mm	0.000001 to 50.000000 mm/rev

	Least command increment		Specifiable lead range	
Inch input	0.001	inch	0.00001 to 500.00000	inch/rev
	0.0001	inch	0.000001 to 50.000000	inch/rev
	0.00001	inch	0.0000001 to 5.0000000	inch/rev

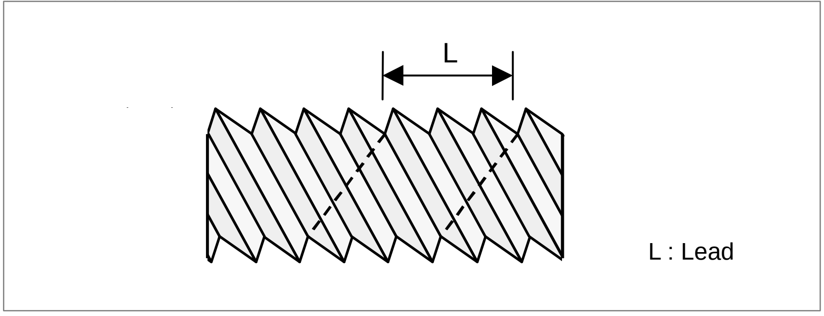
NOTE

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

3.16 MULTIPLE THREADING (T SERIES)

T

Using the Q address to specify an angle between the one-spindle-rotation signal and the start of threading shifts the threading start angle, making it possible to produce multiple-thread screws with ease.



Format

(Constant lead threading)

G32 IP _ F_ Q_ ;

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

G32 IP _ Q_ ;

Explanation

- Available threading commands

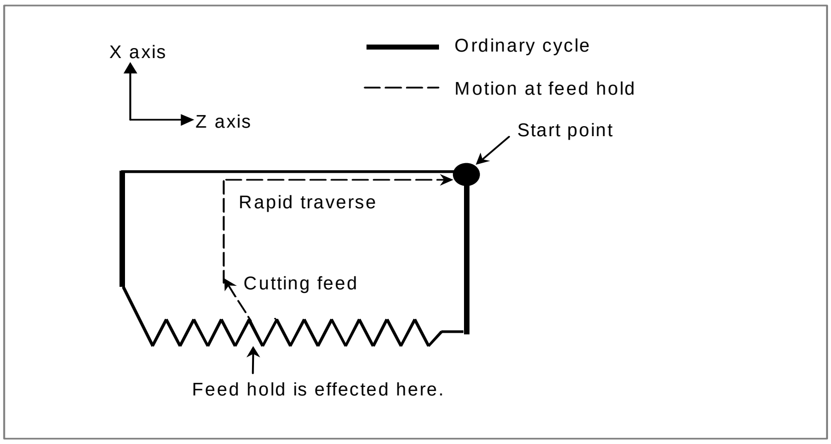
G32 : Constant lead threading
G34 : Variable lead threading
G76 : Combined threading cycle
G92 : Threading cycle

3.17 THREADING RETRACT (T SERIES)

3.17.1 Threading Retract (Canned Cycle) (T Series)

T

In the thread cutting (G92), feed hold may be applied during threading. In this case, the tool immediately retracts with chamfering and returns to the start point on the second axis (X-axis), then the first axis (Z-axis) on the plane.

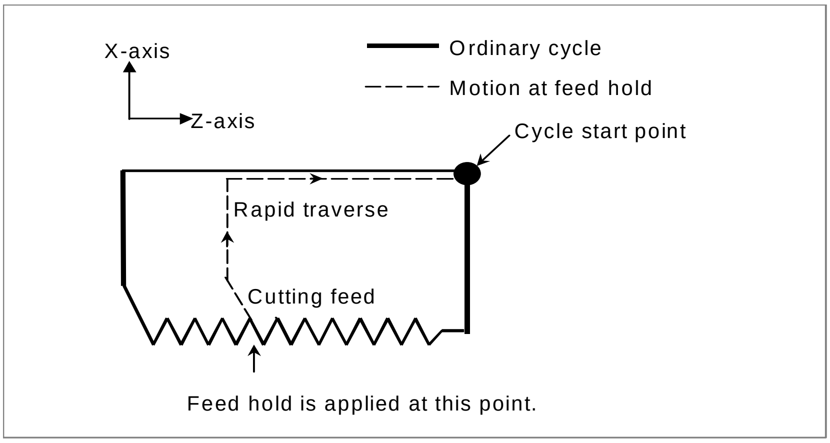


The amount of chamfering during retraction is the same as that of chamfering at the end point.

3.17.2 Threading Retract (Multiple Repetitive Cycle) (T Series)

T

If feed hold is applied during threading in the combined threading cycle (G76), chamfering is performed and a return to the start point of the threading cycle is made as in the threading cycle (G92).
If a cycle start operation is performed here, machining restarts with the threading cycle to which feed hold was applied.



The amount of chamfering during retraction is the same as that of chamfering at the end point.

3.18 CONTINUOUS THREADING (T SERIES)

T

Threading blocks can be programmed successively to eliminate a discontinuity due to a discontinuous movement in machining by adjacent blocks.

Fig. 3.18 (a) Image of continuous threading

3.19 VARIABLE LEAD THREADING (T SERIES)

T

Specifying an increment or a decrement value for a lead per screw revolution enables variable lead threading to be performed.

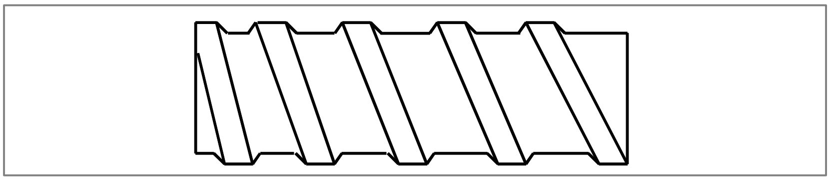


Fig. 3.19 (a) Variable lead screw

Format

G34 IP_ F_ K_ ;

IP_ : End point
F_ : Lead in longitudinal axis direction at the start point
K_ : Increment and decrement of lead per spindle revolution

Explanation

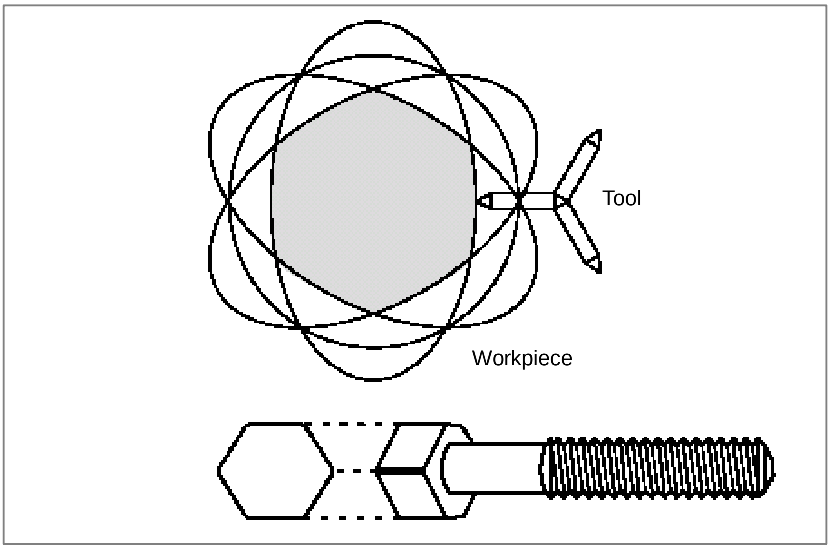
Address other than K are the same as in straight/taper threading with G32.
The K value depends on the increment system of the reference axis, as indicated in following table.

Increment system of the reference axis	Metric input (mm/rev)	Inch input(inch/rev)
IS-A	±0.001 to ± 500.000	±0.00001 to ±50.00000
IS-B	±0.0001 to ± 500.0000	±0.000001 to ±50.000000
IS-C	±0.00001 to ± 50.00000	±0.0000001 to ± 5.0000000

3.20 POLYGON TURNING (T SERIES)

T

Polygon turning means machining a workpiece to a polygonal figure by rotating the workpiece and tool at a certain ratio.



By changing conditions which are rotation ratio of workpiece and tool and number of cutters, the workpiece can be machined to a square or hexagon. The machining time can be reduced as compared with polygonal figure machining using the polar coordinate interpolation. The machined figure, however, is not exactly polygonal. Generally, polygon turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

Format

G51.2 P_ Q_ ; Polygon turning

P,Q : Rotation ratio of spindle and Y-axis
Specify range:
P : Integer from 1 to 999
Q : Integer from -999 to -1 or from 1 to 999
When Q is a positive value, Y-axis makes positive rotation.
When Q is a negative value, Y-axis makes negative rotation.

G50.2 ; Polygon turning cancel

NOTE

Specify G50.2 and G51.2 in a single block.

Explanation

A CNC controlled axis (servo axis) is assigned to the tool rotation axis.
This rotary axis of tool is called Y-axis in the following description. The Y-axis is controlled by the G51.2 command, so that the ratio of the rotation speeds of the spindle (previously specified by S-command) and the tool becomes the specified ratio.
When simultaneous start is specified by G51.2, the one-rotation signal sent from the position codes set on the spindle is detected. After one-rotation signal detection, the Y-axis is controlled using the rotation ratio of the spindle and Y-axis specified by P and Q. So, a position coder needs to be attached to the spindle. This control will be maintained until the polygon turning cancel command is executed (G50.2).

3.21 POLYGON TURNING WITH TWO SPINDLES (T SERIES)

T

When two or more serial spindles are used, the workpiece rotation axis (master axis) and tool rotation axis (polygon synchronization axis) are synchronized at a certain speed ratio.
With this function, it is also possible to specify the phase difference between the master and polygon synchronization axes.
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 a 2-path system, polygonal turning is possible on each path.
By default, the first and second spindles in each system are selected as the master axis and polygon synchronization axis for each system. By setting parameters, however, any spindles belonging to the same system or different systems can be selected as the master axis and polygon synchronization axis.

Format

This is the same as the program command format for polygon turning except for the following points.

1) The command position (R) can be used.
2) Repeated specification in polygon synchronization mode is allowed.

G51.2 Pp Qq Rr ; Polygon synchronization mode start

The command above starts the polygon synchronization mode or modifies specified values (P, Q, R) in the polygon synchronization mode.

P : Master axis rotation ratio
Specifiable range: Integer from 1 to 999
(The rotation direction of the master axis is determined by an ordinary command [such as M03/M04] used for spindle control.)

Q : Polygon synchronization axis rotation ratio
Specifiable range: Integer from 1 to 999 or from -1 to -999
(The rotation direction of a polygon synchronization axis is determined by the sign of a Q command value. Depending on the parameter, however, the rotation direction of the first axis is followed. In this case, no negative value may be specified for Q.)

R : Specification of phase between the master axis and polygon synchronization axis (Specify a relative value.)
Valid specifiable range: 0 ≤ phase angle < 360 deg
The increment system is the same as for the reference axis.
However, the least command increment is 360/4096 (deg), and a specified value smaller than the least command increment is rounded for use.
(R is omissible. If R is not specified even once after the start of the polygon synchronization mode, phase 0 (deg) is used.
If phase control is disabled by parameter setting, the specification of R is ignored.
No particular alarm is issued.)

G50.2 ; Polygon turning cancel

3.22 SKIP FUNCTION

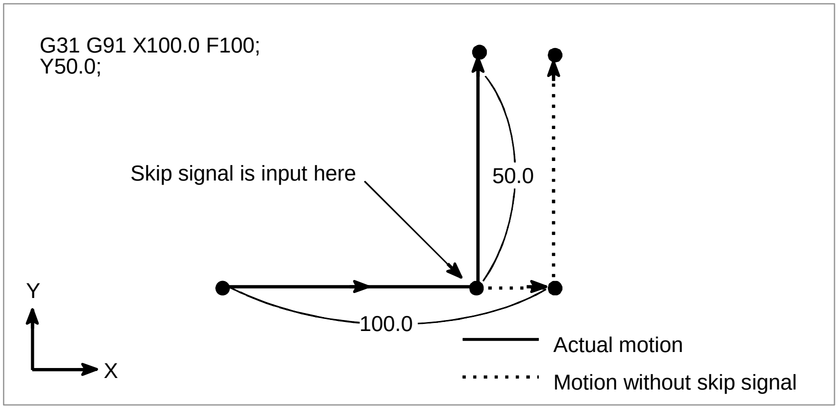
3.22.1 Skip Function

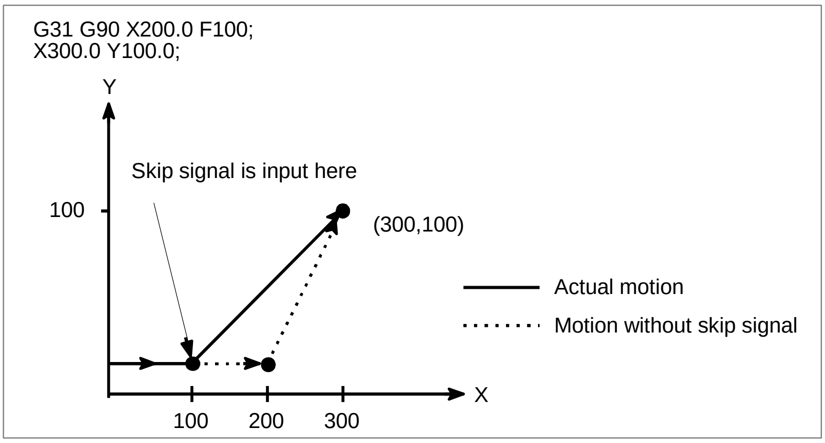
Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP_ ;

G31 : One-shot G code (If is effective only in the block in which it is specified)





3.22.2 Multi-step Skip

In a block specifying P1 to P4 after G31, the multi-step skip function stores coordinates in a custom macro variable when a skip signal (4-point or 8-point ; 4-point when a high-speed skip signal is used) is turned on. In the block where Q1 to Q4 are specified after G04, dwell can be skipped when skip signals (4-point or 8-point ; 4-point when a high-speed skip signal is used) are input.

A skip signal from equipment such as a fixed-dimension size measuring instrument can be used to skip programs being executed.

In plunge grinding, for example, a series of operations from rough machining to spark-out can be performed automatically by applying a skip signal each time rough machining, semi-fine machining, fine-machining, or spark-out operation is completed.

Format

Move command

G31 P_ IP_ F_ ;

P_ : P1 to P4
IP_ : End point
F_ : Feedrate

Dwell

G04X(U,P)_(Q_) ;

X(U,P)_ : Dwell time
Q_ : Q1 to Q4

Explanation

Parameters can be used to specify whether the 4-point or 8-point skip signal is used (4-point when a high-speed skip signal is used). Specification is not limited to one-to-one correspondence. It is possible to specify that one skip signal correspond to two or more Pn or Qn(n=1,2,3,4) commands. The multi-skip function can also be set to skip dwell.

3.22.3 High-speed Skip

The skip function operates based on a high-speed skip signal (connected directly to the CNC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input.

Delay and error of skip signal input is 0 to 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.

Format

G31 IP ;

G31; One-shot G code (If is effective only in the block in which it is specified)

3.22.4 Torque Limit Skip

When the movement command following G31 P99 (or G31 P98) is executed with the servo motor torque limit overridden, cutting feed similar to linear interpolation (G01) can be performed. When the servo motor torque reaches the torque limit (overridden servo motor torque limit) by pushing or the skip signal (including the high-speed skip signal) is input during the movement, the remaining movement commands are canceled and then the next block is executed. (The operation that executes the next block by canceling the remaining movement command is called skip operation later.)
The servo motor torque limit can be overridden by:

- Executing the torque limit override command for the PMC window.

Format

G31 P98 α_ F_

G31 P99 α_ F_

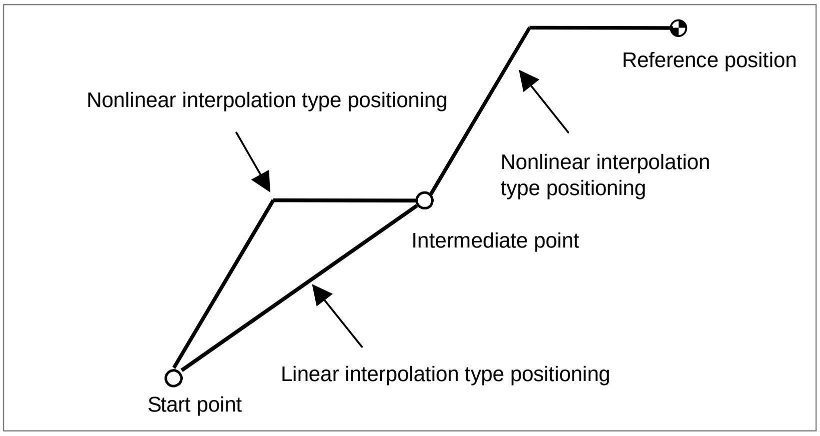
G31 : Skip command (one-shot G code)
P98 : Performs a skip operation if the torque of the servo motor reaches the limit value.
P99 : Performs a skip operation if the torque of the servo motor reaches the limit value or if a skip signal is input.
α : Axis address on any one axis
F : Feedrate

3.23 REFERENCE POSITION RETURN

3.23.1 Automatic Reference Position Return

- Return to reference position (G28)

With the G28 command, the commanded axis is positioned to the reference position via the commanded intermediate point. After positioning, the reference position return end lamp lights.
The reference position must be set in parameter (with the coordinates specified in the machine coordinate system,) before issuing the G28 command.
The tool moves to the intermediate point or reference position at the rapid traverse rate.
For the tool path up to the intermediate point, a selection can be made between nonlinear interpolation type positioning or linear interpolation type positioning.



By parameter setting, linear interpolation positioning can be specified also for the tool path from the intermediate point to the reference position.

Format

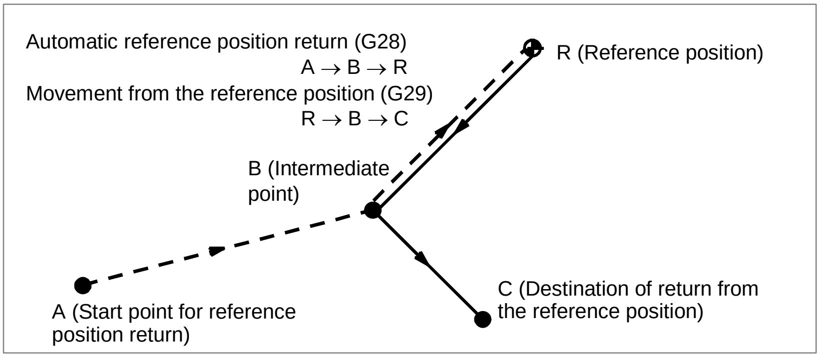
G28 IP_ ;

IP_ : Specify the intermediate position in the absolute coordinate system. (absolute/incremental programming)
There is no need to calculate an actual travel distance between the intermediate position and the reference position.

M

- Movement from reference position (G29)

Based on the G29 command, the tool is positioned along the specified axis at the point specified by G29 through an intermediate point specified by G28.



Format

G29 IP_ ;

IP : Specify the destination of return from the reference position in the absolute coordinate system. (absolute/incremental programming)
The intermediate point is determined by G28 or G30 specified immediately before this command.

3.23.2 Reference Position Return Check

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis for indicating the completion of reference position return goes on.

If the tool has not reached the reference position, an alarm is issued.

When no movement was made along the axis, whether the current position is the reference position is checked.

Format

G27 IP_ ;

IP : Specify positioning to the reference position in the absolute coordinate system so as to return to the reference position. (absolute/incremental programming)

3.23.3 Second, Third, and Fourth Reference Position Return

The G30 command positions the tool to the 2nd, 3rd, or 4th reference position, via the specified intermediate point. Upon completion of positioning, the 2nd, 3rd, or 4th reference position return completion lamp is turned on.

Before issuing the G30 command, The 2nd, 3rd, or 4th reference position must be set in parameters with coordinates in the machine coordinate system.

For the tool path to the intermediate point and the tool path from the intermediate point to the reference position, a selection can be made between nonlinear interpolation position gain and linear interpolation positioning as in the case of G28.

Return to the 2nd, 3rd, or 4th reference position can be performed only after the reference position has been established.

Format

G30 P2 IP_ ; 2nd reference position return (P2 can be omitted.)

G30 P3 IP_ ; 3rd reference position return

G30 P4 IP_ ; 4th reference position return

IP_ : Specify the intermediate point in the absolute coordinate system.
(absolute/incremental programming)
There is no need to calculate an actual travel distance between the intermediate point and the reference position.

3.24 NORMAL DIRECTION CONTROL (M SERIES)

M

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path.

Format

G41.1 ; Normal direction control left side on

G42.1 ; Normal direction control right side on

G40.1 ; Normal direction control cancellation mode

When the workpiece is on the right-hand side of the tool as viewed in the tool advancing direction, normal direction control left side (G41.1) is specified.
Once G41.1 or G42.1 is specified, normal direction control is enabled (the normal direction control mode is set).
When G40.1 is specified, the normal direction control mode is canceled.

Explanation

In the normal direction control, control is made so that the tool may be perpendicular to the advancing direction on the X-Y plane.

On 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.

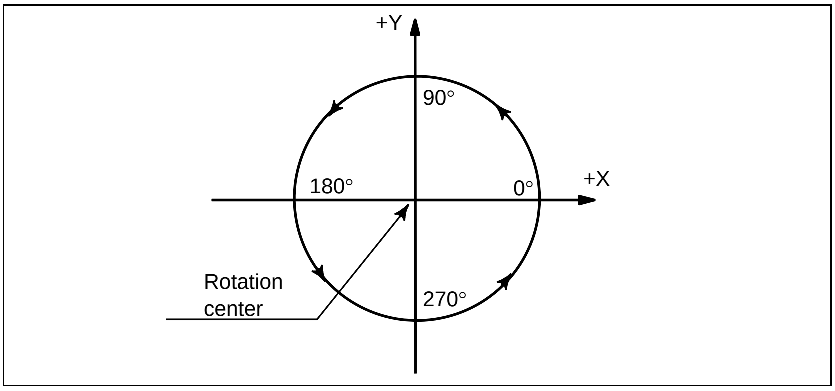


Fig. 3.24 (a) Angle of the C axis

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.

In the cutter compensation mode, the C axis is controlled to face the normal direction relative to the tool path direction after compensation.

The feedrate 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 feedrate is set to the dry run rate. Also, in the case of rapid traverse (G00), it becomes the rapid traverse rate. In the case of circular interpolation, 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 interpolation.

NOTE

The rotation of C axis during normal direction control is controlled at short distance so that 180 degrees or less may result.

3.25 BALANCE CUTTING (T SERIES)

T

When a thin workpiece is to be machined as shown below, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously; this function can prevent the workpiece from warpage that can result when only one side is machined at a time (see the figure below). When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.

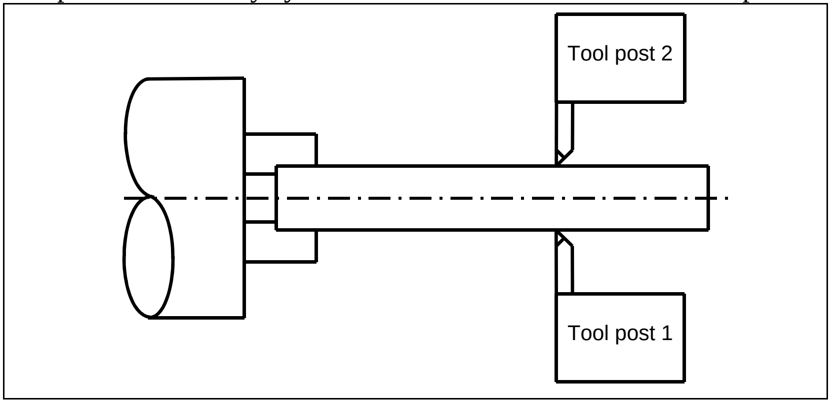


Fig. 3.25 (a) Example of balance cutting

Format

G68 (P_) ;	Balance cutting mode on
G69 ;	Balance cutting mode cancel

Specifying G68, which turns balance cutting mode on, causes balance cutting to be performed with the tool post of path 1 and that of path 2.

NOTE
When the "mirror image for double turrets" function is selected, the balance cutting function cannot be used.

3.26 INDEX TABLE INDEXING (M SERIES)

M

By specifying indexing positions (angles) for the indexing axis (one rotary axis, A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped .

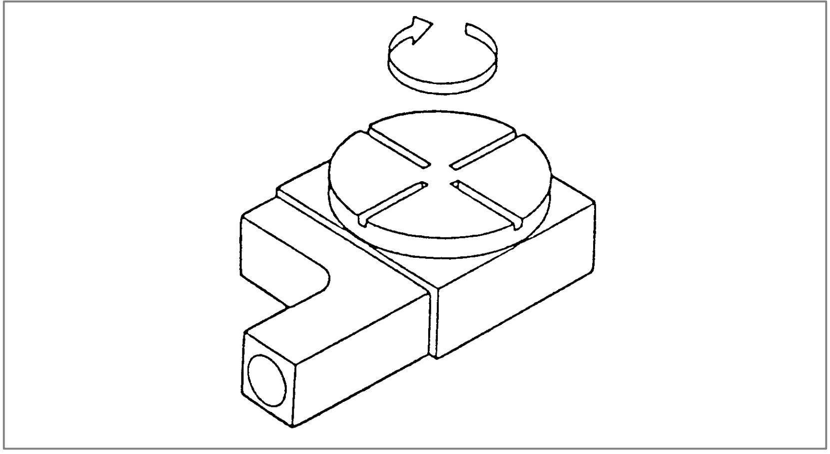


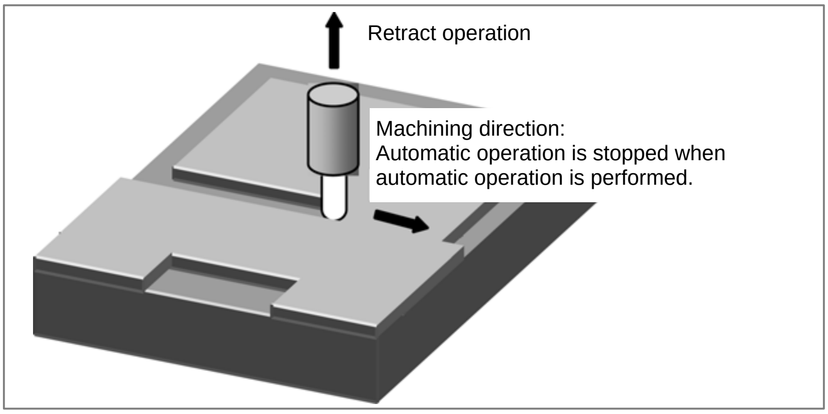
Fig. 3.26 (a) Example of index table

NOTE
This function cannot be used on an axis on which the pole position detection function is used.

3.27 GENERAL PURPOSE RETRACT

In automatic operation mode or in manual operation mode, setting the retract signal to 1 causes this function to capture the rise of this signal, causing the tool to move (retract) along the axis for which a retract amount is specified for parameter. After the end of retraction, the retract completion signal is output. This function is intended to retract the tool from the workpiece immediately when a tool breakage is detected.

- The feedrate assumed during retract is the same as that set in parameter. A feedrate override is invalid.
- A feed hold is invalid to movement during retraction.
- If the retract signal is set to 1 during automatic operation, a retract operation is performed and automatic operation is stopped.
- The retract completion signal becomes 0 when the tool has moved along one of the retract axes.



Retract operation

Machining direction:
Automatic operation is stopped when
automatic operation is performed.

4 FEED FUNCTION

Chapter 4, "FEED FUNCTION", consists of the following sections:

4.1 RAPID TRAVERSE

The positioning command (G00) positions the tool by rapid traverse.

Format

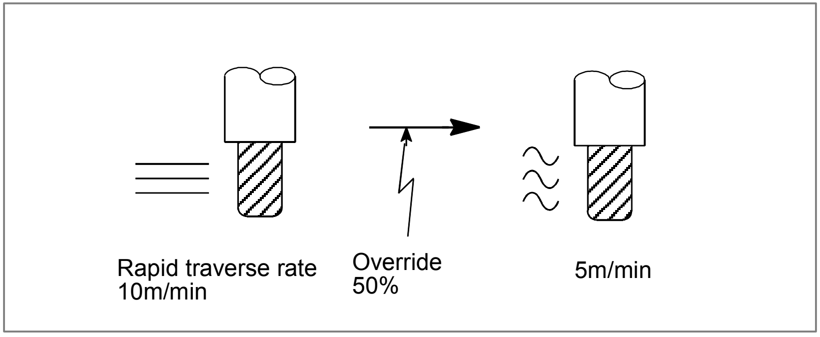
G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ : Dimension word for the end point

In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the parameter (in-position check).
A rapid traverse rate is set for each axis by parameter, so no rapid traverse feedrate need to be programmed.
An override can be applied to the rapid traverse rate.

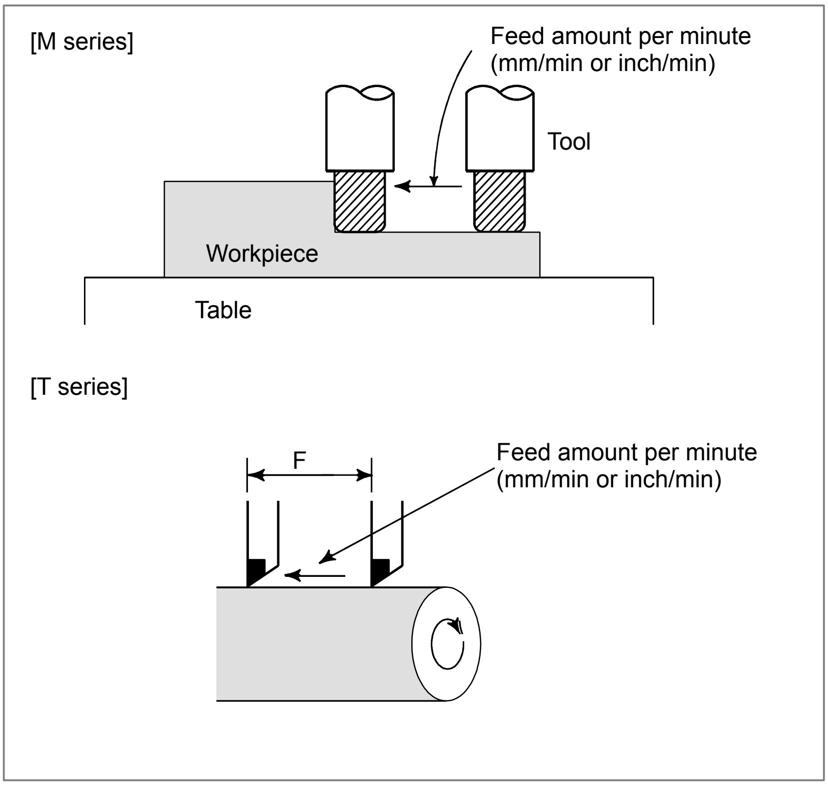
4.2 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter .
Also, 1% rapid traverse override select signal or 0.1% rapid traverse override select signal allows rapid traverse override every 1% or 0.1% in the range of 0 to 100%.



4.3 FEED PER MINUTE

After specifying G94 (G98 for T series) (in the feed per minute mode), the amount of feed of the tool per minute is specified by setting a number after F. G94 (G98 for T series) is a modal code. Once a G94 (G98 for T series) is specified, it is valid until G95 (G99 for T series) (feed per revolution) is specified. At power-on, the feed per minute mode is set. (For the T series, the parameter selects either the feed per revolution mode or the feed per minute mode.)
An override from 0% to 254% (in 1% steps) can be applied to feed per minute, using the feedrate override signal.



Format

M

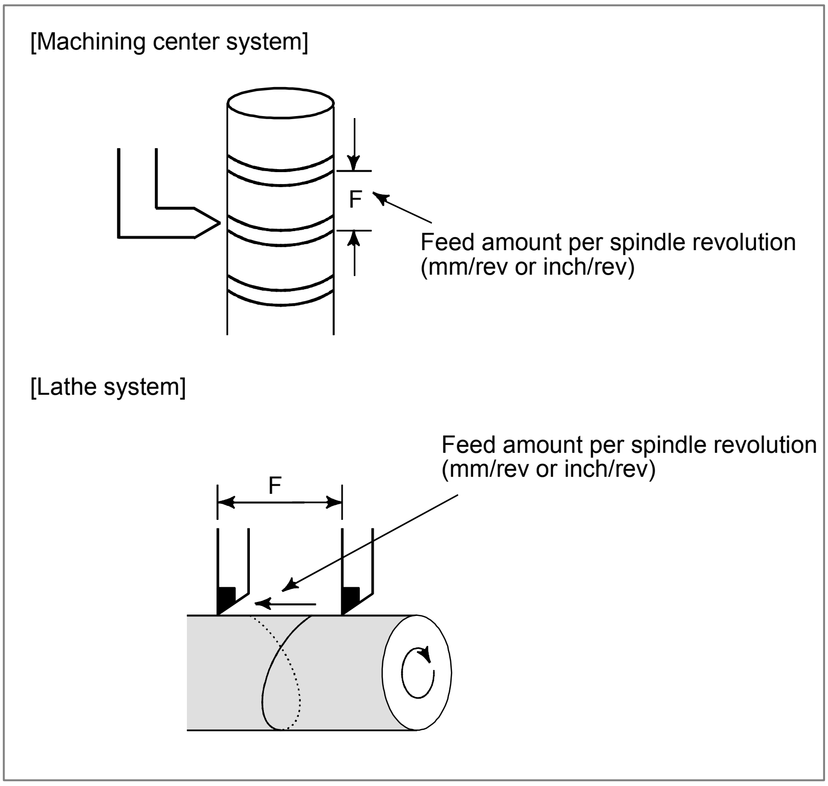
G94 ; G code for feed per minute (Group 05)
F_ ; Feedrate (mm/min or inch/min)

T

G98 ; G code for feed per minute (Group 05)
F_ ; Feedrate (mm/min or inch/min)

4.4 FEED PER REVOLUTION

After specifying G95 (G99 for T series) (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 (G99 for T series) is a modal code. Once a G95 is specified, it is valid until G94 (G98 for lathe system) (feed per minute) is specified.
An override to 0% from 254% (in steps of 1%) can be applied to feed per rotation, using the feedrate override signals.



Format

M

G95 ; G code for feed per revolution (Group 05)
F_ ; Feedrate (mm/rev or inch/rev)

T

G99 ; G code for feed per revolution (Group 05)
F_ ; Feedrate (mm/rev or inch/rev)

4.5 FEED PER REVOLUTION WITHOUT POSITION CODER

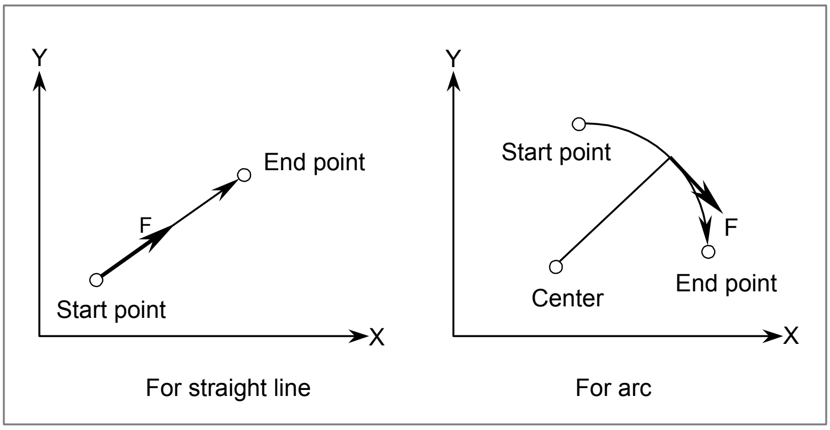
Even when no position coder is attached, a command for feed per revolution can be enabled by parameter setting. (The CNC converts a command for feed per revolution to a command for feed per minute.)

4.6 CONSTANT SURFACE SPEED CONTROL WITHOUT POSITION CODER

In general, feed per revolution cannot be performed on a machine with no position coder attached or used. By parameter setting, feed per revolution can be enabled by spindle commands including a command for constant surface speed control.

4.7 TANGENTIAL SPEED CONSTANT CONTROL

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



4.8 CUTTING FEEDRATE CLAMP

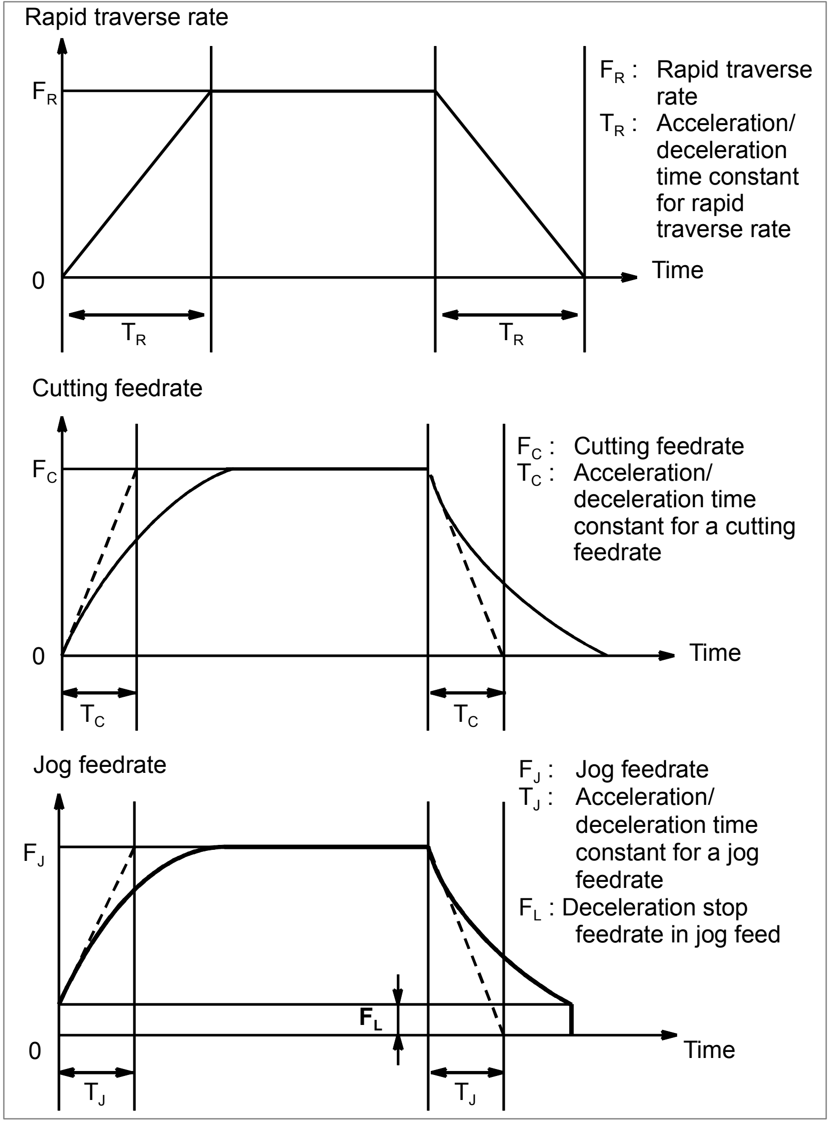
A maximum allowable cutting feedrate can be set on an axis-by-axis basis with a parameter. The cutting feedrate is clamped to such a maximum feedrate that the result of interpolation does not exceed the maximum allowable feedrate on a specified axis.

4.9 AUTOMATIC ACCELERATION/DECELERATION

To prevent a shock from occurring on the mechanical system, acceleration/deceleration is automatically applied to the tool to enable smooth starting and stopping when the tool starts and ends moving. In addition, acceleration/deceleration is applied automatically when the feedrate changes, so that the feedrate can be changed smoothly.
This means that no programming consideration is required for acceleration/deceleration.

There are the following acceleration/deceleration types.
Rapid traverse : Linear acceleration/deceleration (time constant per axis is set by parameter)

		Bell-shaped acceleration/deceleration (See Section II-4.11.)
Cutting feed	:	Exponential acceleration/deceleration (time constant per axis is set by parameter) Linear acceleration/deceleration (time constant per axis is set by parameter) Bell-shaped acceleration/deceleration (See Section II-4.12. This type can be selected only when the bell-type acceleration/deceleration after cutting feed interpolation function option is specified.)
Jog feed	:	Exponential acceleration/deceleration (time constant per axis is set by parameter) Linear acceleration/deceleration (time constant per axis is set by parameter) Bell-shaped acceleration/deceleration (See Section II-4.12. This type can be selected only when the bell-type acceleration/deceleration after cutting feed interpolation function option is specified.)

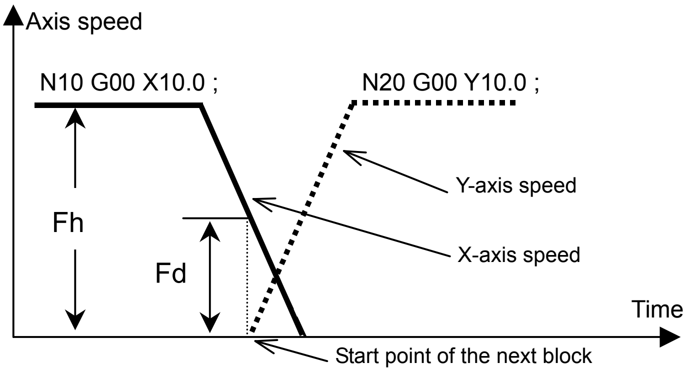


4.10 RAPID TRAVERSE BLOCK OVERLAP

If rapid traverse blocks continue or the block next to a rapid traverse block does not move, the next block can be executed when the feedrate of each axis of a rapid traverse block is decreased to the reduction ratio set by the parameter.

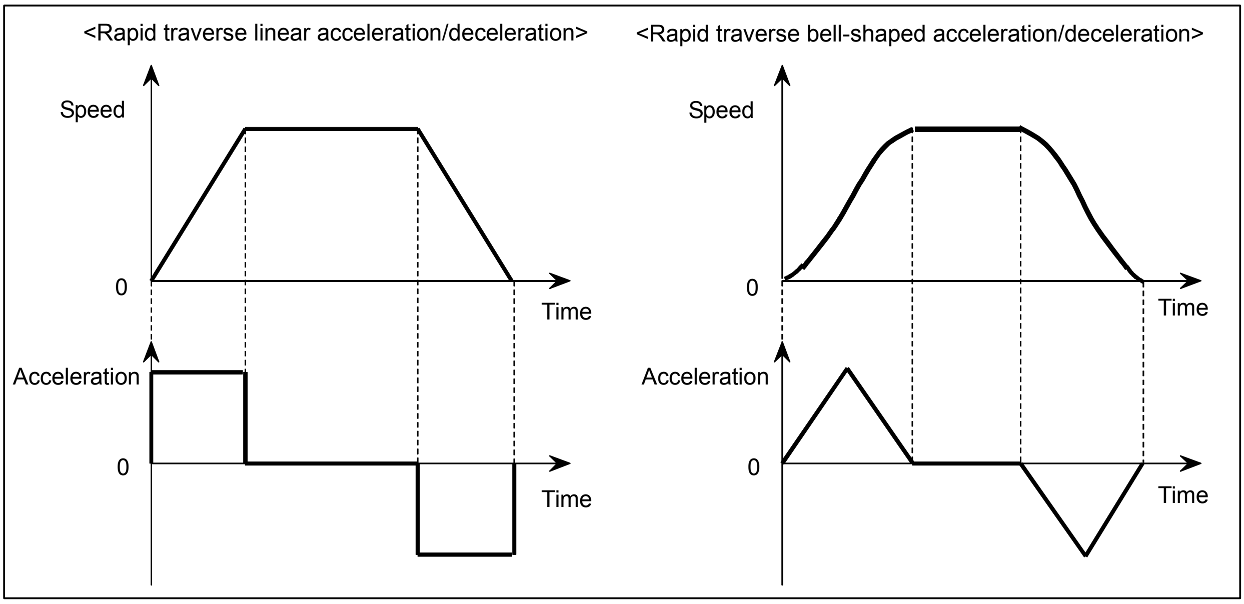
Example)
N10 G00 X10.0 ;
N20 G00 Y10.0 ;

Fh: Rapid traverse rate
α: Reduction ratio (set by parameter No. 1722)
Fd: Deceleration evaluation rate = Fd×α/100



4.11 RAPID TRAVERSE BELL-SHAPED ACCELERATION/DECELERATION

Rapid traverse bell-shaped acceleration/deceleration smoothly increases or decreases the rapid traverse rate, reducing the stress and strain imposed on the machine due to the variation in the acceleration with changes in the feedrate. As the time constant for bell-shaped acceleration/deceleration can be smaller than that for linear acceleration/deceleration, the time needed for acceleration/ deceleration can be reduced.



4.12 BELL-SHAPED ACCELERATION/DECELERATION AFTER CUTTING FEED INTERPOLATION (M SERIES)

M

The bell-shaped acceleration/deceleration after cutting feed interpolation provides smooth acceleration and deceleration to reduce stress and strain on the machine.

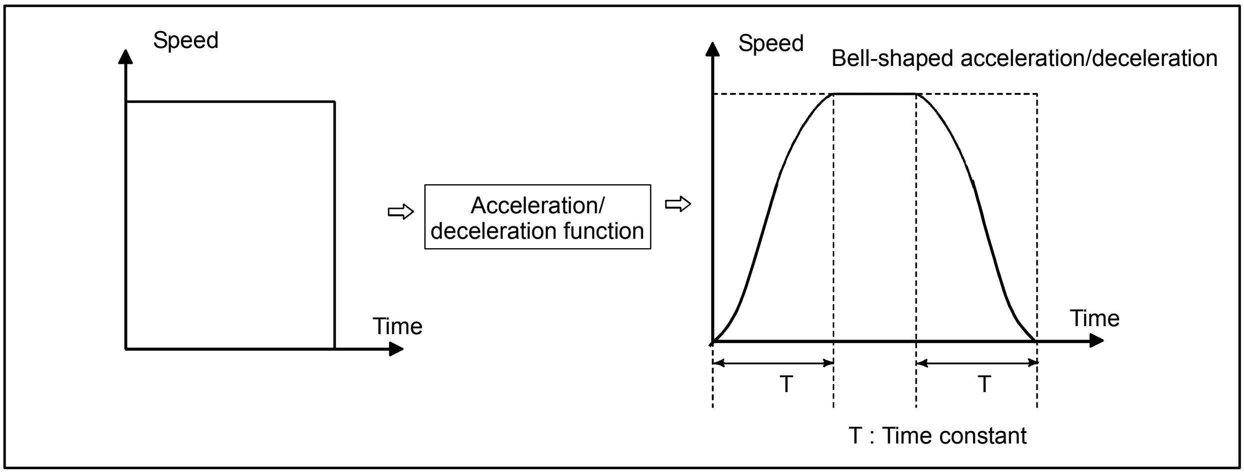
Cutting feed:
Bell-shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter.

Jog feed:

Exponential or bell-shaped acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter.



The time constants for cutting feed and for jog feed on each axis are specified in parameters respectively, in the same way as exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed and the FL feedrate for jog feed are ignored (always assumed to be 0).

4.13 LINEAR ACCELERATION/DECELERATION BEFORE CUTTING FEED INTERPOLATION

Linear acceleration/deceleration can be applied to the tangential feedrate of a specified feedrate. Thus, unlike acceleration/ deceleration after interpolation applied to each axis, this function can eliminate machining profile errors caused by delay in acceleration/ deceleration.

4.14 FEEDRATE OVERRIDE

An override of 0 to 254% (in increments of 1%) can be applied to a cutting feedrate per minute or per revolution.

M

In inverse time (G93), feedrate converted to feed per minute is overridden. Feedrate override cannot be performed to One-digit F code feed.

Feedrate also cannot be performed to functions as threading and tapping in which override is inhibited.

4.15 ONE-DIGIT F CODE FEED (M SERIES)

M

If a one-digit number from 1 to 9 is specified after F, the parameter-set feedrate corresponding to the specified number is set.

When F0 is commanded, rapid traverse rate is set.

If the manual pulse generator is rotated with the one-digit F code feed selection signal set to 1, the feedrate corresponding to the currently selected number is increased or decreased.

A set or modified feedrate is preserved while the power is turned off.

The current feedrate is displayed on the screen.

4.16 INVERSE TIME FEED (M SERIES)

M

Feedrate of the tool can be specified by the move distance of the block and inverse time (FRN). When F0 is specified, an alarm is issued.

- Linear interpolation (G01)
  FRN=1/Time (min) = Speed/Distance
  Speed: mm/ min (metric input)
  inch/ min (inch input)
  Distance: mm (metric input)
  inch (inch input)
- Circular interpolation (G02, G03)
  FRN=1/Time (min) = Speed/Circle radius
  Speed: mm/ min (metric input)
  inch/ min (inch input)
  Circle radius: mm (metric input)
  inch (inch input)

4.17 JOG OVERRIDE

The jog feedrate and incremental feedrate can be overridden by:
0 to 655.34% (in steps of 0.01%)

4.18 OVERRIDE CANCEL

Feedrate override can be clamped to 100% by a signal from the machine side.

4.19 MANUAL PER REVOLUTION FEED (T SERIES)

T

In manual per revolution feed, jog feed is performed at the feedrate obtained by multiplying the spindle speed by the amount of feed per spindle revolution obtained by multiplying the amount of feed per spindle revolution set in the parameter by the jog feedrate override value.
That is, the feedrate during manual per revolution feed is calculated by the following expression.

Parameter setting (mm/rev) × Jog feedrate override value × Spindle actual speed (rev/min)

4.20 EXTERNAL DECELERATION

The control axis is externally decelerated. The feedrate is decelerated by the external deceleration signals from the machine. The deceleration rate is set by the parameters.
The external deceleration signal are provided for each axis and direction.
As three types of deceleration condition settings can be dynamically selected by the signals.
External deceleration can be applied to rapid traverse rate, cutting feedrate, and manual handle feedrate.

- Rapid traverse and cutting feed

Three types of deceleration conditions can be set by the parameters.
When deceleration conditions specified by multiple external signals input during machining are conflicted, the condition with the lowest external deceleration rate is applied.

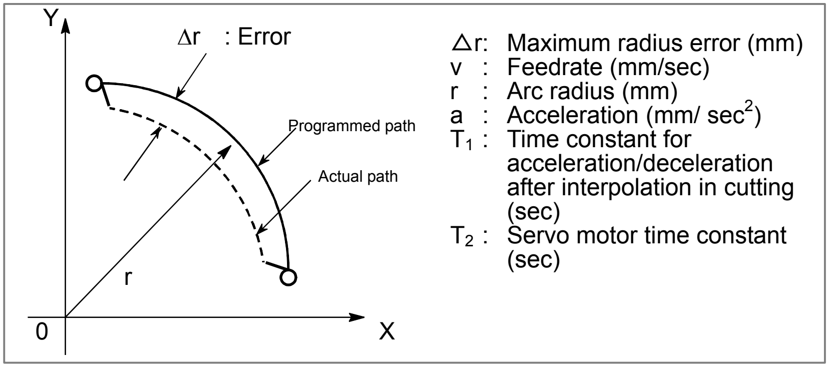
External deceleration settings 2 and 3 can be switched between enabled and disabled by the parameters.

- Manual handle feed

Three types of deceleration conditions can be set by the parameters. For handle feed, the maximum feedrate is switched when the external deceleration signal in the positive or negative direction for the handle axis is set to 0. When multiple conditions are conflicted, the condition with the lowest maximum feedrate is applied.

4.21 SPEED CONTROL WITH ACCELERATION IN CIRCULAR INTERPOLATION

When cutting is performed at high speed based on circular interpolation or helical interpolation, the actual tool path has an error relative to the programmed path. This error can be expressed approximately by the following equation in circular interpolation:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{v^2}{r} = \frac{1}{2}(T_1^2 + T_2^2)\cdot a \quad \cdots\cdots \quad \text{(Equation 1)}$$

Allowable error Δr is given in actual machining, so maximum allowable acceleration a (mm/s^2) is calculated by equation 1.

In acceleration-based speed control in circular interpolation, when a feedrate that causes a radius error to exceed the allowable error along a programmed arc with an arbitrary radius is specified, the feedrate for circular cutting is automatically clamped according to the parameter setting.

Let A be an allowable acceleration rate calculated from an allowable acceleration rate set for each axis. Then, the maximum allowable speed v for the programmed radius r is:

$$v = \sqrt{A \cdot r} \quad \cdots\cdots \quad \text{(Equation 2)}$$

If a specified feedrate exceeds the speed v found from Equation 2, the feedrate is automatically clamped to v.

Parameter is used to specify an allowable acceleration rate. If allowable acceleration rates for two circular interpolation axes differ from each other, the smaller allowable acceleration rate is selected.

If the radius of an arc is small, the calculated reduced speed v may become very low. To prevent the feedrate from becoming too low in such a case, a minimum allowable feedrate can be set in parameter.

Notes

M

- This function can be used in the normal mode.

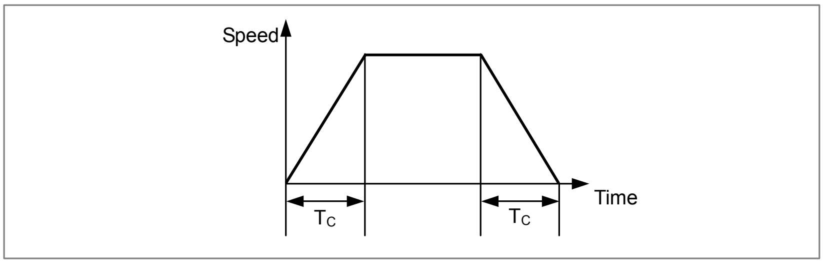
- This function can be used in the advanced preview control (T series)/AI advanced preview control (M series)/AI contour control (M series) mode.

4.22 LINEAR ACCELERATION/DECELERATION AFTER CUTTING FEED INTERPOLATION

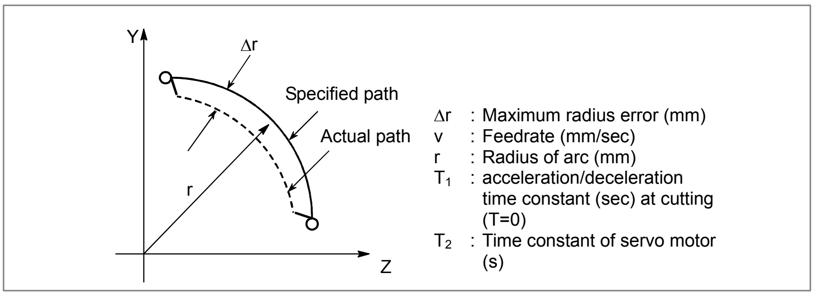
If linear acceleration/deceleration after interpolation for cutting feed is enabled, acceleration/ deceleration is performed as follows:

Cutting feed : Linear acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter.
Jog feed : Exponential or linear acceleration/deceleration (constant acceleration time)
Specify the acceleration/deceleration time constant for each axis in parameter

If an identical time constant is specified, linear acceleration/ deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/deceleration, thus reducing the time needed for acceleration and deceleration.



If circular interpolation is performed, especially when high-speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.



The maximum value of an error in the radius direction can be approximately obtained by the following expressions:

For exponential acceleration/deceleration:

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

For linear acceleration/deceleration after interpolation:

$$\Delta r = \left(\frac{1}{24} T_1{}^2 + \frac{1}{2} T_2{}^2 \right) \frac{V^2}{r}$$

Accordingly, if the same time constant for acceleration/deceleration is used, the value of an error that occurs with linear acceleration/deceleration is 1/12 of the value of an error that occurs with exponential acceleration/deceleration when an error due to the time constant of the servo motor is not considered.

4.23 ADVANCED PREVIEW CONTROL (T SERIES) / AI ADVANCED PREVIEW CONTROL (M SERIES) / AI CONTOUR CONTROL (M SERIES)

The advanced preview control (T series), AI advanced preview control (M series), and AI contour control (M series) are provided for high-speed, high-precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.

Format

T

- Advanced preview control

G08 P_ ;

P1 : Advanced preview control mode on
P0 : Advanced preview control mode off

NOTE
1 Always specify G08 in an independent block.
2 The advanced preview control mode is also cleared by the reset operation.

M

- AI advanced preview control/AI contour control

G05.1 Q_ ;

Q1 : AI advanced preview control mode/AI contour control mode on
Q0 : AI advanced preview control mode/AI contour control mode off

NOTE
1 Always specify G05.1 in an independent block.
2 The AI advanced preview control/AI contour control mode is also cleared by the reset operation.

- Valid functions

The following table shows the functions included in these functions.

	Advanced preview control	**AI advanced preview control**		**AI contour control**
Model	Series 0*i*-TD	Series 0*i* Mate-MD	Series 0*i*-MD	Series 0*i*-MD
Standard/option	Option	Standard		Option
Look-ahead block count	1	12	20	40
Look-ahead linear acceleration/deceleration before interpolation	○	○		○
Look-ahead bell-shaped acceleration/deceleration before interpolation	–	–		☆
Function for changing time constant of bell-shaped acceleration/deceleration	–	–		☆

	Advanced preview control	AI advanced preview control	AI contour control
Advanced feed forward	○	○	○
Acceleration setting for each axis	○	○	○
Speed control based on the feedrate difference on each axis	○	○	○
Speed control with acceleration in circular interpolation	○	○	○
Speed control with the acceleration on each axis	－	○	○

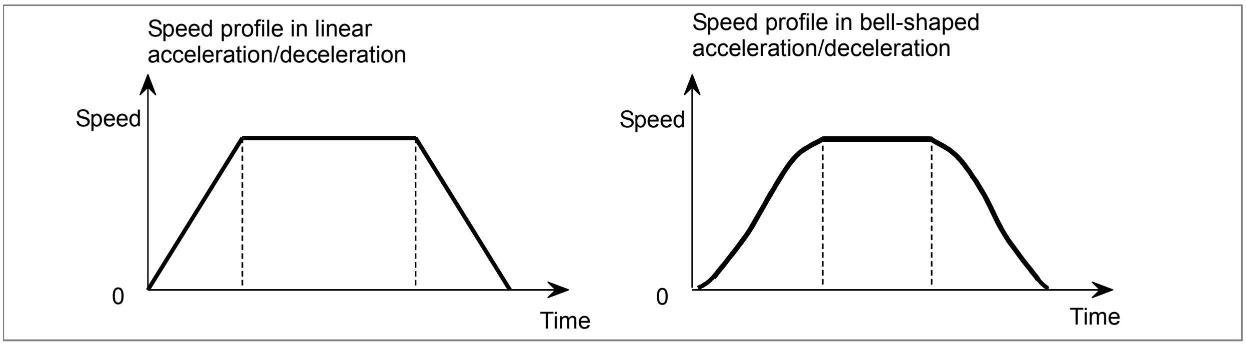
－: Function not supported
○: Standard function
☆: Optional function

The function for changing time constant of bell-shaped acceleration/deceleration is included in look-ahead bell-shaped acceleration/deceleration before interpolation.

4.24 BELL-SHAPED ACCELERATION/DECELERATION BEFORE LOOK-AHEAD INTERPOLATION (M SERIES)

M

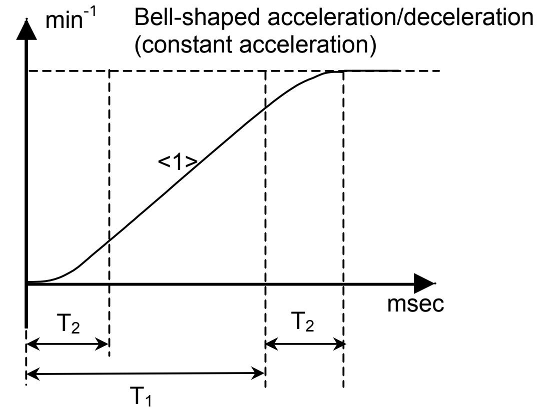
By producing a bell-shaped feedrate profile for acceleration/ deceleration before interpolation in AI contour control, machining profile errors caused by delay in acceleration/ deceleration can be eliminated, and a shock to the machine can be reduced for smoother acceleration/deceleration.



4.25 RIGID TAPPING BELL-SHAPED ACCELERATION/DECELERATION (M SERIES)

M

In rigid tapping, bell-shaped acceleration/deceleration of constant acceleration type can be applied by parameter setting.
In parameters, set a time required until a maximum spindle speed is reached (linear portion) and a time required for the curved portion.
The actual time constant for the linear portion is the proportion of the maximum spindle speed to a specified S value.
The time constant for the curved portion is determined not by an S command but by a set value.
Accordingly, the actual time constant is [proportionally calculated value for linear portion + value for curved portion].



Set time constants T_1 and T_2 in parameters.
(From the maximum speed and setting of T_1, the acceleration of linear portion <1> is constant.)

4.26 SPEED COMMAND EXTENSION IN LEAST INPUT INCREMENTS C

When the least input increment C (IS-C) is selected, the limitations indicated in Table 4.26 (a) and Table 4.26 (b) have been applied to the speed and acceleration parameters.

·Speed and angular velocity parameters

Table 4.26 (a)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.0001	0.0000 to +99999.9999
inch/min	IS-C	0.00001	0.00000 to +4000.00000

·Acceleration and angular acceleration parameters

Table 4.26 (b)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 degree/sec^2	IS-C	0.0001	0.0000 to +99999.9999
inch/sec^2	IS-C	0.00001	0.00000 to +9999.99999

In speed command extension in least input increments C, the parameters are set to extend the data ranges as shown in Tables 4.26 (c) and 4.26 (d).
This extension increases the upper limit of the maximum rapid traverse rate and maximum cutting speed to:

- Approximately 1000000 mm/min (when the data unit is mm)

·Speed and angular velocity parameters after extension

Table 4.26 (e)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-C	0.001	0.000 to +999000.000
inch/min	IS-C	0.0001	0.0000 to +9600.0000

·Acceleration and angular acceleration parameters after extension

Table 4.26 (f)

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 degree/sec^2	IS-C	0.001	0.000 to +999999.999
inch/sec^2	IS-C	0.0001	0.0000 to +99999.9999

NOTE

After extension, the minimum data unit is increased by a factor of 10.

5 PROGRAM INPUT

Chapter 5, "PROGRAM INPUT", consists of the following sections:

5.1 PROGRAM CODE

EIA codes, ISO codes, or ASCII codes can be used for program codes. When ASCII codes are used for codes of a program to be input, parameters need to be set. When EIA codes or ISO codes are used, automatic discrimination is performed with the first end of block code (CR for EIA codes or LF for ISO codes), so no parameters need to be set.
Parameters are used to specify which code is used to output a program.
For the usable program codes, see the list of program codes in Appendix.

5.2 LABEL SKIP

In one of the following cases, the label skip function is enabled, and "LSK" is indicated on the screen:

- When the power is turned on
- When the CNC is reset

When the label skip function is enabled, all codes are ignored until the first end-of-block (EOB) code is read. A portion ignored is referred to as a "leader portion", and the data after the first end-of-block (EOB) code is regarded as "significant information".

5.3 PARITY CHECK

- Program horizontal direction parity check (TH check)
  Each character of an input program is checked for parity. If a parity error is detected, an alarm is issued.
- Program vertical direction parity check (TV check)
  Each block of an input program is checked for parity. If a block (from the code immediately after an EOB to the next EOB) includes an odd number of characters, an alarm is issued.

The TH check function and TV check function do not operate on a portion skipped by the label skip function. In a comment portion, no TH check is made. However, whether to include the number of characters of a comment portion for a TV check can be chosen by parameter setting. The TV check function can be enabled or disabled by setting through the MDI panel.

5.4 CONTROL-IN / CONTROL-OUT

Any information enclosed by the control-out and control-in codes is regarded as a comment.
A portion ignored is referred to as a "comment portion".
In a comment portion, the reset code (ISO code: % or EIA code: ER) cannot be used.

Name	ISO code	EIA code
Control-out	(	2-4-5 channel on
Control-in	)	2-4-7 channel on

5.5 OPTIONAL BLOCK SKIP

If a slash followed by a number (/n) is coded at the start of a block, and optional block skip switch n on the machine operator's panel is turned on, the information of the block where number /n corresponding to switch number n is specified is ignored.
If optional block skip switch n is turned off, the information of the block where /n is specified is not ignored. This means that the block where /n is specified can be skipped according to the choice by the operator.
As n, 1 can be used. The number 1 of /1 can be omitted.
Example)

/1 N12345 G00 X100. Z200. ;

5.6 OPTIONAL BLOCK SKIP EXTENSION

As n of /n for optional block skip, a number from 2 to 9 can be used.

5.7 MAXIMUM COMMAND VALUES

The basic addresses and specifiable value ranges are indicated below. Note, however, that the information below represents restrictions imposed by the CNC, and is totally irrelevant to the restrictions imposed by each machine.

Table 5.7 (a) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (*1)	1 to 9999	1 to 9999
Sequence number		N	1 to 99999	1 to 99999
Preparatory function		G	0 to 9999	0 to 9999
Dimension word	Increment system IS-A	X,Y,Z,U,V, W,A,B,C,I, J,K,R	±999999.99 mm ±999999.99 deg	±99999.999 inch(*2) ±999999.99 deg
	Increment system IS-B		±999999.999 mm ±999999.999 deg	±99999.9999 inch(*2) ±999999.999 deg
	Increment system IS-C		±99999.9999 mm ±99999.9999 deg	±9999.99999 inch(*2) ±99999.9999 deg
Feed per minute	Increment system IS-A	F	0.01 to 999000.00 mm/min	0.001 to 96000.000 inch/min
	Increment system IS-B		0.001 to 999000.000 mm/min	0.0001 to 9600.0000 inch/min
	Increment system IS-C		0.0001 to 99999.9999 mm/min	0.00001 to 4000.00000 inch/min
Feed per revolution		F	0.0001 to 500.0000 mm/rev	0.000001 to 9.999999 inch/rev
Spindle speed function		S (*3)	0 to 99999	0 to 99999
Tool function		T (*3)	0 to 99999999	0 to 99999999
Auxiliary function		M (*3)	0 to 99999999	0 to 99999999
		B (*3)	0 to 99999999	0 to 99999999
Offset number (M series only)		H, D	0 to 400	0 to 400
Dwell	Increment system IS-A	X, U (T series only)	0 to 999999.99 sec	0 to 999999.99 sec
	Increment system IS-B		0 to 99999.999 sec	0 to 99999.999 sec
	Increment system IS-C		0 to 9999.9999 sec	0 to 9999.9999 sec
Dwell		P	1 to 99999999	1 to 99999999
Designation of a program number		P	1 to 9999	1 to 9999
Number of subprogram repetitions		L	1 to 99999999	1 to 99999999
		P	0 to 9999	0 to 9999

*1 In ISO code, the colon (:) can also be used as the address of a program number.

*2 For inch input/millimeter machines, the maximum specifiable range of dimension words is as follows:

Increment system	Maximum specifiable range
IS-A	±39370.078 inch
IS-B	±39370.0787 inch
IS-C	±3937.00787 inch

*3 The maximum value of addresses M, T, and B is 99999999(8 digits). The maximum value of address S is 99999(5 digits). Note that, however, values longer than the permissible number of digits set in parameter No. 3030 to 3033 cannot be specified. The values and uses for some codes are limited by parameter setting. (For example, some M codes are not buffered.) For details, refer to the parameter manual.

5.8 SEQUENCE NUMBER

At the start of a block in a program, an 5-digit sequence number can be specified after address N. The sequence numbers of the program being executed are displayed on the screen at all times. The sequence number search function can be used to search for a sequence number in a program.

5.9 ABSOLUTE PROGRAMMING / INCREMENTAL PROGRAMMING

There are two ways to programming travels of the each axis; the absolute programming, and the incremental programming. In the absolute programming, coordinate value of the end position is programmed. The incremental programming is used to program the amount of each axis movement.
G90 and G91 are used to command absolute or incremental programming, respectively.

Format

G90 IP_ ; Absolute programming
G91 IP_ ; Incremental programming

T

When G code system A is used on the T series, an absolute/incremental programming is not identified by a G90/G91 command but by an address word.

Move command axis	Absolute programming	Incremental programming
X axis	X	U
Z axis	Z	W
Y axis	Y	V
C axis	C	H
A axis	A	None
B axis	B	None

5.10 DECIMAL POINT PROGRAMMING / POCKET CALCULATOR TYPE DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. The decimal point can basically be used for a command value with a unit such as for distance, angle, time, or speed, and the position of the decimal point represents mm, inch, deg, or sec.
There are two types of decimal point programming: decimal point programming and pocket calculator type decimal point programming.

When pocket calculator type decimal point programming is used, a value without decimal point is considered to be specified in mm, inch, deg, or sec. When standard type decimal point programming is used, such a value is considered to be specified in least input increments. Select either pocket calculator type decimal point programming or standard type decimal point programming by using the parameter. Values can be specified both with and without decimal point in a single program.

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

5.11 INPUT UNIT 10 TIME MULTIPLY

By parameter setting, a least input increment 10 times greater than a least command increment can be set as indicated in the table below.

Table 5.11 (a) Least input increments 10 times greater than least command increments

Increment system	Least input increment		Least command increment	
IS-B	0.01	mm	0.001	mm
	0.001	inch	0.0001	inch
	0.01	deg	0.001	deg
IS-C	0.001	mm	0.0001	mm
	0.0001	inch	0.00001	inch
	0.001	deg	0.0001	deg

NOTE

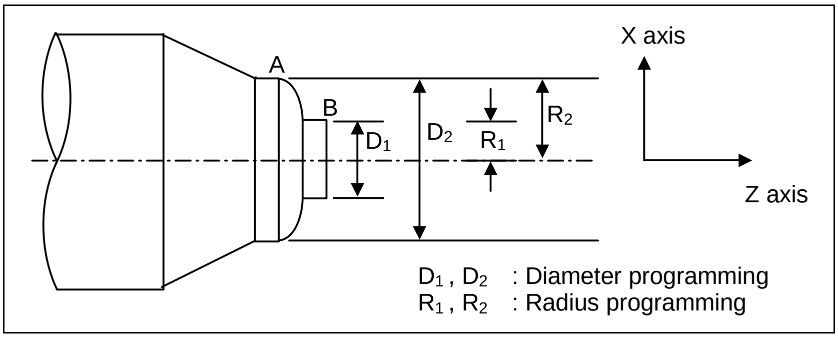
When the increment system is IS-A or pocket calculator type decimal point input is used, a least input increment 10 times greater than a least command increment cannot be set.

5.12 DIAMETER PROGRAMMING / RADIUS PROGRAMMING

When turning is performed, the cross section of a workpiece is usually a circle. The size of a circle may be specified by its diameter or radius.

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.

Whether to use radius programming or diameter programming can be chosen for each axis by parameter setting.



NOTE

For the FS0*i*-C, to enable an axis for which diameter programming is specified to travel a specified distance, it was necessary to make one of the following two changes in addition to setting bit 3 (DIAx) of parameter No. 1006.

- Halving the command multiply (CMR) (The detection unit is not changed.)
- Halving the detection unit and doubling the flexible feed gear (DMR)

For the FS0*i*-D, however, only if bit 3 (DIAx) of parameter No. 1006 is set, the CNC halves the specified pulse. Therefore, changes shown above are not necessary (when the detection unit is not changed).

When the detection unit is halved, both CMR and DMR need to be doubled.

5.13 PLANE SELECTION

A plane to be used for circular interpolation, plane to be used for cutter compensation (M series), plane to be used for coordinate system rotation (M series), and plane perpendicular to hole machining can be selected using G codes.

G code	Selected plane
G17	Xp-Yp plane
G18	Zp-Xp plane
G19	Yp-Zp plane

Xp : X axis or an axis parallel to it
Yp : Y axis or an axis parallel to it
Zp : Z-axis or an axis parallel to it

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
Parameter is used to specify that an optional axis be parallel to the each axis of the X-, Y-, and Z-axes as the basic three axes.
The movement instruction is irrelevant to the plane selection.

Example

- Example 1

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

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

- Example 2

In a block where none of G17, G18, and G19 is specified, the plane remains unchanged.

G18 X_ Z_.................. ZX plane
X_ Y_.................. The plane remains unchanged (ZX plane).

- Example 3

If a block specifies G17, G18, or G19 but has no axis address specified, the omission of the addresses of three basic axes is assumed.

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

5.14 ROTARY AXIS SPECIFICATION

By parameter setting, a controlled axis can be set as a rotary axis that operates according to a command based on an angular displacement.
Two types of rotary axes are available for selection by parameter setting as indicated below.

Rotary axis (A type)	Rotary axis (B type)
(1) Inch/metric conversion is not done. (2) Machine coordinate values are rounded in 0 to 360°. Absolute coordinate values and relative coordinate values are rounded or not rounded by parameters. (3) Stored pitch error compensation is the rotary axis type. (4) Automatic reference position return (G28, G30) is done in the reference position return direction and the move amount does not exceed one rotation.	(1) Inch/metric conversion is not done. (2) Machine coordinate values, absolute coordinate values, and relative coordinate values are linear axis type. (Is not rounded in 0 to 360°). (3) Stored pitch error compensation is linear axis type (4) Cannot be used with the rotary axis roll-over function and the index table indexing function (M series).

5.15 ROTARY AXIS ROLL-OVER

The absolute coordinate values and relative coordinate values on a rotary axis are rounded to coordinates within one rotation at all times. This function can prevent a coordinate overflow from occurring.
For an incremental command, a command value itself serves as a travel distance.
For an absolute command, a command value is rounded to within one rotation, and a rounded coordinate represents an end point. In this case, whether the direction of movement follows the sign of a command value or the shortcut direction with a shorter travel distance is used can be selected by parameter setting.

5.16 POLAR COORDINATE COMMAND (M SERIES)

M

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

Format

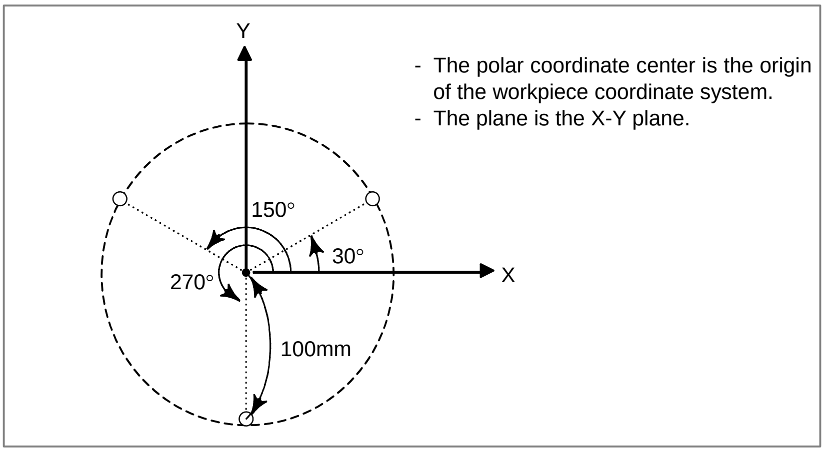
G16; Polar coordinate system command starts
G15; Polar coordinate system command cancel

Explanation

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 programming (G90, G91).
If a radius value is specified in the absolute mode, the polar coordinate center is the origin of the workpiece coordinate system (or the origin of a local coordinate system if set.) If a radius value is specified in the incremental mode, the polar coordinate center is the current position.

Example

Bolt hole circle



- The polar coordinate center is the origin of the workpiece coordinate system.
- The plane is the X-Y plane.

```
N1 G17 G90 G16 ;                        Polar coordinate command, X-Y plane
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ; 100mm radius, 30° angle
N3 Y150.0 ;                             100mm radius, 150° angle
N4 Y270.0 ;                             100mm radius, 270° angle
N5 G15 G80 ;                            Polar coordinate command cancel
```

5.17 COORDINATE SYSTEM SETTING

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system.

Coordinates are specified in one of following three coordinate systems:

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

As required, specify the position that the tool must reach, by using coordinates in a coordinate system.

5.17.1 Machine Coordinate System

The machine coordinate system is a coordinate system whose origin is a machine-specific point used as a machine reference. A coordinate system in which the reference position becomes the parameter-preset coordinate value when manual reference position 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.
Specifying P1 in a G53 block enables the high-speed G53 function.
In this case, this function enables the inter-rapid traverse block overlap function between machine coordinate selection command (G53) and positioning (rapid traverse) command (G00) blocks, thus making it possible to execute the next rapid traverse command (G00) without decelerating to a stop at the end of the machine coordinate selection command (G53). Therefore, high-speed positioning is available even when the machine coordinate selection command (G53) is used.

Format

G53 IP_ P1;

IP_ : Absolute dimension word
P1 : Enables the high-speed G53 function.

5.17.2 Workpiece Coordinate System

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

- Using G92 (G50 when G code system A is used on the T series)
  A workpiece coordinate system is established using the numeric value programmed after G92 (G50 when G code system A is used on the T series).
- Automatic setting
  A workpiece coordinate system is automatically established upon a manual reference position return, as specified in a parameter.
  This method is invalid when the workpiece coordinate system function is used.
- Using G54 to G59
  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.

5.17.2.1 Setting a Workpiece Coordinate System

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

Format

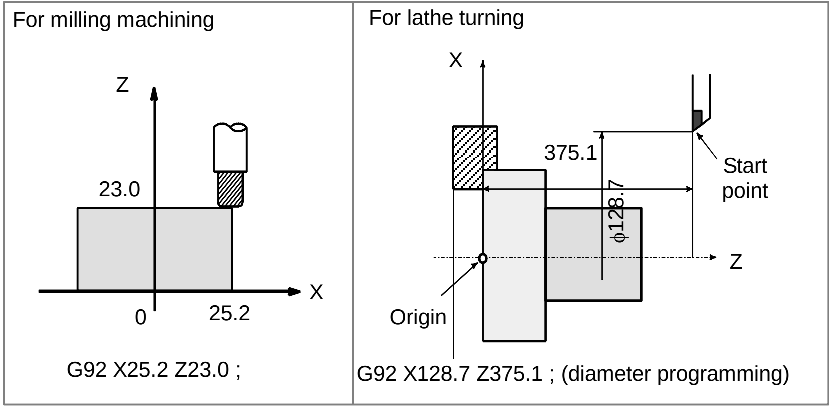
G92 IP_ ;

(G50 when G code system A is used on the T series)

Example

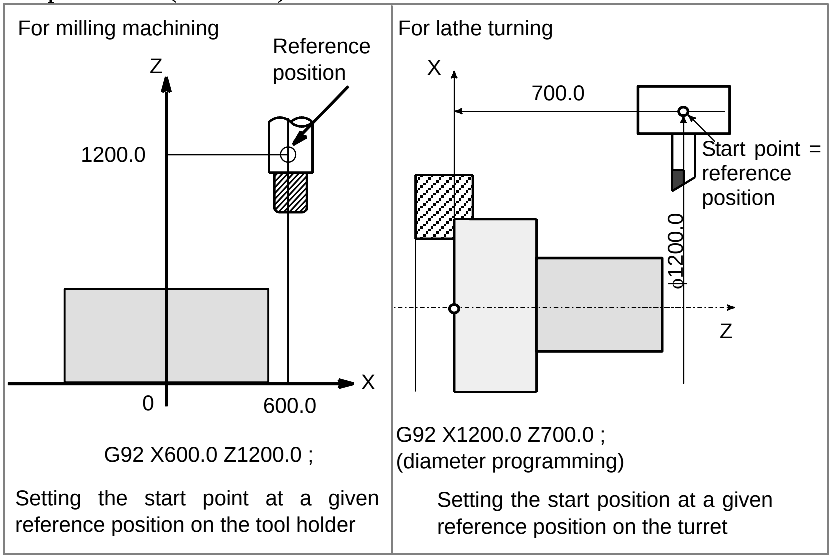
- Example 1

As shown below, place the tool tip at the start point of the program, and specify G92 (G50 when G code system A is used on the T series) at the start of the program.



- Example 2

Set the reference position on the tool holder or turret as shown in the figure below, then specify G92 (G50 when G code system A is used on the T series) at the beginning of the program. By specifying an absolute programming in this condition, the reference position is moved to a specified position. To move the tool tip to a specified position, compensate the distance between the reference position and the tool tip by using tool length compensation (M series) or tool offset.



G92 X600.0 Z1200.0 ;

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

G92 X1200.0 Z700.0 ;
(diameter programming)

Setting the start position at a given reference position on the turret

When a new workpiece coordinate system is created by specifying G92 (G50 when G code system A is used on the T series), 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 (G50 when G code system A is used on the T series) command is useful. In this case, a desired coordinate system can newly be created even if an old workpiece coordinate system is invalid.

5.17.2.2 Automatic Coordinate System Setting

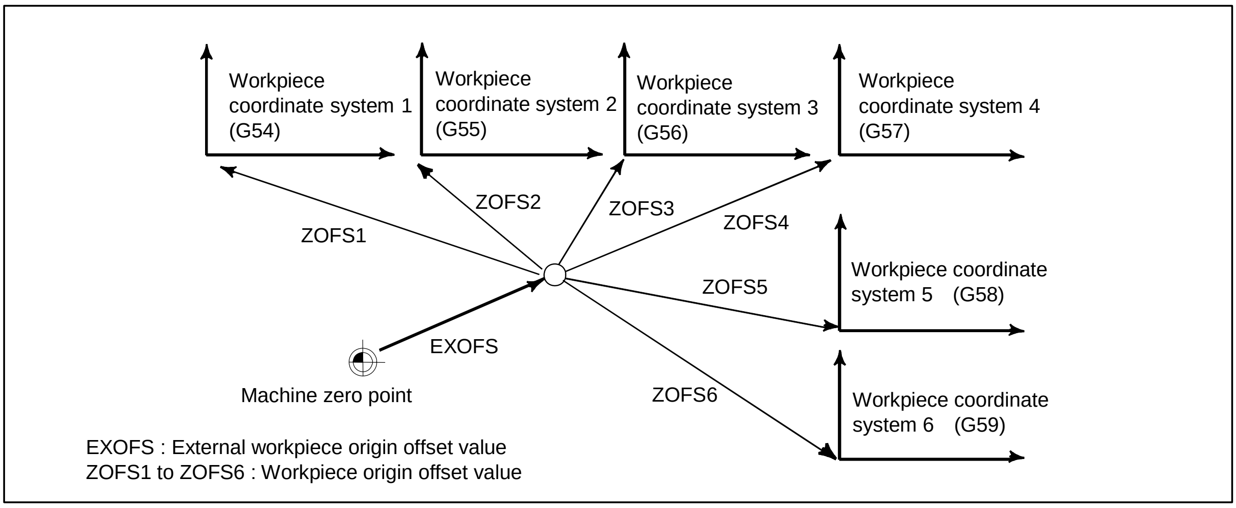
When manual reference position return is performed, a workpiece coordinate system can be set automatically so that the current tool position at the reference position becomes a desired position which is set using a parameter in advance. This functions as if **G92IP__;** (G50 when G code system A is used on the T series) were specified at the reference position.
This function can not be used when the workpiece coordinate system function is used.

5.17.2.3 Setting a Workpiece Coordinate System

Six workpiece coordinate systems can be set. The six workpiece coordinate systems can be changed by changing an external workpiece origin offset value or workpiece origin offset value.
Three methods are available to change an external workpiece origin offset value or workpiece origin offset value.

(1) Inputting from the MDI panel
(2) Programming (using a programmable data input G code or a workpiece coordinate system setting G code)
(3) Using the external data input function
An external workpiece origin offset value can be changed by input signal to CNC.

With a command from G54 to G59, one of six coordinate systems can be selected.
Workpiece coordinate systems 1 to 6 are established correctly when a reference position return operation has been performed after power-up. G54 is selected after the power is turned on.



EXOFS : External workpiece origin offset value
ZOFS1 to ZOFS6 : Workpiece origin offset value

Fig. 5.17 (a) Workpiece coordinate system

Format

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

5.17.3 Local Coordinate System

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. To change the local coordinate system, specify the zero point of a new local coordinate system with G52 in the workpiece coordinate system. 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.

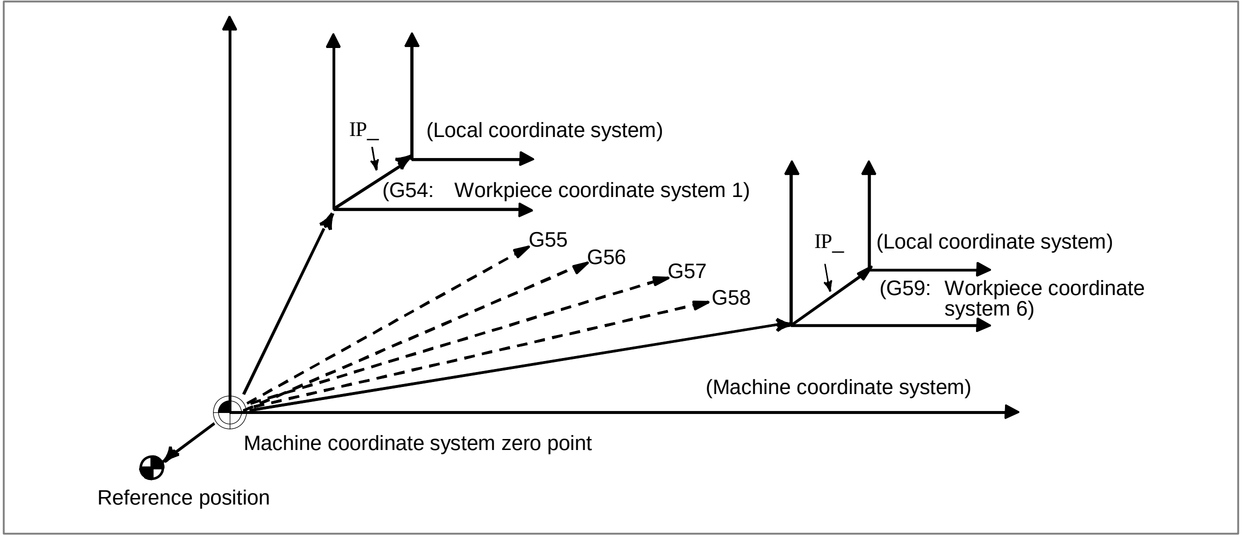


Fig. 5.17 (b) Local coordinate system

Format

G52 IP_ ;	Setting the local coordinate system
G52 IP 0 ;	Canceling of the local coordinate system

Explanation

When local coordinate system is set, local coordinate systems 1 to 6, corresponding to workpiece coordinate systems 1 to 6 is set.

5.18 WORKPIECE COORDINATE SYSTEM PRESET

The workpiece coordinate system with its zero point away by the workpiece origin offset value from the machine coordinate system zero point is set by returning the tool to the reference position 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 position return operation. The set workpiece coordinate system may shift by any of the following commands or operation:

- Manual intervention performed when the manual absolute signal is off
- Move command executed in the machine lock state
- Movement by manual handle interruption
- Operation using the mirror image function
- Shifting the workpiece coordinate system by setting the local coordinate system or workpiece coordinate system

By G code specification or MDI operation, a workpiece coordinate system shifted by an operation above can be preset to a workpiece coordinate system offset from the machine zero point by a workpiece origin offset as in the case of manual reference position return.

Format

G92.1 IP 0;

IP 0 : Specifies axis addresses subject to the workpiece coordinate system preset operation.
Axes that are not specified are not subject to the preset operation.

5.19 EACH AXIS WORKPIECE COORDINATE SYSTEM PRESET SIGNALS

The each axis workpiece coordinate system preset signals are functions for presetting a workpiece coordinate system shifted due to manual intervention, a machine lock, etc. to a workpiece coordinate system offset from the pre-shift machine zero point by a workpiece origin offset value, using an input signal.

5.20 ADDITION OF WORKPIECE COORDINATE SYSTEM PAIR (M SERIES)

M

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece origin offset in an additional workpiece coordinate system:

(1) The workpiece origin offset value setting screen can be used to display and set a workpiece origin offset value.
(2) The G10 function enables a workpiece origin offset value to be set by programming (refer to II-7.2.3).
(3) A custom macro allows a workpiece origin offset value to be handled as a system variable.
(4) Workpiece origin offset data can be entered or output as external data.
(5) The PMC window function enables workpiece origin offset data to be read as program command modal data.

Format

G54.1 Pn ;

n = 1 to 48 (number of the additional workpiece coordinate system)

5.21 DIRECT INPUT OF WORKPIECE ORIGIN OFFSET VALUE MEASURED

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

5.22 MANUAL ABSOLUTE ON AND OFF

Whether the travel distance of the tool manually moved is to be added to the absolute coordinates in the workpiece coordinate system can be selected using an input signal from the PMC.
If the manual absolute signal is turned on, the travel distance of the tool manually moved is added to the absolute coordinates.
If the manual absolute signal is turned off, the travel distance of the tool manually moved is ignored and is not added to the absolute coordinates. So, by the travel distance of the tool manually moved, the workpiece coordinate system can be shifted.

5.23 DIRECT DRAWING DIMENSION PROGRAMMING (T SERIES)

T

Angles of straight lines, chamfering values, corner R values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner R can be inserted between straight lines having an arbitrary angle.

Format

The straight line angle, chamfering value, or corner R must be specified with a comma as follows:

,A_ : Angle
,C_ : Chamfering
,R_ : Corner R

NOTE

When A or C is not used as an axis name, the line angle, chamfering value, or corner R can be specified in the parameter without comma as follows:

A_
C_
R_

5.24 G CODE SYSTEM

5.24.1 G Code for T Series

T

With the CNC for the T series, the G codes listed below are available. Three types of G code systems are usable: A, B, and C. A G code system can be selected by parameter setting.
In this manual, G code system B is used for the descriptions of the preparatory functions below.

Table 5.24.1 (a) G code list

G code system			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
G05.4	G05.4	G05.4		HRV3 on/off
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
◤G13.1 (G113)	◤G13.1 (G113)	◤G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
◤G18	◤G18	◤G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
◤G22	◤G22	◤G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
◤G25	◤G25	◤G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G31	G31	G31		Skip function

G code system			**Group**	**Function**
A	**B**	**C**		
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G36	G36	G36		Automatic tool offset (X axis)
G37	G37	G37		Automatic tool offset (Z axis)
G39	G39	G39		Tool nose radius compensation: corner rounding 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 clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
◤ G50.2 (G250)	◤ G50.2 (G250)	◤ G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5		Cancel composite control
G50.6	G50.6	G50.6		Cancel superimposed control
G51.4	G51.4	G51.4		Start synchronous control
G51.5	G51.5	G51.5		Start composite control
G51.6	G51.6	G51.6		Start superimposed control
G52	G52	G52		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
G61	G61	G61	15	Exact stop mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call
G66	G66	G66	12	Macro modal call
◤ G67	◤ G67	◤ G67		Macro modal call cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
◤ G69	◤ G69	◤ G69		Mirror image off for double turret or balance cutting 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 cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct sizing/grinding cycle (for grinding machine)
G73	G73	G74		Oscillation grinding cycle (for grinding machine)
G74	G74	G75		Oscillation direct sizing/grinding cycle (for grinding machine)

G code system			Group	Function
A	B	C		
◤ G80	◤ G80	◤ G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G81	G81	G81		Spot drilling (FS10/11-T format) Electronic gear box : synchronization start
G82	G82	G82		Counter boring (FS10/11-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS10/11-T format)
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS10/11-T format)
G85	G85	G85	10	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		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check
G96	G96	G96	02	Constant surface speed control
◤ G97	◤ G97	◤ G97		Constant surface speed control cancel
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2		Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3		Spindle indexing completion check
G96.4	G96.4	G96.4		SV speed control mode ON
G98	G94	G94	05	Feed per minute
◤ G99	◤ G95	◤ G95		Feed per revolution
-	◤ G90	◤ G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G98	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the G codes are placed in the states marked with ◤ as indicated in Table.

5.24.2 G Code System for M Series

M

With the CNC for the M series, the G codes listed below are available.

Table 5.24.2 (a) G code list

G code	Group	Function
◤ G00	01	Positioning (rapid traverse)
◤ G01		Linear interpolation (cutting feed)
G02		Circular interpolation CW or helical interpolation CW
G03		Circular interpolation CCW or helical interpolation CCW
G04	00	Dwell, Exact stop
G05.1		AI advanced preview control / AI contour control
G05.4		HRV3 on/off
G07.1 (G107)		Cylindrical interpolation
G09		Exact stop
G10		Programmable data input
G11		Programmable data input mode cancel
◤ G15	17	Polar coordinates command cancel
G16		Polar coordinates command

G code	Group	Function	
◤ G17	02	XpYp plane selection	Xp: X axis or its parallel axis
◤ G18		ZpXp plane selection	Yp: Y axis or its parallel axis
◤ G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20	06	Input in inch	
G21		Input in mm	
◤ G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G31		Skip function	
G33	01	Threading	
G37	00	Automatic tool length measurement	
G39		Cutter compensation : corner circular interpolation	
G40	07	Cutter compensation : cancel	
G41		Cutter compensation : left	
G42		Cutter compensation : right	
G40.1	19	Normal direction control cancel mode	
G41.1		Normal direction control on : left	
G42.1		Normal direction control on : right	
G43	08	Tool length compensation +	
G44		Tool length compensation -	
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 setting	
◤ G54	14	Workpiece coordinate system 1 selection	
G54.1		Additional workpiece coordinate system selection	
G55		Workpiece coordinate system 2 selection	
G56		Workpiece coordinate system 3 selection	
G57		Workpiece coordinate system 4 selection	
G58		Workpiece coordinate system 5 selection	
G59		Workpiece coordinate system 6 selection	
G60	00	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 system rotation mode on	
◤ G69		Coordinate system rotation mode off	
G73	09	Peck drilling cycle	
G74		Left-handed tapping cycle	
G75	01	Plunge grinding cycle (for grinding machine)	

G code	Group	Function
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/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 Electronic gear box : synchronization cancellation
◤ G80.4	34	Electronic gear box : synchronization cancellation
G81.4		Electronic gear box : synchronization start
◤ G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS10/11 format)
G84.3		Left-handed rigid tapping cycle (FS10/11 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
◤ G90	03	Absolute programming
◤ G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
◤ G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
◤ G97		Constant surface speed control cancel
◤ G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
◤ G160	20	In-feed control cancel (for grinding machine)
G161		In-feed control (for grinding machine)

When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power-up or reset, the G codes are placed in the states marked with ◤ as indicated in Table.

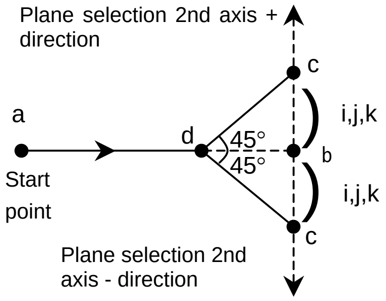
5.25 CHAMFERING AND CORNER R (T SERIES)

T

A chamfer or corner R are can be inserted between two blocks which intersect at a right angle as follows.

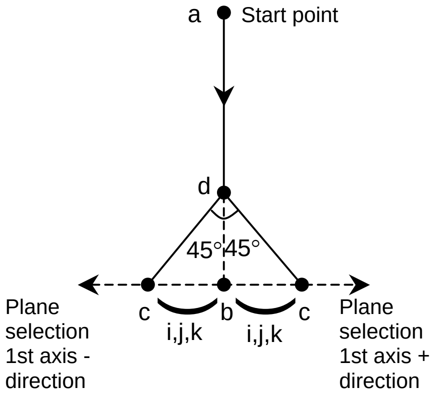
Format

- Chamfering
Plane selection 1st axis → Plane selection 2nd axis
(G17 plane : $X_P \rightarrow Y_P$, G18 plane : $Z_P \rightarrow X_P$, G19 plane : $Y_P \rightarrow Z_P$)

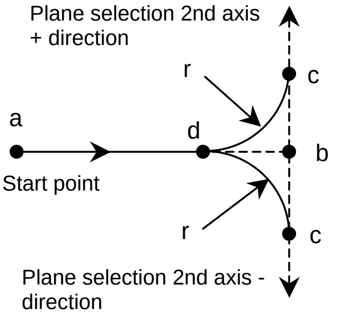
<table>
<tr><th colspan="3">Format</th></tr>
<tr><td colspan="3">**G17 plane : G01 X_P(U)_ J(C)±j ;**
G18 plane : G01 Z_P(W)_ I(C)±i ;
G19 plane : G01 Y_P(V)_ K(C)±k ;</td></tr>
<tr><th colspan="2">Description of symbols</th><th>Tool movement</th></tr>
<tr><td>**X_P(U)_**
Y_P(V)_
Z_P(W)_</td><td>Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.
X_P is the X-axis (basic axis) or its parallel axis, Y_P is the Y-axis (basic axis) or its parallel axis, and Z_P is the Z-axis (basic axis) or its parallel axis.</td><td rowspan="2">

Movements are made from a to d to c.
(In the plane selection 2nd axis + direction if the I, J, K, or C command specifies +, or in the plane selection 2nd axis - direction if I, J, K, or C command specifies -)</td></tr>
<tr><td>**I(C)±i**
J(C)±j
K(C)±k</td><td>Specify the distance between points b and c in the right-hand figure with address J, J, K, or C followed by a sign. (When bit 4 (CCR) of parameter No. 3405 is set to 0, use I, J, or K. When the bit is set to1, use C.)</td></tr>
</table>

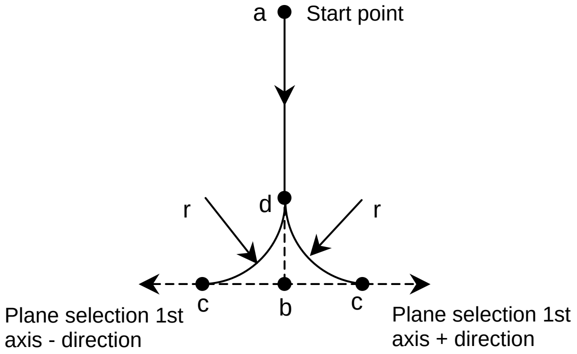
- Chamfering
Plane selection 2nd axis → Plane selection 1st axis
(G17 plane : $Y_P \rightarrow X_P$, G18 plane : $X_P \rightarrow Z_P$, G19 plane : $Z_P \rightarrow Y_P$)

<table>
<tr><th colspan="3">Format</th></tr>
<tr><td colspan="3">G17 plane : G01 Y_P(V)_ I(C)±i ;
G18 plane : G01 X_P(U)_ K(C)±k ;
G19 plane : G01 Z_P(W)_ J(C)±j ;</td></tr>
<tr><th colspan="2">Description of symbols</th><th>Tool movement</th></tr>
<tr><td>X_P(U)_
Y_P(V)_
Z_P(W)_</td><td>Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.
X_P is the X-axis (basic axis) or its parallel axis, Y_P is the Y-axis (basic axis) or its parallel axis, and Z_P is the Z-axis (basic axis) or its parallel axis.</td><td rowspan="2">Movements are made from a to d to c.
(In the plane selection 1st axis + direction if the I, J, K, or C command specifies +, or in the plane selection 1st axis - direction if I, J, K, or C command specifies -)

 </td></tr>
<tr><td>I(C)±i
J(C)±j
K(C)±k</td><td>Specify the distance between points b and c in the right-hand figure with address J, J, K, or C followed by a sign. (When bit 4 (CCR) of parameter No. 3405 is set to 0, use I, J, or K. When the bit is set to1, use C.)</td></tr>
</table>

- Corner R
Plane selection 1st axis → Plane selection 2nd axis
(G17 plane : $X_P \rightarrow Y_P$, G18 plane : $Z_P \rightarrow X_P$, G19 plane : $Y_P \rightarrow Z_P$)

<table>
<tr><th colspan="3">Format</th></tr>
<tr><td colspan="3">G17 plane : G01 X_P(U)_ R±r ;
G18 plane : G01 Z_P(W)_ R±r ;
G19 plane : G01 Y_P(V)_ R±r ;</td></tr>
<tr><th colspan="2">Description of symbols</th><th>Tool movement</th></tr>
<tr><td>X_P(U)_
Y_P(V)_
Z_P(W)_</td><td>Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.
X_P is the X-axis (basic axis) or its parallel axis, Y_P is the Y-axis (basic axis) or its parallel axis, and Z_P is the Z-axis (basic axis) or its parallel axis.</td><td rowspan="2">
Movements are made from a to d to c.
(In the plane selection 2nd axis + direction if the R command specifies +r, or in the plane selection 2nd axis - direction if the R command specifies -r)</td></tr>
<tr><td>R±r</td><td>Specify the radius of an arc connecting points d and c in the right-hand figure with address R followed by a sign.</td></tr>
</table>

- Corner R
Plane selection 2nd axis → Plane selection 1st axis
(G17 plane : $Y_P \rightarrow X_P$, G18 plane : $X_P \rightarrow Z_P$, G19 plane : $Z_P \rightarrow Y_P$)

<table>
<tr><th colspan="3">Format</th></tr>
<tr><td colspan="3">G17 plane : G01 Y_P(V)_ R±r ;
G18 plane : G01 X_P(U)_ R±r ;
G19 plane : G01 Z_P(W)_ R±r ;</td></tr>
<tr><th colspan="2">Description of symbols</th><th>Tool movement</th></tr>
<tr><td>X_P(U)_
Y_P(V)_
Z_P(W)_</td><td>Specifies movement from point a to point b with an absolute or incremental programming in the figure on the right.
X_P is the X-axis (basic axis) or its parallel axis, Y_P is the Y-axis (basic axis) or its parallel axis, and Z_P is the Z-axis (basic axis) or its parallel axis.</td><td rowspan="2">Movements are made from a to d to c.
(In the plane selection 1st axis + direction if the R command specifies +r, or in the plane selection 1st axis - direction if the R command specifies -r)
 </td></tr>
<tr><td>R±r</td><td>Specify the radius of an arc connecting points d and c in the right-hand figure with address R followed by a sign.</td></tr>
</table>

5.26 OPTIONAL CHAMFERING AND CORNER R (M SERIES)

M

Chamfering and corner R blocks can be inserted automatically between the following:

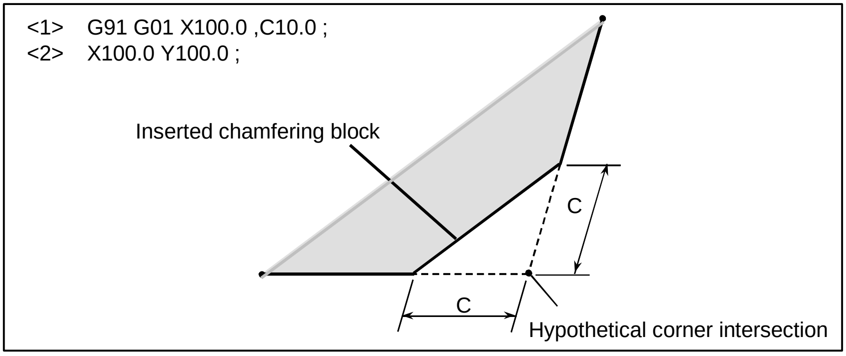
- Between linear interpolation and linear interpolation blocks
- Between linear interpolation and circular interpolation blocks
- Between circular interpolation and linear interpolation blocks
- Between circular interpolation and circular interpolation blocks

- Chamfering

When the above specification is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02 or G03), a chamfering block is inserted.

,C_

After C, specify the distance from the hypothetical corner intersection to the start and end points. The hypothetical corner point is the corner point that would exist if chamfering were not performed.

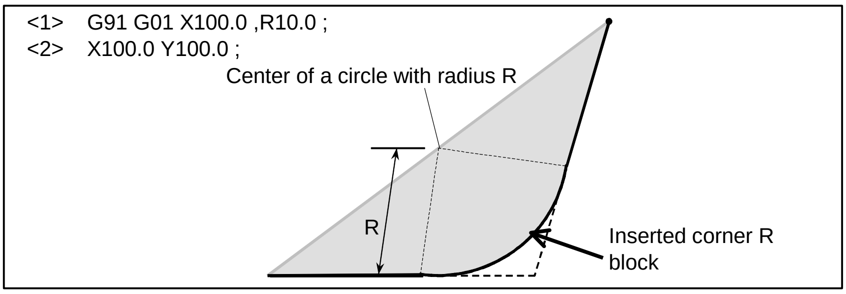


- Corner R

When the above specification is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02 or G03), a corner R block is inserted.

,R_

After R, specify the radius for corner R.



5.27 PROGRAMMABLE DATA INPUT

By executing programs specified in the formats after a G10 command, various types of data as indicated in the table below can be set.

Data	Format
Pitch error compensation	G10 L50
Workpiece origin offset value	G10 L2
Tool compensation value (M series)	G10 L10/L11/L12/L13
Tool compensation value (T series)	G10 L10/L11

Data	Format
Data input mode cancel	G11

5.27.1 Setting the Pitch Error Compensation Data

By using a programmed command, pitch error compensation data can be set.
This function can be used, for example, to modify pitch error compensation data at the time of attachment replacement.

Format

G10L50 ;	Pitch error compensation data entry mode setting
N_R_ ;	Pitch error compensation data entry
:	
G11 ;	Pitch error compensation data entry mode cancel

N_ : Compensation position number for pitch errors compensation +10,000
R_ : Pitch error compensation data

5.27.2 Setting the Workpiece Origin Offset Value

By specifying a G10 command, a workpiece origin offset modification can be made. When G10 is specified in the absolute mode (G90), a specified workpiece origin offset becomes a new workpiece origin offset. When G10 is specified in the incremental mode (G91), a specified workpiece origin offset added to the currently set workpiece origin offset becomes a new workpiece origin offset.

Format

G10 L2 Pp IP_ ;

p=0 : External workpiece origin offset value
p=1 to 6 : Workpiece origin offset value correspond to workpiece coordinate system 1 to 6
IP_ : For an absolute command, workpiece origin offset for each axis.
For an incremental command, value to be added to the set workpiece origin offset for each axis (the result of addition becomes the new workpiece origin offset).

5.27.3 Setting the Tool Compensation Offset Value

M

By specifying a G10 command, a tool compensation value setting/modification can be made.
When G10 is specified in the absolute mode (G90), a specified value becomes a new tool compensation value.
When G10 is specified in the incremental mode (G91), a specified value added to the currently set tool compensation value becomes a new tool compensation value.

Format

- For tool compensation memory A

G10 L11 P_ R_ Q_ ;

P_ : Tool compensation number
R_ : Tool compensation value
Q_ : Imaginary tool nose number

- For tool compensation memory C

G10 L_ P_ R_ Q_ ;

L_ : Type of compensation memory
  L10 : Geometry compensation value corresponding to an H code
  L11 : Wear compensation value corresponding to an H code
  L12 : Geometry compensation value corresponding to a D code
  L13 : Wear compensation corresponding to a D code
P_ : Tool compensation number
R_ : Tool compensation value
Q_ : Imaginary tool nose number

NOTE

1 Address R follows the increment system for tool offset values.
2 If L is omitted for compatibility with the conventional CNC format, or L1 is specified, the same operation as when L11 is specified is performed.
3 Set a imaginary tool nose number when the cutter compensation function is specified and a imaginary tool nose direction is used.

T

By specifying a G10 command, a tool compensation value setting/modification can be made.
If G code system B or C is used, a specified value becomes a new tool compensation value when G10 is specified in the absolute mode (G90). When G10 is specified in the incremental mode (G91), a specified value added to the currently set tool compensation value becomes a new tool compensation value. However, address R is handled as an absolute command, and address C is handled as an incremental command, regardless of the setting of G90/G91.

Format

- For G code system A

G10 P_ X_ Z_ R_ Q_ Y_ ;
or
G10 P_ U_ W_ C_ Q_ V_ ;

P_ : Tool compensation number
  0 : Workpiece coordinate system shift command
  Offset number : Tool compensation value or tool wear compensation value command
  10000 + offset number : Tool geometry compensation value command
X_ : X axis compensation value (absolute value)
U_ : X axis compensation value (incremental value)
Z_ : Z axis compensation value (absolute value)
W_ : Z axis compensation value (incremental value)
R_ : Tool nose radius compensation value (absolute value)
C_ : Tool nose radius compensation value (incremental value)
Y_ : Y axis compensation value (absolute value)
V_ : Y axis compensation value (incremental value)
Q_ : Imaginary tool nose number

- For G code system B/C

G10 P_ X_ Z_ R_ Q_ Y_ ;

or

G10 P_ X_ Z_ C_ Q_ Y_ ;

X_ : X axis compensation value
Z_ : Z axis compensation value
R_ : Tool nose radius compensation value (absolute value)
C_ : Tool nose radius compensation value (incremental value)
Y_ : Y axis compensation value
Q_ : Imaginary tool nose number

When the Series 10/11 program format is valid, the following format can be used in addition to the formats above:

G10 L_ P_ X_ Z_ R_ Q_ Y_ ; (G code system A/B/C)

or

G10 L_ P_ U_ W_ C_ Q_ V_ ; (G code system A)

L_ : Type of tool compensation memory
  L10 : Geometry compensation value
  L11 : Wear compensation value
P_ : Tool compensation number

NOTE

1. Addresses X, Z, R, Y, U, W, C, and V follow the increment system of tool offset values.
2. Address C used for a tool nose radius compensation value is valid only when an axis named C is used.
3. Set a tool nose radius compensation value or imaginary tool nose number the tool nose radius compensation function is used.
4. Set a Y axis compensation value when the Y-axis offset function is used.
5. Set a tool geometry compensation value when the tool geometry/wear compensation function is used.
6. For compatibility with the conventional models, addresses U, W, and V are handled as incremental commands for X, Z, and Y axis compensation values, respectively, when G code system B or C is used.

5.28 PROGRAMMABLE PARAMETER INPUT

By programming, the values of parameters can be set or change.
This function can be used for applications such as modification to a maximum cutting feedrate or cutting time constant due to a machining condition change.
This function is used for the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

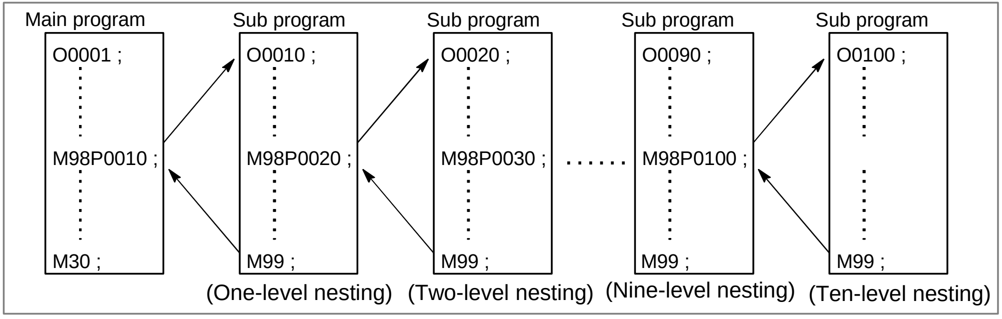
G10 L52 ; Parameter entry mode setting
N_ R_ ; For parameters other than the axis type
N_ P_ R_ ; For axis type parameters
:
G11 ; Parameter entry mode cancel

N_ : Parameter number
R_ : Parameter setting value (Leading zeros can be omitted.)
P_ : Axis number 1 to maximum controlled axis number (to be specified when an axis type parameter or spindle type parameter is specified)

NOTE
Some parameters cannot be set, when this command is used.

5.29 SUB PROGRAM CALL

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a sub program in memory to simplify the program. M98 is used to call a sub program, and M99 is used to return from a sub program. Up to ten levels of sub programs can be nested.



Format

- Sub program configuration

One sub program

```
Oxxxx ;
  :
  :
M99;
```

Sub program number (or the colon (:) optionally in the case of ISO)
Program end

M99 need not constitute a separate block as indicated below.

Example) X100.0 Y100.0 M99;

- Sub program call

M98 Pxxxx xxxx ;

Sub program number

Number of times the sub program is called repeatedly

or

M98 Pxxxx Lxxxxxxxx ;

Sub program number

Number of times the sub program is called repeatedly

NOTE

1. When a sub program is called repeatedly (using an 8-digit number preceded by P), if the number of digits of a sub program number is less than 4, the higher-order bits need to be padded with 0's.
   Example)
   P100100: Call sub program No. 100 ten times.
   P50001: Call sub program No. 1 five times.
2. When the number of repetitions is omitted, 1 is assumed. In this case, the number of digits of a subprogram number does not need to be 4.
3. When a sub program is called repeatedly (using an 8-digit number preceded by P), do not specify address L in the same block.

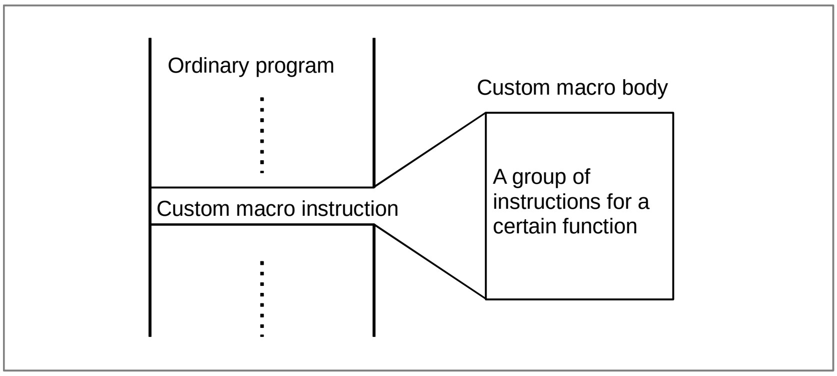
- Return from a sub program

M99 ;

need not be specified in a block that specifies no other commands.

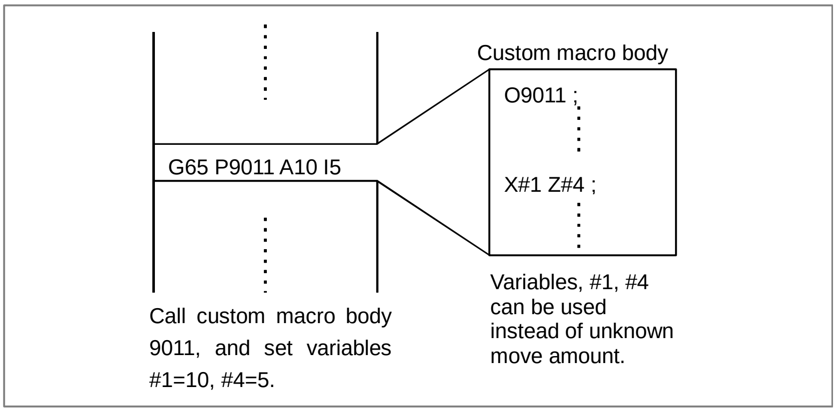
5.30 CUSTOM MACRO

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.

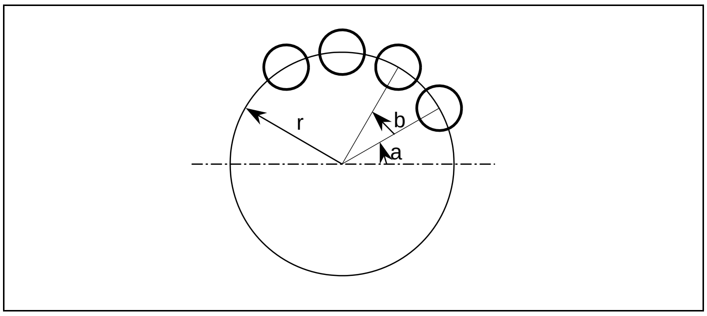


The programmer need not remember all the instructions in the custom macro body. He needs only to remember the representative, custom macro instruction.

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.



Example



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.

Explanation

- Use of Variable

Variables : #i (i=1, 2, 3, . . .)
Quotation of variables : F#33 (#33: speed expressed by 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), MOD (remainder), SIN (sine), ASIN (arc sine), COS (cosine), ACOS (arc 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), LN (natural logarithm), EXP (exponential function), POW (power), ADP (decimal point addition)

Example) #5=SIN [[#2+#4]*3.14+#4]*ABS[#10]

- Control command

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

- **If [<conditional expression>] GOTO n (n = sequence number)**

  When <conditional expression> is satisfied, the next execution is done from block with sequence number n.

  When <conditional expression> is not satisfied, the next block is executed.

  When the [<IF conditional expression>] is committed, it executes from block with n unconditionally.

  The following <conditional expressions> are available:

  #j EQ #k whether #j = #k
  #j NE #k whether #j ≠ #k
  #j GT #k whether #j > #k
  #j LT #k whether #j < #k
  #j GE #k whether #j ≥ #k
  #j LE #k whether #j ≤ #k

- **IF [<conditional expression>] THEN macro statement**

  If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

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

  While <conditional expression> is satisfied, blocks from DO m to END m is repeated.

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

Example)

```
#120=1 ;
   WHILE [#120 LE 10] DO 1 ;

    #120=#120+1 ;
    END1
```

Repeated 10 times.

- Format of custom macro body

The format is the same as the sub program.

O Macro number ;

Custom macro body

M99

- Custom macro instruction

- **Simple call**

  G65 P (macro number) L (times to repeat) <argument assignment> ;

  A value is set to a variable by <argument assignment>.

  Write the actual value after the address.

Example)

A5.0 E3.2 M13.4

There is a regulation on which address (A - Z) corresponds to which variable number.

- **Modal call (calling a move command)**

  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, for example, a drilling cycle is created as a custom macro body.

- **Macro call using G code**

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

  N_ G65 Pxxxx <argument assignment> ;

  macro can be called just by simple commanding:

  N_ Gyy <argument assignment> ;

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

- **Macro call using M code**

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

  The following command

  N_ G65 Pxxxx <Argument assignment> ;

  is equivalent to the following command:

  N_ Myy <Argument assignment> ;

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

  M codes are not sent out the same as the sub program call using M code.

- **Sub program call using M code**

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

  N_ G_ X_ Y_ . . . M98 Pxxxx ; ,

  the same operation can be performed simply by commanding:

  N_ G_ X_ Y_ . . . Myy ;.

  As for M98, M codes are not transmitted.

  The M code (Myy) for calling the sub program and the sub program number (Pxxxx) to be called are coupled together and set by parameter.

- **Sub program call using 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 code t is stored as arguments of common variable #149.

- **Sub program call using specific address**

  By setting parameter, sub program can be called by specific addresses (A). When commanded:

  N_ G_ X_ Y_ . . . Ab ;

  the same operation is done as when commanded:

  #146=b ;

  N_ G_ X_ Y_ . . . M98 P9004 ;

  The A address b is stored as arguments of common variable #146.

  However, a restriction is imposed on the codes that can be set in specific addresses.

- 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 to #33**

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

- **Common variables #100 - #199, #500 - #999**

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

  Common variables basically have the read/write attribute. By parameter setting, however, the common variable with a specified variable number can be protected (to make the common variable read-only). The system does not define common variable applications. Instead, the user can freely use common variables.

  Common variables #100 to #199 are cleared when the power is turned off. The values of common variables #500 to #999 are preserved even when the power is turned off.

- **System variables**

  A variable with a certain variable number has a certain value. If the variable number is changed, the certain value is also changed. The certain value are the following:
  - 128 DI points (for read only)
  - 128 DO points (for output only)
  - Tool compensation value, workpiece origin offset value, and workpiece coordinate system value (T series)
  - Position information (current 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, auxiliary function end wait hold
  - Feed hold, Feedrate 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.

- Limitation

- **Usable variables**

  See "Types of variables".
- **Usable variable values**

  Maximum : ±10^{308} (bit 0 (F0C) of parameter No.6008 = 0)
  ±10^{47} (bit 0 (F0C) of parameter No.6008 = 1)
  Minimum : ±10^{-308} (bit 0 (F0C) of parameter No.6008 = 0)
  ±10^{-29} (bit 0 (F0C) of parameter No.6008 = 1)
- **Constants usable in <expression>**

  Maximum : ±999999999999
  Minimum : ±0.00000000001
  Decimal point allowed

- **Arithmetic precision**
  15-digit decimal number (bit 0 (F0C) of parameter No.6008 = 0)
  When bit 0 (F0C) of parameter No. 6008 is 1, the arithmetic precision is shown in the table below.

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error $\left\lvert\frac{\varepsilon}{a}\right\rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
a = $\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	$\mathrm{MIN}\left\lvert\frac{\varepsilon}{b}\right\rvert, \left\lvert\frac{\varepsilon}{c}\right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error $\lvert\varepsilon\rvert$ degrees
a = ATAN [b] / [c]	1.8×10^{-6}	3.6×10^{-6}	

NOTE

1 The relative error depends on the result of the operation.
2 Smaller of the two types of errors is used.
3 The absolute error is constant, regardless of the result of the operation.

- **Macro call nesting**
  Maximum 5 folds.
- **Repeated ID numbers used in DO to END**
  1 to 3
- **[ ] nesting used in a custom macro expression**
  Maximum 5 folds.
- **Sub program call nesting**
  Up to 10 folds in a sub program call
  Up to 15 folds including macro call nesting

5.31 ADDITION OF CUSTOM MACRO COMMON VARIABLES

600 common variables consisting of #100 to #199 and #500 to #999 can be used. Common variables #100 to #199 are cleared when the power is turned off. The values of common variables #500 to #999 are preserved even when the power is turned off.

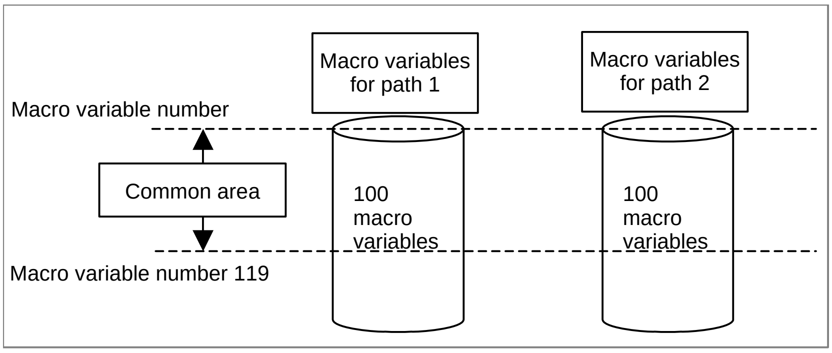
5.32 CUSTOM MACRO COMMON VARIABLES BETWEEN EACH PATH (T SERIES)

T

In a 2-path system, some custom macro common variables or all custom macro common variables can be made common to both paths by parameter setting.

Example)

When macro variable numbers 100 to 119 are made common to path 1 and path 2



5.33 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine.
This function is referred to as an interruption type custom macro function.

Format

M96 Pxxxx ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

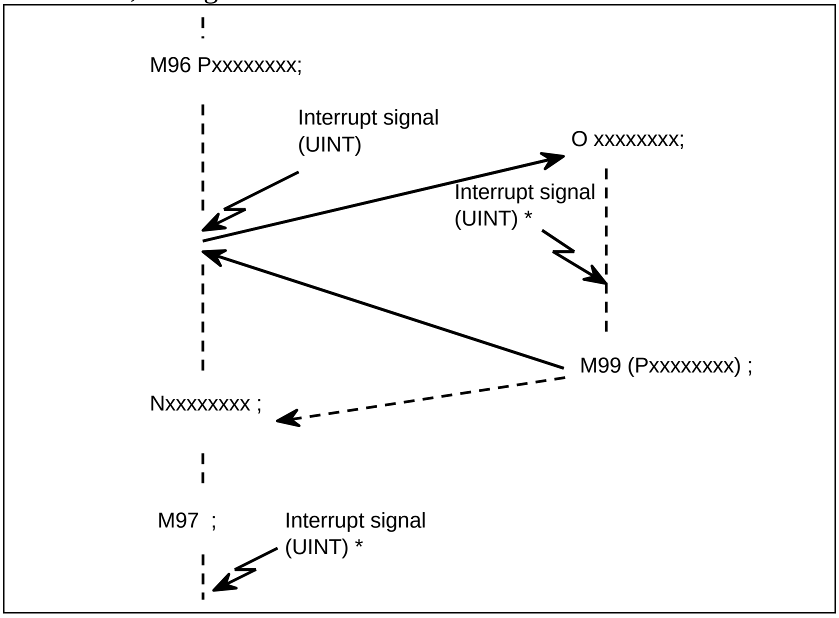
Explanation

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.
(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.
(3) At regular intervals, information on current machining is read.
Listed above are examples like adaptive control applications of the interruption type custom macro function.

Example)

When M96 Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx in following figure. When the interrupt signal ((UINT)** and (UINT)* in the figure) is input during execution of the interrupt program or after M97, it is ignored.



5.34 PATTERN DATA INPUT

In the program of the fixed form processing with the custom macro, the operator select the processing pattern on the menu screen and specified the size, number and so on to the variable on the custom macro screen. As above mentioned, this function enables users to perform programming simply without programming using an existing NC language.

With the aid of this function, a machine tool builder can prepare the program of a hole machining cycle (such as a boring cycle or tapping cycle) using the custom macro function, and can store it into the program memory.

This cycle is assigned pattern names, such as BOR1, TAP3, and DRL2.

An operator can select a pattern from the menu of pattern names displayed on the screen.

Data (pattern data) which is to be specified by the operator should be created in advance with variables in a drilling cycle.

The operator can identify these variables using names such as DEPTH, RETURN RELIEF, FEED, MATERIAL or other pattern data names. The operator assigns values (pattern data) to these names.

The operator selects the pattern on the menu screen, and the selected pattern number is assigned to the system variable. The custom macro of the selected pattern can be started by starting a program then referring to the system variable in the program.

5.35 CANNED CYCLE (T SERIES)

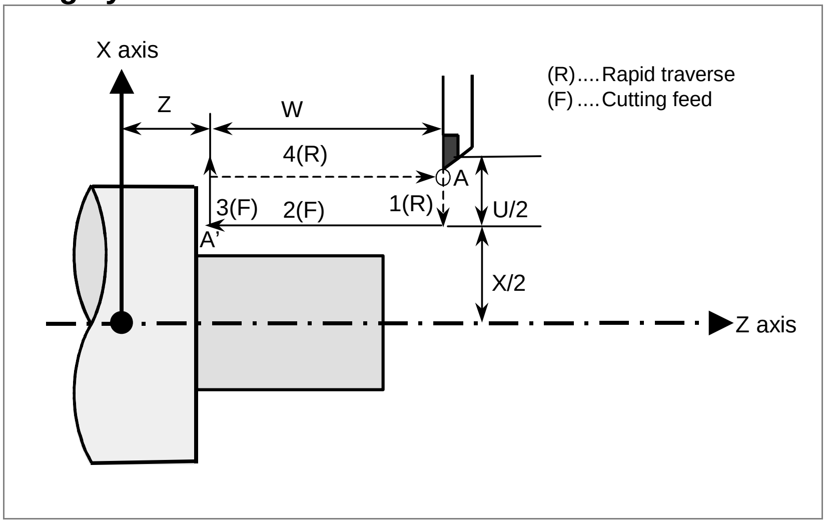
T

There are three canned cycles : the outer diameter/internal diameter cutting canned cycle (G77), the threading canned cycle (G78), and the end face turning canned cycle (G79).

5.35.1 Outer Diameter/Internal Diameter Cutting Cycle (T Series)

Format

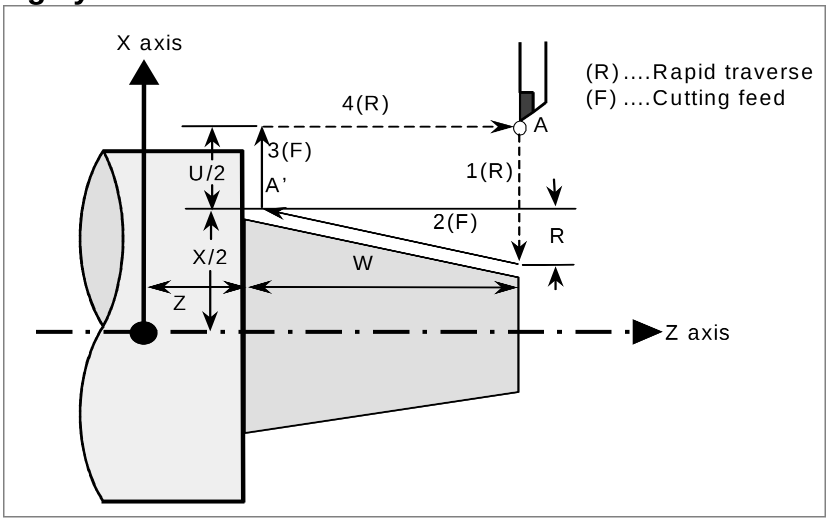
- **Straight cutting cycle**



G77 X(U)_ Z(W)_ F_ ; (G90 with G code system A)

X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the length
U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the length
F_ : Cutting feedrate

- **Taper cutting cycle**



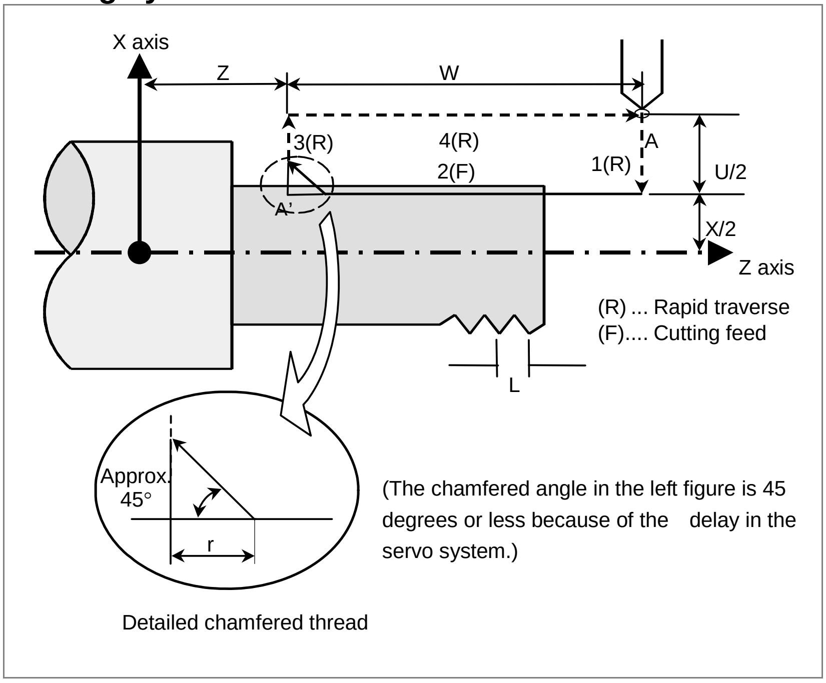
G77 X(U)_ Z(W)_ R_ F_ ; (G90 with G code system A)

X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the length
U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the length
R_ : Taper amount (R in the above figure)
F_ : Cutting feedrate

5.35.2 Threading Cycle (T Series)

Format

- **Straight threading cycle**



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

Detailed chamfered thread

G78 X(U)_ Z(W)_ F_ Q_ ; (G92 with G code system A)

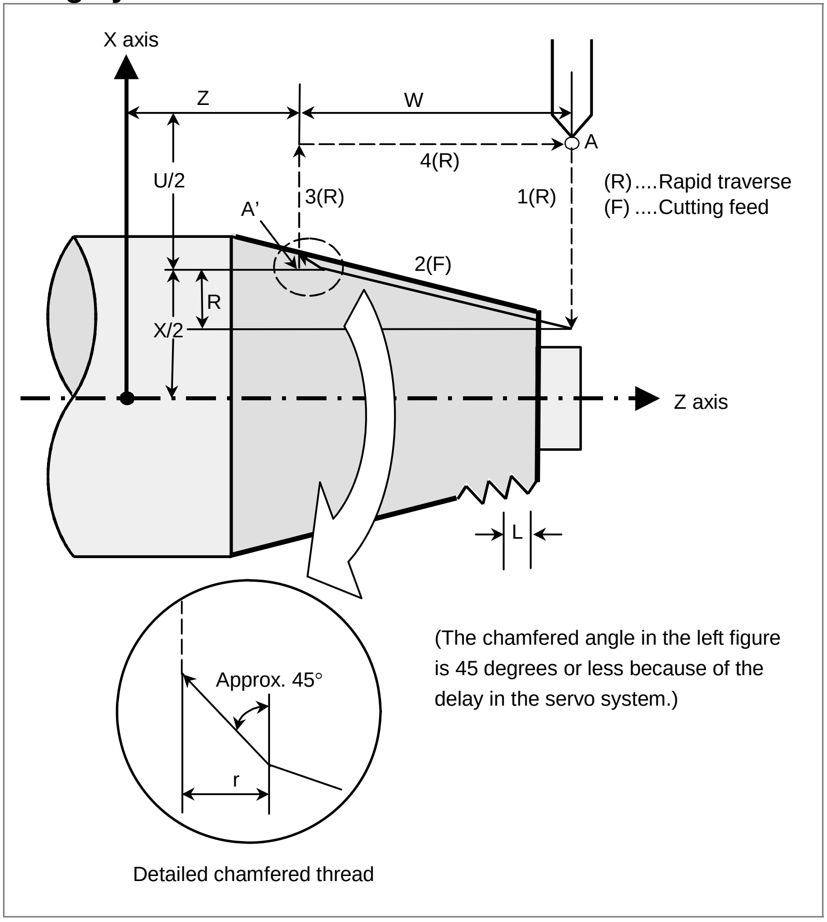
X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the length

U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the length

Q_ : Angle for shifting the threading start angle
(Increment: 0.001 degrees,
Valid setting range: 0 to 360 degrees)

F_ : Thread lead (L in the above figure)

- Taper threading cycle



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

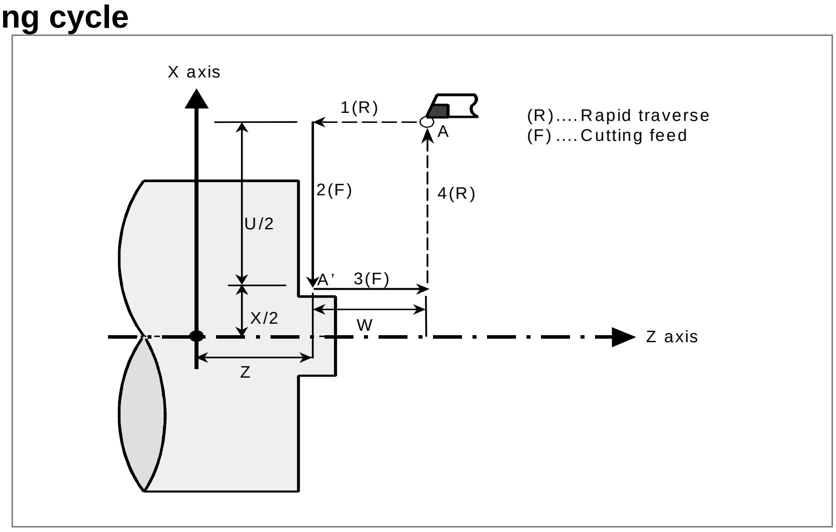
G78 X(U)_ Z(W)_ R_ F_ Q_ ; (G92 with G code system A)

X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the length
U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the length
Q_ : Angle for shifting the threading start angle (Increment: 0.001 degrees, Valid setting range: 0 to 360 degrees)
R_ : Taper amount (R in the above figure)
F_ : Thread lead (L in the above figure)

5.35.3 End Face Turning Cycle (T Series)

Format

- Face cutting cycle



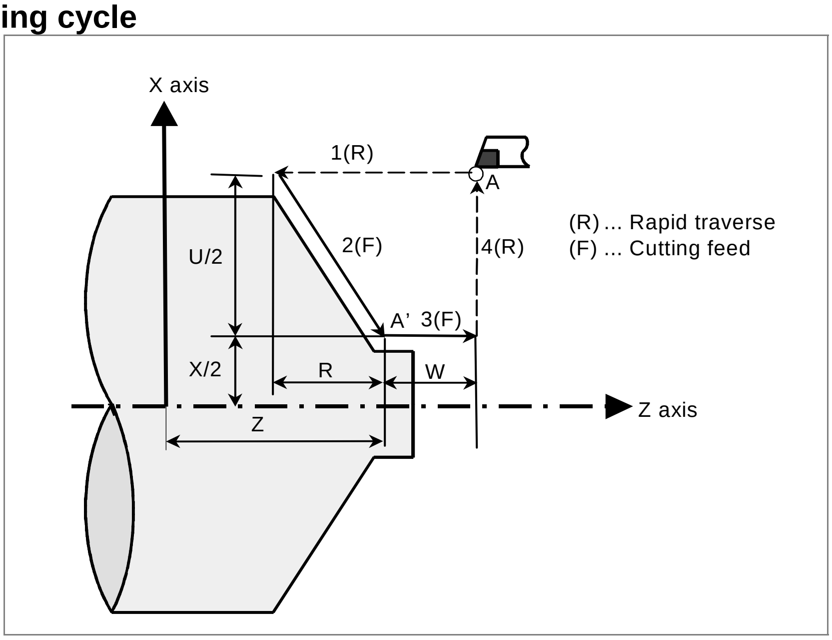
G79 X(U)_ Z(W)_ F_ ; (G94 with G code system A)

X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the end face

U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the end face

F_ : Cutting feedrate

- Taper cutting cycle



G79 X(U)_ Z(W)_ R_ F_ ; (G94 with G code system A)

X_,Z_ : Coordinates of the cutting end point (point A' in the above figure) in the direction of the end face
U_,W_ : Travel distance to the cutting end point (point A' in the above figure) in the direction of the end face
R_ : Taper amount (R in the above figure)
F_ : Cutting feedrate

5.36 MULTIPLE REPETITIVE CYCLE (T SERIES)

T

This function is canned cycles to make CNC programming easy. For instance, the data of the target figure (finishing shape). describes the tool path for rough machining. And also, a canned cycles for the threading is available.

5.36.1 Stock Removal in Turning (T Series)

There are two types of stock removals in turning : Type I and II.

Format

ZpXp plane

G71 U(Δd) R(e) ;

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

N (ns) ;

...

N (nf) ;

The move commands for the target figure from A to A' to B are specified in the blocks with sequence numbers ns to nf.

YpZp plane

G71 W(Δd) R(e) ;

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

N (ns) ;

...

N (nf) ;

XpYp plane

G71 V(Δd) R(e) ;

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

N (ns) ;

...

N (nf) ;

Δd : Depth of cut
The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

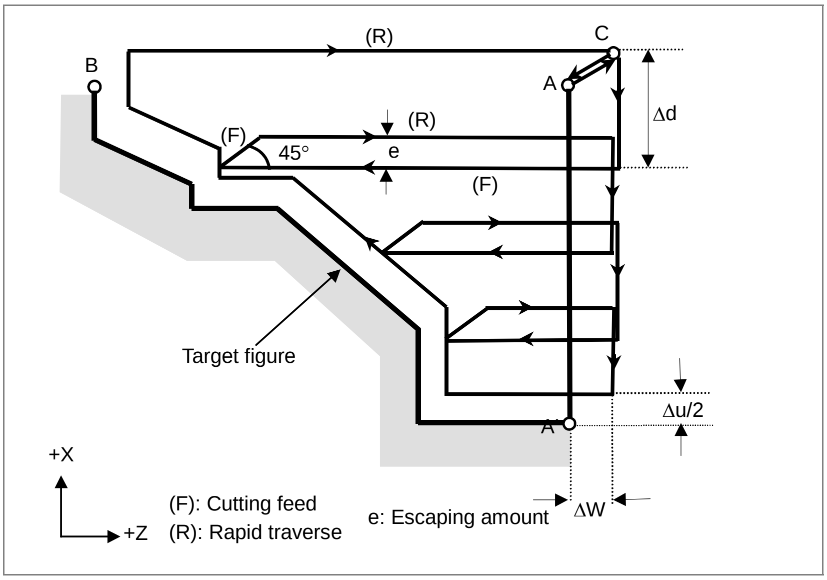
Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)

Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

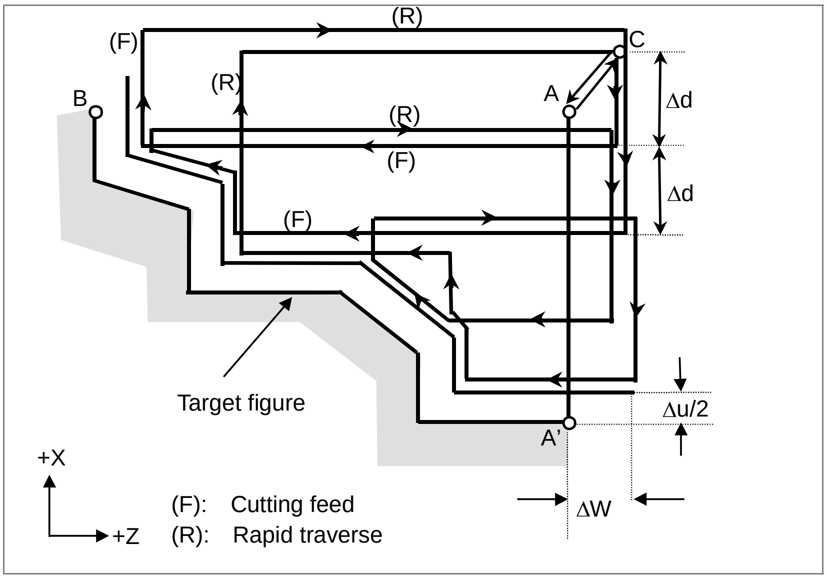
Explanation

- Type I



When the target figure of A → A' → B is programmed, cutting is performed with a depth of cut of Δd per cut, leaving the finishing allowances Δu/2 and Δw. After the last cut is performed in the plane second axis direction (X-axis direction in the case of the ZX plane), rough finishing is performed along the target figure. Upon completion of rough finishing, the block that follows the sequence block specified by Q is executed.

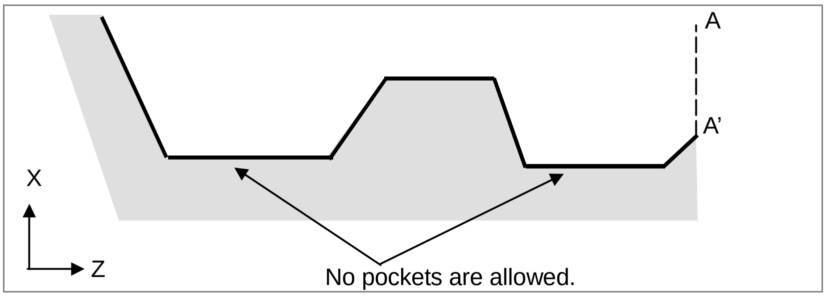
- Type II



When a target figure passing through A, A', and B in this order is given by the program for a target figure as shown in the figure, the specified area is removed by Δd (depth of cut), with the finishing allowance specified by Δu/2 and Δw left. Type II differs from type I in cutting the workpiece along the figure after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane).
After the last cutting, the tool returns to the start point specified in G71 and rough cutting is performed as finishing along the target figure, with the finishing allowance specified by Δu/2 and Δw left.

- Selection of type I or II

When the target figure has pockets, be sure to use type II.



No pockets are allowed.

Escaping operation after rough cutting in the direction of the first axis on the plane (Z-axis for the ZX plane) differs between types I and II. With type I, the tool escapes to the direction of 45 degrees. With type II, the tool cuts the workpiece along the target figure. When the target figure has no pockets, determine the desired escaping operation and select type I or II.

- Selecting type I or II

In the start block for the target figure (sequence number ns), select type I or II.

(1) When type I is selected
Specify the second axis on the plane (X-axis for the ZX plane). Do not specify the first axis on the plane (Z-axis for the ZX plane).

(2) When type II is selected
Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).
When you want to use type II without moving the tool along the first axis on the plane (Z-axis for the ZX plane), specify the incremental programming with travel distance 0 (W0 for the ZX plane).
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)_;
   :                     :
   :                     :
N200.............;    N200.............;
```

5.36.2 Stock Removal in Facing (T Series)

This cycle is the same as G71 except that cutting is performed by an operation parallel to the second axis on the plane (X-axis for the ZX plane).

Format

ZpXp plane

G72 W(Δd) R(e) ;
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

The move commands for the target figure from A to A’ to B are specified in the blocks with sequence numbers ns to nf.

YpZp plane

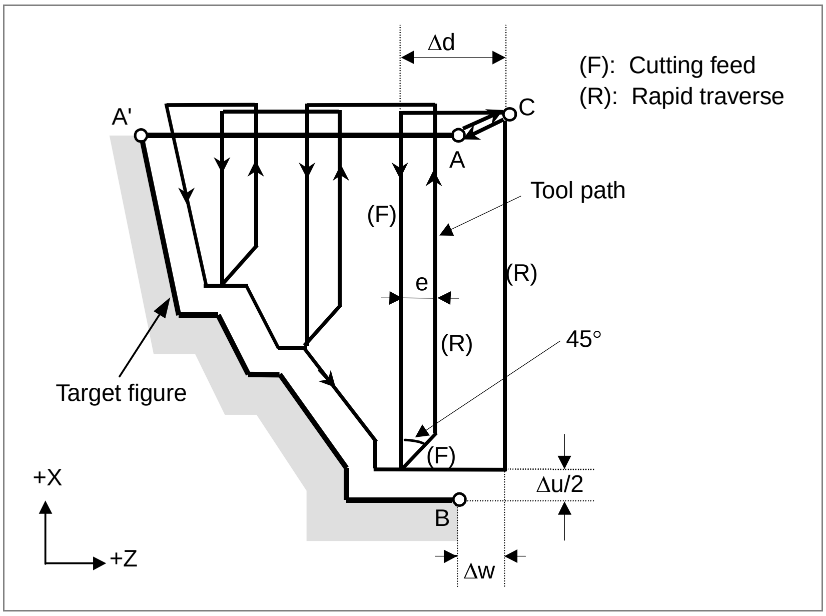
G72 V(Δd) R(e) ;
G72 P(ns) Q(nf) V(Δw) W(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

XpYp plane

G72 U(Δd) R(e) ;
G72 P(ns) Q(nf) U(Δw) W(Δu) F(f) S(s) T(t) ;
N (ns) ;
...
N (nf) ;

Δd : Depth of cut
The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.
e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.
ns : Sequence number of the first block for the program of finishing shape.
nf : Sequence number of the last block for the program of finishing shape.
Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)
Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)
f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G72 block is effective.

Explanation



When a target figure passing through A, A', and B in this order is given by a program, the specified area is removed by Δd (depth of cut), with the finishing allowance specified by Δu/2 and Δw left.

- Selection of type I or II

For G72, there are types I and II.
When the target figure has pockets, be sure to use type II.
Escaping operation after rough cutting in the direction of the second axis on the plane (X-axis for the ZX plane) differs between types I and II. With type I, the tool escapes to the direction of 45 degrees. With type II, the tool cuts the workpiece along the target figure. When the target figure has no pockets, determine the desired escaping operation and select type I or II.

- Selecting type I or II

In the start block for the target figure (sequence number ns), select type I or II.

(1) When type I is selected
Specify the first axis on the plane (Z-axis for the ZX plane). Do not specify the second axis on the plane (X-axis for the ZX plane).

(2) When type II is selected
Specify the second axis on the plane (X-axis for the ZX plane) and first axis on the plane (Z-axis for the ZX plane).
When you want to use type II without moving the tool along the second axis on the plane (X-axis for the ZX plane), specify the incremental programming with travel distance 0 (U0 for the ZX plane).
Example)

```
Type I                Type II
G72 10.0 R5.0 ;       G72 10.0 R5.0 ;
G72 P100 Q200....;    G72 P100 Q200....;
N100 Z(W)_ ;          N100 X(U)_ Z(W)_;
   :
   :
N200.............;    N200.............;
```

- Differences from G71 (Type I)

G72 differs from G71 in the following points:

(1) G72 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).

(2) In the start block in the program for a target figure (block with sequence number ns), only the first axis on the plane (Z-axis (W-axis) for the ZX plane) must be specified.

- Differences from G71 (Type II)

G72 differs from G71 in the following points:

(1) G72 cuts the workpiece with moving the tool in parallel with the second axis on the plane (X-axis on the ZX plane).

(2) The figure need not show monotone increase or decrease in the direction of the first axis on the plane (Z-axis for the ZX plane) and it may have concaves (pockets). The figure must show monotone change in the direction of the second axis on the plane (X-axis for the ZX plane), however.

(3) When a position parallel to the second axis on the plane (X-axis for the ZX plane) is specified in a block in the program for the target figure, it is assumed to be at the bottom of a pocket.

(4) After all rough cutting terminates along the second axis on the plane (X-axis for the ZX plane), the tool temporarily returns to the start point. Then, rough cutting as finishing is performed.

5.36.3 Pattern Repeating (T Series)

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

Format

ZpXp plane

G73 W(Δk) U(Δi) R(d) ;

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

N (ns) ;

...

N (nf) ;

The move commands for the target figure from A to A’ to B are specified in the blocks with sequence numbers ns to nf.

YpZp plane

G73 V(Δk) W(Δi) R(d) ;

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

N (ns) ;

...

N (nf) ;

XpYp plane

G73 U(Δk) V(Δi) R(d) ;

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

N (ns) ;

...

N (nf) ;

Δi : Distance of escape in the direction of the second axis on the plane (X-axis for the ZX plane)
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5135, and the parameter is changed by the program command.

Δk : Distance of escape in the direction of the first axis on the plane (Z-axis for the ZX plane)
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5136, and the parameter is changed by the program command.

d : The number of division
This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by the parameter No. 5137, and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

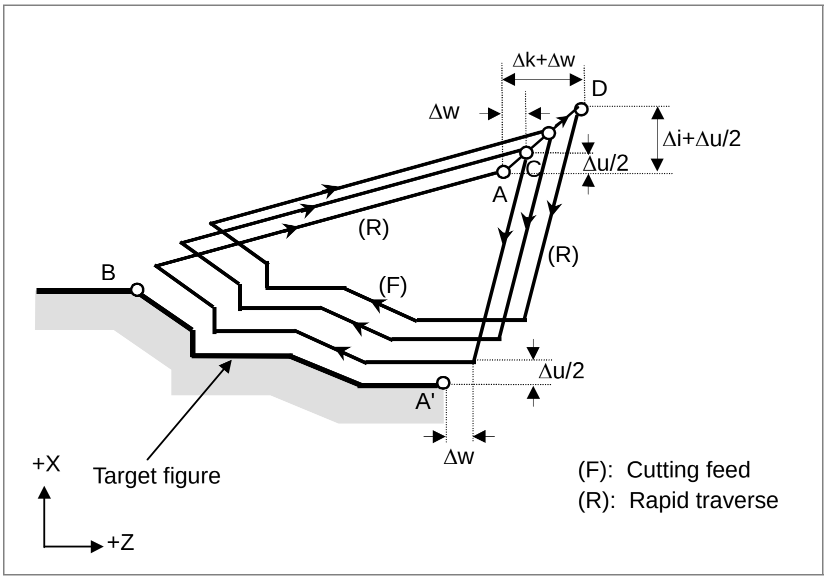
nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance of the finishing allowance in the direction of the second axis on the plane (X-axis for the ZX plane)

Δw : Distance of the finishing allowance in the direction of the first axis on the plane (Z-axis for the ZX plane)

f, s, t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73 block are effective.

Explanation



When a target figure passing through A, A', and B in this order is given by a program, rough cutting is performed the specified number of times, with the finishing allowance specified by Δu/2 and Δw left.

5.36.4 Finishing Cycle (T Series)

After rough cutting by G71, G72 or G73, the following command permits finishing.

Format

G70 P(ns) Q(nf) ;

ns : Sequence number of the first block for the program of target figure (finishing shape)

nf : Sequence number of the last block for the program of target figure (finishing shape)

Explanation

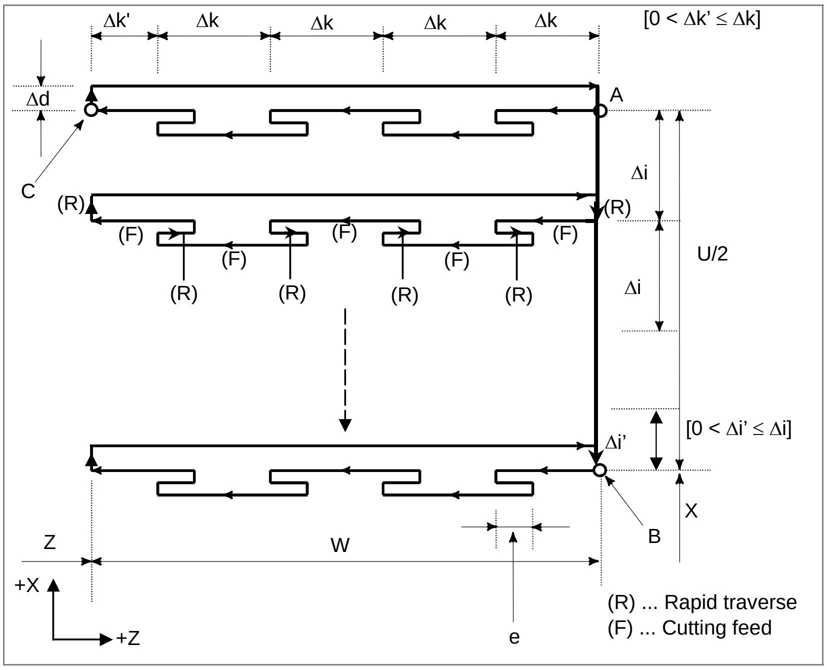
The blocks with sequence numbers ns to nf in the program for a target figure are executed for finishing. The F, S, T, M, and second auxiliary functions specified in the G71, G72, or G73 block are ignored and the F, S, T, M, and second auxiliary functions specified in the blocks with sequence numbers ns to nf are effective.

When cycle operation terminates, the tool is returned to the start point in rapid traverse and the next G70 cycle block is read.

5.36.5 End Face Peck Drilling Cycle (T Series)

The operation shown in the figure below is performed according to the command described below. This function enables chip breaking in outer diameter cutting. If the second axis on the plane (X-axis (U-axis) for the ZX plane) and address P are omitted, operation is performed only along the first axis on the plane (Z-axis for the ZX plane), that is, a peck drilling cycle is performed.

Format



G74R (e) ;

G74X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

e : Return amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5139, and the parameter is changed by the program command.

X_,Z_ : Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C

U_,W_: Travel distance along the second axis on the plane (U for the ZX plane) from point A to B
Travel distance along the first axis on the plane (W for the ZX plane) from point A to C
(When G code system A is used. In other cases, X_,Z_ is used for specification.)

Δi : Travel distance in the direction of the second axis on the plane (X-axis for the ZX plane)

Δk : Depth of cut in the direction of the first axis on the plane (Z-axis for the ZX plane)

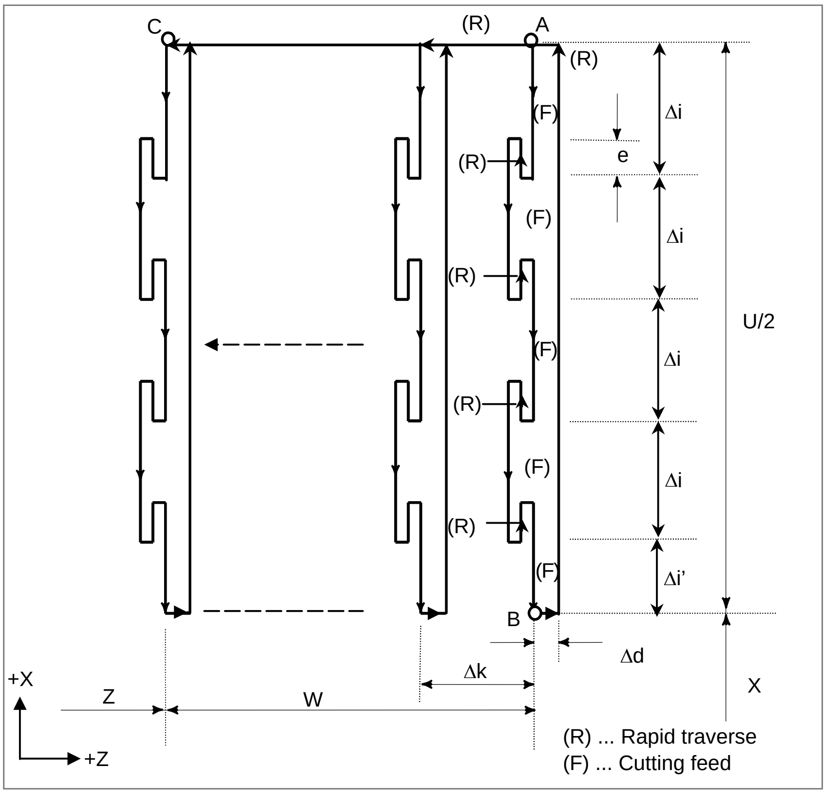
Δd : Relief amount of the tool at the cutting bottom

f : Feedrate

5.36.6 Outer Diameter / Internal Diameter Drilling Cycle (T Series)

The operation shown in the figure below is performed according to the command described below. This cycle is equivalent to G74 except that the second axis on the plane (X-axis for the ZX plane) changes places with the first axis on the plane (Z-axis for the ZX plane). This cycle enables chip breaking in end facing. It also enables grooving during outer diameter cutting and cutting off (when the Z-axis (W-axis) and Q are omitted for the first axis on the plane).

Format



G75R (e) ;

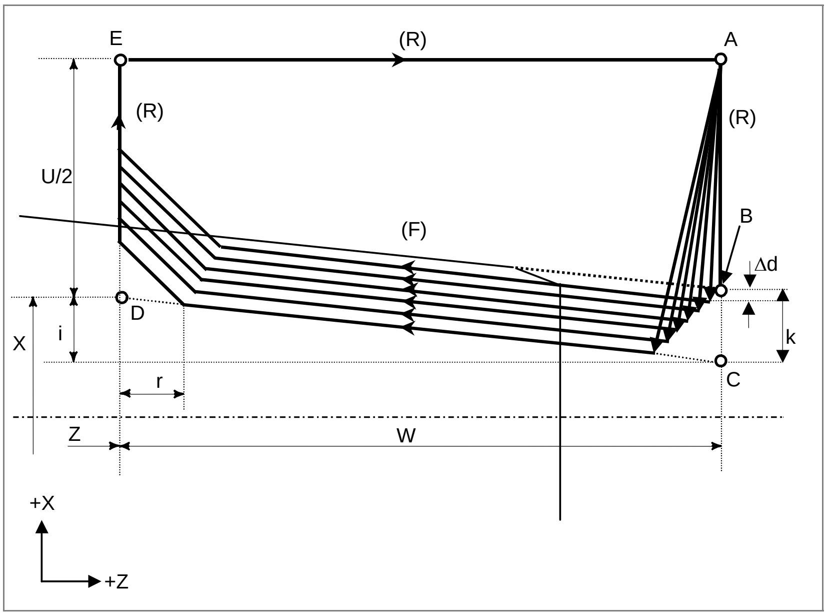
G75X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

e	:	Return amount This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5139, and the parameter is changed by the program command.
X_, Z_	:	Coordinate of the second axis on the plane (X-axis for the ZX plane) at point B and Coordinate of the first axis on the plane (Z-axis for the ZX plane) at point C
U_, W_	:	Travel distance along the second axis on the plane (U for the ZX plane) from point A to B Travel distance along the first axis on the plane (W for the ZX plane) from point A to C (When G code system A is used. In other cases, X_,Z_ is used for specification.)
Δi	:	Depth of cut in the direction of the second axis on the plane (X-axis for the ZX plane)
Δk	:	Travel distance in the direction of the first axis on the plane (Z-axis for the ZX plane)
Δd	:	Relief amount of the tool at the cutting bottom
f	:	Feedrate

5.36.7 Multiple Threading Cycle (T Series)

The threading cycle with a constant depth of cut using a one-side cutter as shown in the figure below is executed according the command described below.

Format



G76 P(m) (r) (a) Q(Δdmin) R(d) ;

G76 X(U)_ Z(W)_ R(i) P(k) Q(Δd) F (L) ;

m : Repetitive count in finishing (1 to 99)
This value can be specified by the parameter No. 5142, and the parameter is changed by the program command.

r : Chamfering amount (0 to 99)
When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2-digit number). This value can be specified by the parameter No. 5130, and the parameter is changed by the program command.

a : Angle of tool nose
One of six kinds of angle, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected, and specified by 2-digit number. This value can be specified by the parameter No. 5143, and the parameter is changed by the program command.

m, r, and a are specified by address P at the same time.

(Example) When m=2, r=1.2L, a=60°, specify as shown below (L is lead of thread).

```
P   02    12   60
│   │     └── a
│   └── r
└── m
```

Δdmin : Minimum cutting depth
When the cutting depth of one cycle operation becomes smaller than this limit, the cutting depth is clamped at this value. This value can be specified by parameter No. 5140, and the parameter is changed by the program command.

d : Finishing allowance
This value can be specified by parameter No. 5141, and the parameter is changed by the program command.

X_, Z_ : Coordinates of the cutting end point (point D in the figure) in the direction of the length

U_, W_ : Travel distance to the cutting end point (point D in the figure) in the direction of the length
(When G code system A is used. In other cases, X_,Z_ is used for specification.)

i : Taper amount
If i = 0, ordinary straight threading can be made.

k : Height of thread

Δd : Depth of cut in 1st cut

L : Lead of thread

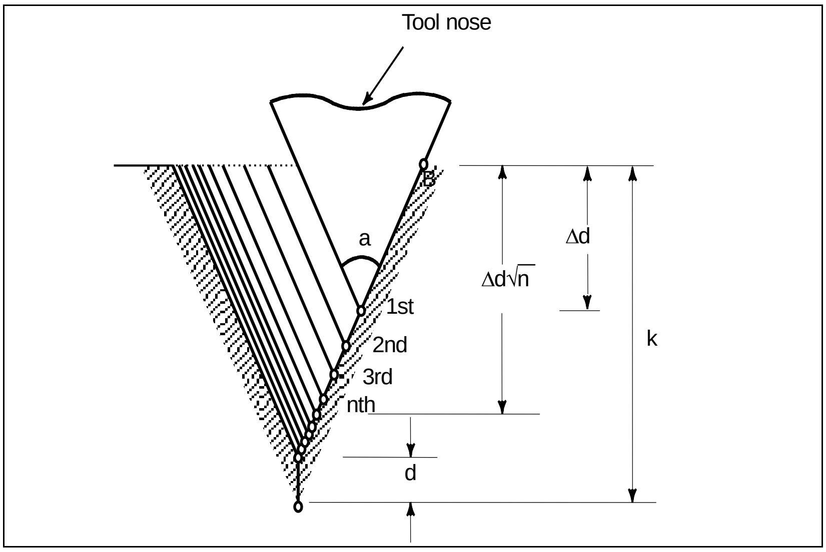


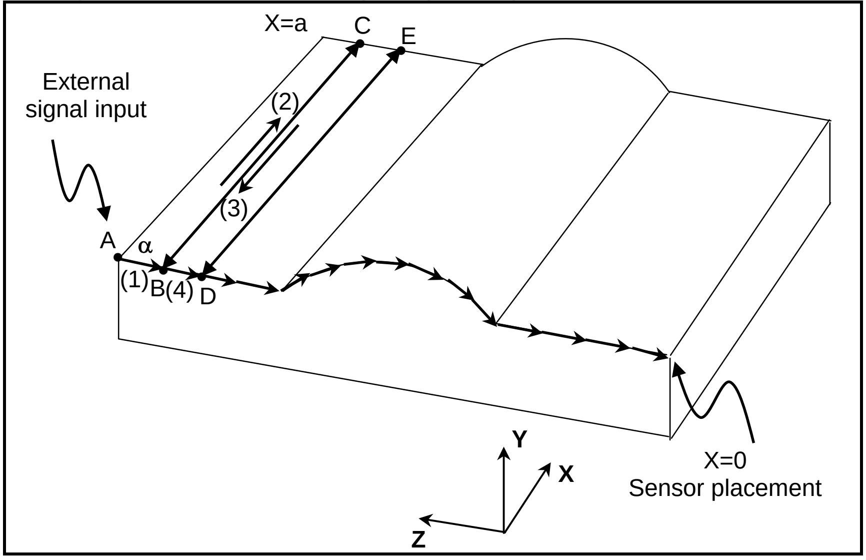
Fig. 5.36.7 (a) Detail of cutting

5.37 IN-FEED CONTROL (FOR GRINDING MACHINE) (M SERIES)

M

Overview

Each time an external signal is input at the table swing end point, a workpiece is cut by a constant depth of cut along a programmed figure on the specified Y-Z plane. This makes it possible to perform grinding and cutting in a timely manner and facilitating the grinding of a workpiece with a profile.



For example, it is possible to machine a workpiece with a profile programmed with linear interpolation, circular interpolation, and linear interpolation on the YZ plane, such as that shown in the figure above.

A sensor is placed at a X = 0 position so that the external signal is input when the sensor detects the grinding wheel. When the program is started at point A, the machine is first placed in the state in which it waits for the input of the external signal. Then, when the sensor detects the grinding wheel, the external signal is input, and the machine makes a cut by the constant amount α along the programmed profile on the specified YZ plane and moves to point B (operation (1)). The machine is then placed in the state in which it waits for the input of the external signal again, and performs a grinding operation along the X-axis. It grinds from point B to point C (operation (2)) and grinds back from point C to point B (operation (3)). When the machine returns to point B, the sensor detects the grinding wheel again, and the external signal is input, so that the machine makes a cut by the amount of α and moves to point D (operation (4)). At point D, the machine performs a grinding operation along the X-axis.

Afterwards, each time the external signal is input, the machine makes a cut by the amount of α along the profile program, so that the workpiece is machined to a profile such as that shown in the figure above.

5.38 CANNED GRINDING CYCLE (FOR GRINDING MACHINE)

Overview

With the canned grinding cycle, repetitive machining operations that are specific to grinding and are usually specified using several blocks can be specified using one block including a G function. So, a program can be created simply. At the same time, the size of a program can be reduced, and the memory can be used more efficiently. Four types of canned grinding cycles are available:

T

- Traverse grinding cycle

- Traverse direct constant-size grinding cycle
- Oscillation grinding cycle
- Oscillation direct constant-size grinding cycle

M

- Plunge grinding cycle
- Direct constant-dimension plunge grinding cycle
- Continuous-feed surface grinding cycle
- Intermittent-feed surface grinding cycle

5.39 CANNED CYCLE FOR DRILLING

M

The canned cycles for drilling are functions for performing machining operations such as boring, drilling, and tapping with a more simplified command.

The table below indicates the relationships between positioning planes and drilling axes.

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

Xp: X axis or an axis parallel to the X axis
Yp: Y axis or an axis parallel to the Y axis
Zp: Z axis or an axis parallel to the Z axis

The following canned cycles for drilling are available:

G code	Drilling (-Z direction)	Operation at the bottom of a hole	Retraction (+Z direction)	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Dwell → Spindle CW	Feed	Left-hand tapping cycle
G76	Feed	Spindle orientation	Rapid traverse	Fine boring cycle
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell → Spindle CCW	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G88	Feed	Dwell → Spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

NOTE

The 3 basic axes (X, Y, and Z) are needed.

T

When the canned cycles for drilling are used, a machining operation specified using several blocks can be specified using a single block including a G function, making programming much simpler.

The table below indicates the relationships between positioning axes and drilling axes.

G code	Positioning axis	Drilling axis
G83, G84, G85	X axis, C axis	Z axis

G code	Positioning axis	Drilling axis
G87, G88, G89	Z axis, C axis	X axis

The following canned cycles for drilling are available:

G code	Drilling axis	Drilling	Operation at the bottom of a hole	Retraction	Application
G80					Cancel
G83	Z axis	Intermittent feed Feed	Dwell	Rapid traverse	Front drilling cycle
G84	Z axis	Feed	Dwell → Spindle CCW	Feed	Front tapping cycle
G85	Z axis	Feed	Dwell	Feed	Front boring cycle
G87	X axis	Intermittent feed Feed	Dwell	Rapid traverse	Side drilling cycle
G88	X axis	Feed	Dwell → Spindle CCW	Feed	Side tapping cycle
G89	X axis	Feed	Dwell	Feed	Side boring cycle

NOTE

X,Z in 3 basic axes are needed.

5.40 CIRCULAR INTERPOLATION BY R PROGRAMMING

In the case of circular interpolation command, the distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.

In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed

G02R_ ; (The cutter does not move.)

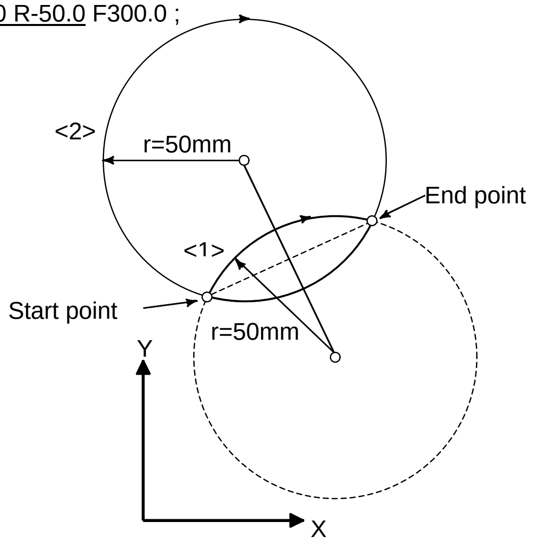
Example)

For arc <1> (less than 180°)

```
G91 G02 X60.0 Y55.0 R50.0 F300.0 ;
```

For arc <2> (greater than 180°)

```
G91 G02 X60.0 Y55.0 R-50.0 F300.0 ;
```



5.41 MIRROR IMAGE FOR DOUBLE TURRET (T SERIES)

T

By applying mirror image to the X-axis with a G code, a machining program for the opposite tool post can be created for symmetric cutting as if the program were created in the coordinate system on the same side.

Format

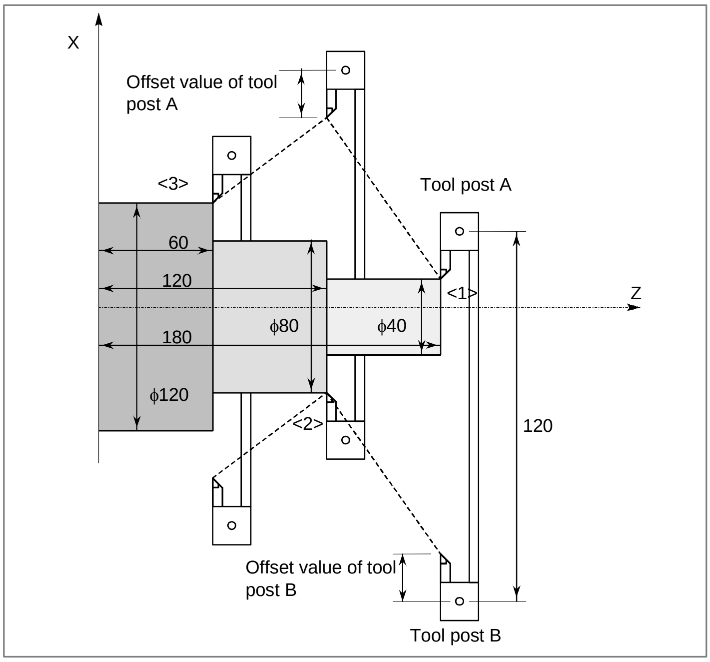
G68 : Double turret mirror image on

G69 : Mirror image cancel

Explanation

When G68 is designated, the coordinate system is shifted to the double turret side, and the X-axis sign is reversed from the programmed command to perform symmetrical cutting.
To use this function, set the distance between the two tool posts to a parameter.

Example



X40.0 Z180.0 T0101 ;	Position tool post A at <1>
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position tool post B at <2>
G69 ;	Shift the coordinate system by the distance B to A, and cancel mirror image.
X120.0 Z60.0 T0101 ;	Position tool post A at <3>

(*) In this example, a diameter value is specified for the X-axis.

5.42 AUTOMATIC CORNER OVERRIDE (M SERIES)

M

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.

5.43 SCALING (M SERIES)

M

A programmed figure can be magnified or reduced (scaling).
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.
Two types of scaling are available, one in which the same magnification rate is applied to each axis and the other in which different magnification rates are applied to different axes. Which type to use is determined by parameter setting.

Format

- Scaling up or down along all axes at the same rate of magnification

```
G51 IP_ P_ ;Scaling start
  :  ⎫         Scaling is effective.
  :  ⎭         (Scaling mode)
G50 ;          Scaling cancel
```

IP_ : Absolute programming for center coordinate value of scaling
P_ : Scaling magnification

- Scaling of each axis

```
G51 IP_ I_ J_ K_ ; Scaling start
  :  ⎫               Scaling is effective.
  :  ⎭               (Scaling mode)
G50 ;                Scaling cancel
```

IP_ : Absolute programming for center coordinate value of scaling
I_J_K_ : Scaling magnification for basic 3 axes (X, Y, and Z axes) respectively

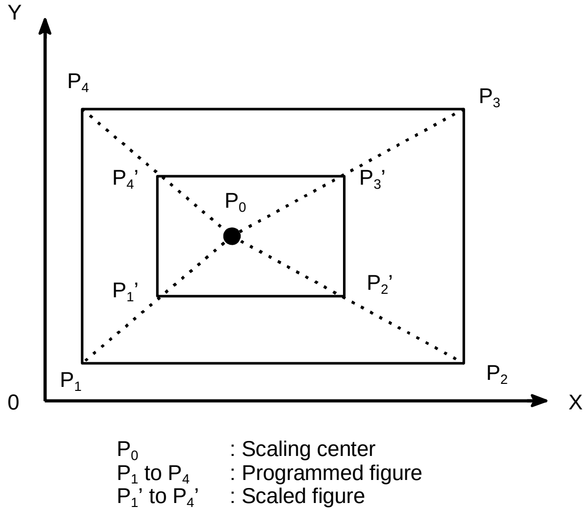
⚠ CAUTION

1 Specify G51 in a separate block.
2 After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

Explanation

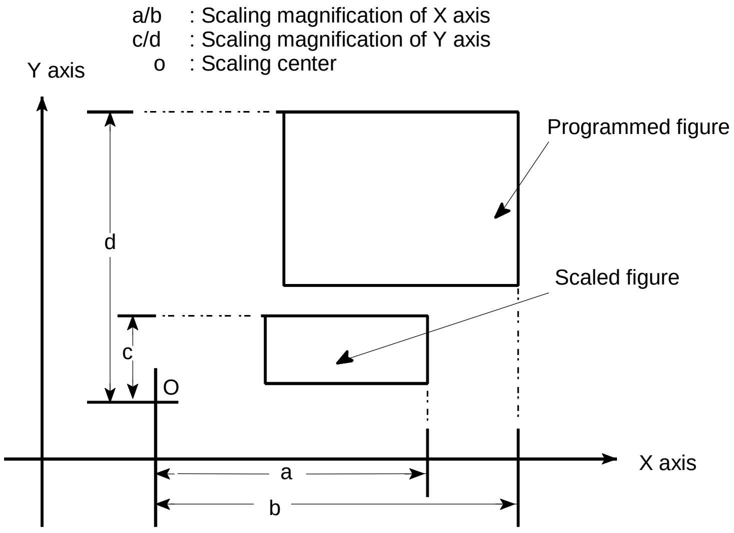
- Scaling along each axis at the same rate of magnification

A movement command in the scaling mode is scaled by the magnification specified by P_ with the point specified by X_Y_Z_ in the G51 block centered.



- Scaling of each axis

Scaling is performed by the magnification of each axis specified by I_J_K_ with the point specified by X_Y_Z_ centered. If I, J, and K are not specified, scaling is performed by a parameter-set magnification. Also when a negative magnification is specified, a mirror image is applied. The axis subject to the mirror image is the one that contains the scaling center.



- Scaling center

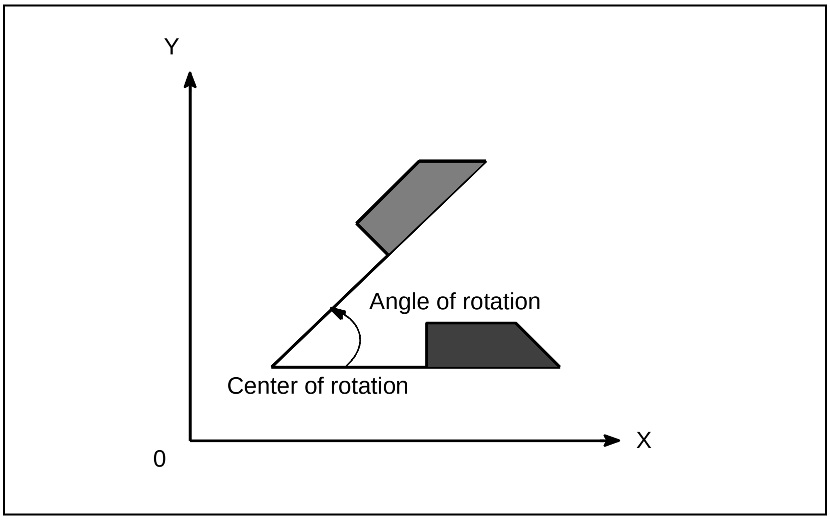
Even in incremental command (G91) mode, the scaling center coordinates IP_ specified in the G51 block are assumed those of an absolute position.

If the scaling center coordinates are omitted, the position assumed when G51 is specified is assumed the scaling center.

5.44 COORDINATE SYSTEM ROTATION (M SERIES)

M

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a sub program of the shape and calling it after rotation.



Format

$$\left\{\begin{matrix}\mathbf{G17}\\ \mathbf{G18}\\ \mathbf{G19}\end{matrix}\right\}\mathbf{G68}\ \alpha\_\ \beta\_\ \mathbf{R\_ ;}$$ Start rotation of a coordinate system.

:
: Coordinate system rotation mode (The coordinate system is rotated.)

G69 ; Coordinate system rotation cancel command

G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.

α, β : Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68

R : Angular displacement with a positive value indicates counter clockwise rotation. Parameter RIN (No. 5400#0) selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).
Least input increment : 0.001 deg
Valid data range : -360,000 to +360,000

Explanation

Once this function is specified, a subsequent command is rotated about the center specified by α_ β_ by the angle specified by R_.
The rotation plane (G17, G18, G19) selected when G68 was specified is used. G17, G18, or G19 may not be specified in the block including G68, but must not be specified in the coordinate system rotation mode.
If α_ β_ is omitted, the position where G68 was specified becomes the center of rotation.

5.45 PROGRAMMABLE MIRROR IMAGE (M SERIES)

M

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.

Format

- Setting of programmable mirror image

Following format is commanded and mirror image is commanded to each axis (as if mirror was set on the axis).

```
G51.1 IP_ ;
```

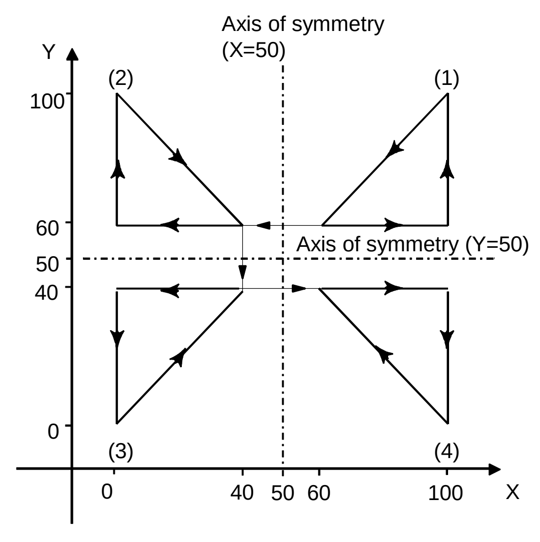
- Programmable mirror image cancel

Following format is commanded and the programmable mirror image is canceled.

```
G50.1 IP_ ;
```

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

5.46 SYNCHRONOUS, COMPOSITE, AND SUPERIMPOSED CONTROL BY PROGRAM COMMAND (T SERIES)

T

Synchronous control, composite control, and superimposed control can be started or canceled using a program command instead of a DI signal.

Synchronous control, composite control, and superimposed control based on a DI signal is also possible.

For the basic operations of synchronous control, composite control, and superimposed control, see Sections, "SYNCHRONOUS CONTROL AND COMPOSITE CONTROL", and "SUPERIMPOSED CONTROL" in the CONNECTION MANUAL (FUNCTION) (B-64303EN-1).

Format

G51.4 P_ Q_ (L_) ; **Start synchronous control**
(L_ can be omitted)

G50.4 Q_ ; **Cancel synchronous control**

P: Number to identify synchronous master axis
Q: Number to identify synchronous slave axis
L: Parking state command
1: Master parking (slave parking cancel)
2: Slave parking (master parking cancel)
0: No parking (parking cancel)
(When L is omitted, the specification of L0 is assumed.)

G51.5 P_ Q_ ; **Start composite control**

G50.5 P_ Q_ ; **Cancel composite control**

P: Number to identify composite axis 1
Q: Number to identify composite axis 2

G51.6 P_ Q_ ; **Start superimposed control**

G50.6 Q_ ; **Cancel superimposed control**

P: Number to identify superimposed master axis
Q: Number to identify superimposed slave axis

Identification numbers are unique values that set into parameter No.12600 in order to identify each axes.

G51.4/G50.4, G51.5/G50.5, and G51.6/G50.6 are one-shot G codes of group 00.

5.47 PROGRAM FORMAT FOR Series 10/11

By setting a setting parameter, the following functions programmed in the Series 10/11 program format can be executed by memory operation:

M

- Sub program call (M98)
- Canned cycle for drilling (G73, G74, G76, G80 to G89)

T

- Sub program call (M98)
- Canned cycle (G77, G78, G79)
  (G90, G92, G94 with G code system A)
- Multiple repetitive canned cycle (G71 to G76)
- Canned cycle for drilling (G80 to G85, G89)

NOTE

Address and specifiable value range
The ordinary program format restriction is imposed on the specifiable value range for basic addresses used. If a value beyond the allowable range is specified, an alarm is issued. Moreover, the usable addresses may be limited.

5.48 MACRO EXECUTOR

Some NC programs such as programs created using custom macros need not be modified once created. Others such as machining programs differ depending on the machining target.
This function can convert a custom macro program created by the machine tool builder to an executable macro program, load the executable macro program (P-CODE macro) into F-ROM (Flash ROM module), and execute it.
The function which converts a custom macro program to an executable macro program is called the macro compiler. The function which reads and executes a P-CODE macro is called the macro executor.

Features

- The execution speed is high because a custom macro program is loaded after converted to an executable so that the machining time can be reduced and the machining precision can be improved.
- Any custom macro is not destroyed because it is loaded into F-ROM so that reliability is improved.
- A program converted to execution format is not displayed on the program screen, so the know-how possessed by each machine tool builder can be protected.
- An execution format macro program is registered in the F-ROM, so the program storage space can be efficiently used.
- The user can call the execution format macro program with an easy call procedure without being conscious of the registered program.
- 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/puncher interface control, and PMC data read/write functions.

5.49 C LANGUAGE EXECUTOR

The C language executor function is used to customize screen display and implement a mechanism for user-specific operation as with the macro executor function. Instead of macro statements, application programs for display and operation can be created using the general C language.

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

To execute the C language executor and application programs, the size of the flash DRAM may need 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

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. 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.

5.50 CUSTOM SOFTWARE SIZE

The required size of custom software to be used with the macro executor and C language executor can be selected from the following:

512K / 2M / 4M (Unit : byte)

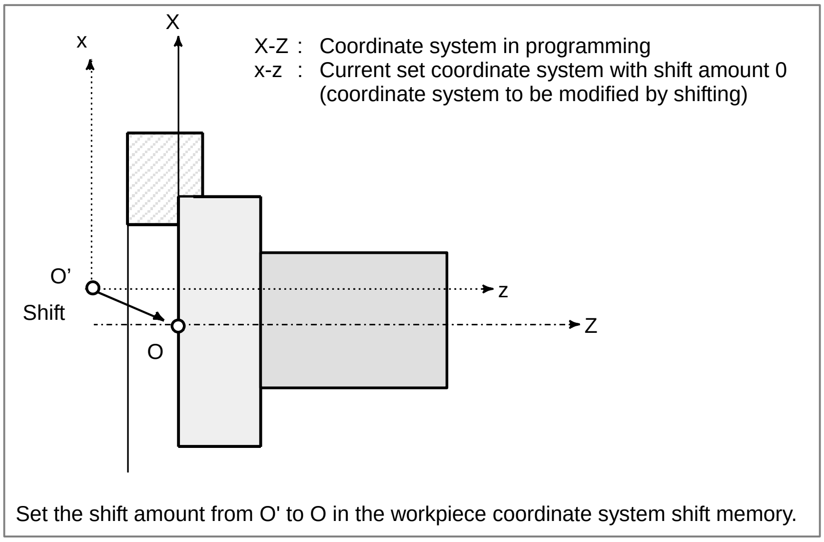
When a 2-path system is used, the total size required for both paths needs to be selected.

When using the C language executor, select 1M byte or more.

5.51 WORKPIECE COORDINATE SYSTEM SHIFT (T SERIES)

T

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



Set the shift amount from O' to O in the workpiece coordinate system shift memory.

5.52 SMALL-HOLE PECK DRILLING CYCLE (M SERIES)

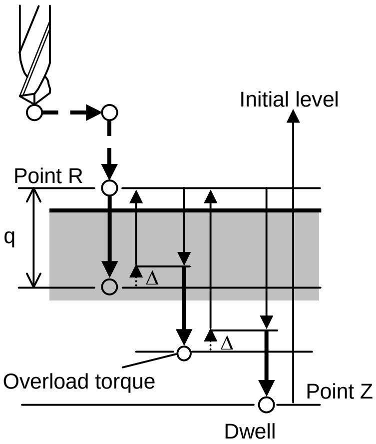
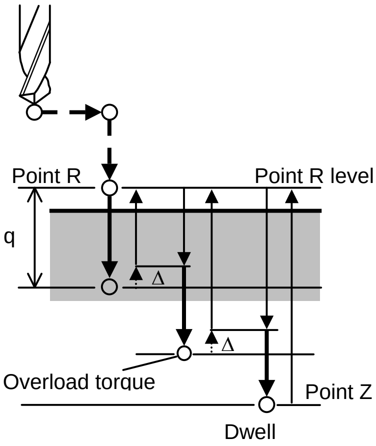
M

An arbor with the overload torque detection function is used to retract the tool when the overload torque detection signal (skip signal) is detected during drilling. Drilling is resumed after the spindle speed and cutting feedrate are changed. These steps are repeated in this peck drilling cycle.
The mode for the small–hole peck drilling cycle is selected when the M code in parameter is specified. The cycle can be started by specifying G83 in this mode. This mode is canceled when G80 is specified or when a reset occurs.

Format

G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;

X_ Y_ : Hole position data
Z_ : Distance from point R to the bottom of the hole
R_ : Distance from the initial level to point R
Q_ : Depth of each cut
F_ : Cutting feedrate
I_ : Forward or backward traveling speed (same format as F above)
K_ : Number of times the operation is repeated (if required)
P_ : Dwell time at the bottom of the hole
(If this is omitted, P0 is assumed as the default.)

G83 (G98)	G83 (G99)
	

Δ: Initial clearance when the tool is retracted to point R and the clearance from the bottom of the hole in the second or subsequent drilling
q: Depth of each cut
– – ▶ Path along which the tool travels at the rapid traverse rate
━━▶ Path along which the tool travels at the programmed cutting feedrate
──▶ (········▶) Path along which the tool travels at the forward or backward rate during the cycle specified with parameters

6 GUIDANCE FUNCTION

Chapter 6, "GUIDANCE FUNCTION", consists of the following sections:

6.1 MANUAL GUIDE *i*

MANUAL GUIDE *i* is an integrated operation and programming guidance function for assisting machine operators in all routine operations from machining program creation and actual machining.

6.1.1 Basic Functions

MANUAL GUIDE *i* features a single screen, on which all operations, from machining program creation to actual machining, are collected. On this screen, detail data can be displayed by opening windows as required.
The integrated operation screen has advanced ISO code program editing functions, such as copy & paste, word search, redo/undo, fixed-phrase insertion, M code listing, and guidance message display.

6.1.2 Milling Cycle

Many milling cycles can be used.
Hole machining, facing, contouring, pocketing, grooving, and emboss machining can be performed.
(The above machining operations except facing and emboss machining support polar coordinates and cylindrical coordinates.)
This function can be used to create a complicated machining program with a simple operation.

6.1.3 Turning Cycle (T Series)

T

Many turning cycles can be used.
Hole machining, turning, residual machining by turning, threading, and grooving can be performed.
This function can be used to create a complicated machining program with a simple operation.

6.1.4 Animation

How milling or turning is performed can easily be checked through tool path drawing or animation by a solid model.
Background machining simulation (a machine can be used to check machining programs while it is running on another machining program) is available.
Foreground tool path drawing (the path of a tool can be drawn while the tool is being used in machining) is also available.

6.1.5 Set-up Guidance Functions

These functions assist the setup of machining.

Tool compensation measurements, workpiece centering measurements, or workpiece in-machine measurements can easily be made from the menu.
In addition, manual measurement and automatic measurement are enabled.

6.2 MANUAL GUIDE *i* MULTI-PATH LATHE FUNCTIONS (T SERIES)

T

These functions assist the user in programming operations for and actual machining on 2-path laths.
They make it possible to use process editing based on a process directory, which enables the user to recognize at a glance which path (tool post) is to be used for a specific spindle by dividing an ISO code program into units of processes.
In machining simulation, how a workpiece is concurrently machined by each path (tool post) can be checked.

6.3 MANUAL GUIDE 0*i*

MANUAL GUIDE 0*i* is an easy-to-operate programming guidance function tailored only to programming.

6.3.1 Basic Functions

MANUAL GUIDE 0*i* provides the menu screens for operations required to create a machining program. On these menu screens, it is possible to select the operation screen for specifying a tool or spindle, guiding G codes or M codes, creating a machining cycle, or contour programming.

6.3.2 Milling Cycle (M series)

M

The fixed form machining cycles (drilling, facing, pocketing, and grooving) frequently used by a machining center or milling machine are provided.
This function can be used to create a machining cycle program with a simple operation.

6.3.3 Turning Cycle (T Series)

T

Machining cycles required for lathes are provided.
Turning drilling, turning, threading, and grooving can be performed.
This function can be used to create a machining cycle program with a simple operation.

6.3.4 Contour Programming Function

MANUAL GUIDE 0*i* can easily input a contour profile consisting of straight lines and arcs and convert it into NC command blocks (G01/G02/G03). In addition, advanced profile calculation functions including seven auxiliary calculation functions can be used.

6.4 TURN MATE *i* (T SERIES)

T

TURN MATE *i* is an integrated operation guidance function that achieves machining by general lathes without using an NC machining program.
If data is input according to guidance displayed on the screen, turning can be performed easily without using an NC program .

6.4.1 Basic Functions

TURN MATE *i* has all information required for operation on one operation screen. Only if information on the touch panel of TURN MATE *i* is pressed, the corresponding operation screen can be displayed.
On the operation screens, the workpiece coordinate system, spindle speed, feedrate, and other items can be set.

6.4.2 Turning Cycle

All cycle machining required for general lathes is covered. The machining profiles frequently used for a drawing are collected as fixed format patterns to save the effort of inputting profiles.
Turning, grooving, threading, thread repair, and turning drilling can be performed.
This function can be used to perform complicated machining with a simple input operation.
In addition, machining cycles can be converted into NC statements. Up to 20 machining cycles can be executed continuously.

7 AUXILIARY FUNCTION / SPINDLE SPEED FUNCTION

Chapter 7, "AUXILIARY FUNCTION / SPINDLE SPEED FUNCTION", consists of the following sections:

7.1 AUXILIARY FUNCTION

When a numeral is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. However, up to three M codes can be specified depending on the parameter setting.
The maximum number of digits can be specified by parameter setting.

The following M codes have special meanings.

- M02, M30 : End of program
- M00 : Program stop
- M01 : Optional stop

M98, M99, M198, the M code for calling a sub program (set by a parameter), the M code for calling a custom macro (set by a parameter), and the M codes for turning an interrupt macro on and off (M96 and M97, or parameter-set M codes) are processed internally by the CNC, so no signal is output for these M codes.

7.2 SECOND AUXILIARY FUNCTION

If a value with a maximum of eight digits is specified after address B, the code signal and strobe signal are transferred for calculation of the rotation axis. The code signal is retained until the next B code is specified.
Only one B code can be specified for each block. When the maximum number of digits and decimal point programming are specified by parameter.
In addition, the address used for specifying the second auxiliary function can be changed to an address other than address B (address A, C, U, V, or W) by setting parameter.

NOTE
In T series, addresses U, V, and W can be used only with G code system B or C.

7.3 AUXILIARY FUNCTION LOCK

The auxiliary function lock signal disables execution of a specified M, S, T, or B function. This means that the code signal and strobe signal are not output.
This function is used together with the machine lock function at the time of program checking.
Even in the auxiliary function lock state, M00, M01, M02, M30, M98, M99, and M198 (sub program call function) are executed. The M code for calling a sub program (set by a parameter) and the M code for calling a custom macro (set by a parameter) are also executed.

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

The communication of execution command signal (strobe signal) and completion signal in 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 description below uses the auxiliary functions (M code commands) as an example. The same description applies to the spindle speed function (S code), tool function (T code) and 2nd auxiliary function (B code).

(1) Assume that the following program is given:
Mxx;
Myy;

(2) In response to an M command, the CNC system sends out the code signals M00 to M31.
The CNC system inverts the logical level of the strobe signal MF, that is, from "0"to "1", or from "1" to "0".

(3) The CNC system inverts the strobe signal, then when the logical level of the auxiliary function completion signal MFIN becomes the same as the strobe signal, the CNC assumes the completion of PMC sequence.
With the usual method, the operation is assumed to be completed when a falling edge ("1" to "0") of the M/S/T/B completion signal FIN is received after a rising edge ("0" to "1") of the FIN signal is detected. This new system, on the other hand, assumes the operation has been completed upon detection of only one transition of the completion signal.
In addition , the conventional system uses only one completion signal (FIN) common to the M/S/T/B functions. This new system uses a different completion signal for each of the M, S, T, and B functions; the completion signals for the M, S, T, and B functions are MFIN, SFIN, TFIN, and BFIN, respectively.

The Fig. 7.4 (a) below shows the timing chart of these signals with the new system. For comparison, Fig. 7.4 (b) shows the timing chart of the conventional system.

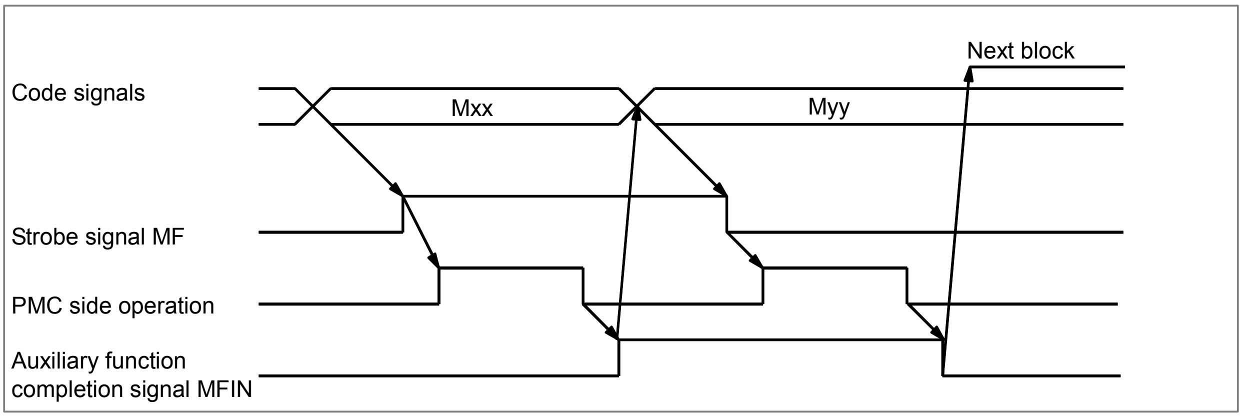


Fig. 7.4 (a) Timing chart of the high-speed system

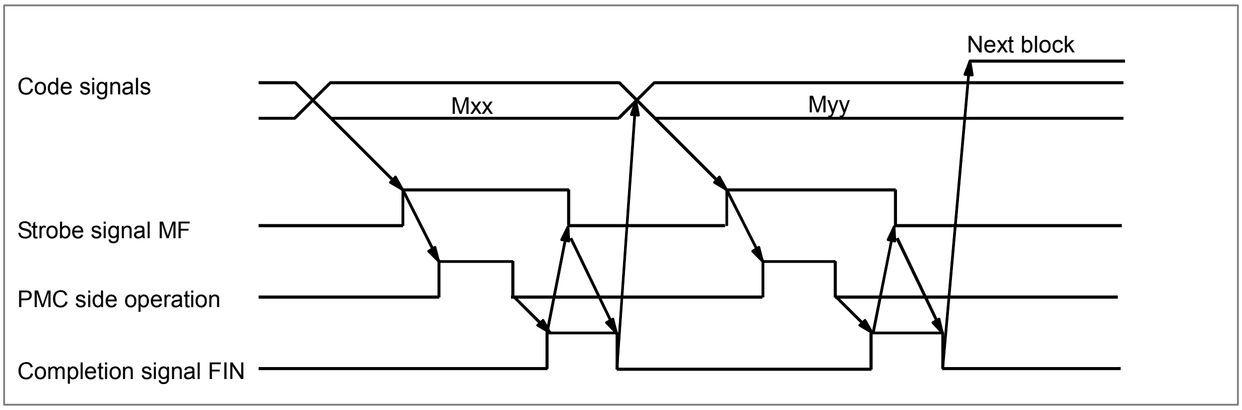


Fig. 7.4 (b) Timing chart of the conventional system

NOTE

1 Which system, the conventional system or high-speed system, is to be used for transferring the strobe signal and completion signal can be specified by parameter setting.

2 The conventional system used just one completion signal (FIN), which is common to the M, S, T, and B functions. In the high-speed system, a separate completion signal is provided for each of the M, S, T, and B functions.

7.5 WAITING FUNCTION (T SERIES)

T

An M code controls waiting between two paths at a midpoint of machining.

When an M code for waiting is specified in a block for one path during automatic operation, another path waits for the same M code to be specified before staring the execution of the next block.

A range of M codes used as M codes for waiting is to be set in the parameters beforehand. Waiting can be ignored using a signal.

⚠ **CAUTION**

1 Be sure to specify a waiting M code in an independent block.

2 Unlike other M codes, waiting M codes are not output to PMCs.

3 If you want to independently operate one path only, you need not delete that waiting M code. Using the waiting invalidation signal, you can invalidate the waiting M code specified in the machining program. For details, refer to the manual issued by the machine tool builder.

⚠ CAUTION

4 If using a waiting M code in 1 block multiple-M code command, be sure to specify it as the first M code.

7.6 MULTIPLE COMMAND OF AUXILIARY FUNCTION

Usually, one block has been able to contain only one M code. Parameter setting makes it possible to specify up to three M codes in one block at the same time.
Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

(Example)

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28 G91 X0 Y0 Z0 ; :	M40 M50 M60 ; G28 G91 X0 Y0 Z0 ; : : :

7.7 SPINDLE SPEED FUNCTION (S CODE OUTPUT)

A numeric value following address S specifies the spindle speed.
The specified value is output to the PMC in the form of 32-bit binary code. The code is maintained until another S is specified. In addition, the maximum number of input digits can be specified with a parameter.

7.8 SPINDLE SERIAL OUTPUT

The speed of the serial interface spindle is controlled.
The spindle speed is specified by up to five digits numeric value following 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 according to the spindle speed reached after constant surface speed control.

7.9 SPINDLE ANALOG OUTPUT

An analog spindle is subjected to spindle rotation control.
A 5-digit (maximum) numeric value following address S specifies the rotation speed of the spindle. A speed command voltage is output to the spindle motor according to the specified spindle speed. During constant surface speed control, a speed command is output according to the spindle speed that will be reached after constant surface speed control.

7.10 CONSTANT SURFACE SPEED CONTROL

When a surface speed (in m/min or feet/min) is specified with an S code (a numeric value following S), the spindle speed is controlled so that the surface speed can be kept constant with respect to the change in tool position.

With a machine having (or using) no position coder, feed per revolution is usually impossible. However, parameter setting makes it possible to specify feed per revolution by using spindle commands including a constant surface speed control command. (Without position coder constant surface speed control)

Format

G96 S_ ; Constant surface speed control mode

S_ : Surface speed (m/min or feet/min)

G97 S_ ; Constant surface speed control cancel mode

S_ : Spindle speed (min^{-1})

Constant surface speed control command G96 is a modal G code.
Once G96 is specified, the constant surface speed control mode (G96 mode is active) and assumes the specified S value is assumed to be the surface speed.

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

G96 Pα ;

P0 : Axis set in the parameter (No. 3770)
P1 : X axis, P2 : Y axis, P3 : Z axis, P4 : 4th axis
P5 : 5th axis

The maximum spindle speed can also be set by using the following command:

G92 S_ ;

The maximum spindle speed (min^{-1}) follows S.

T

G50 S_ ;

The maximum spindle speed (min^{-1}) follows S.

NOTE

G50 can be used with G code system A.

7.11 SPINDLE OVERRIDE

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

7.12 ACTUAL SPINDLE SPEED OUTPUT (T SERIES)

T

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

7.13 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. (Spindle orientation by a position coder)

When second spindle in each path is oriented, spindle orientation expansion is used.

7.14 SPINDLE OUTPUT SWITCHING FUNCTION

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.

When spindle output switching is performed for second spindle in each path, spindle output switching expansion is used.

7.15 SPINDLE SYNCHRONOUS CONTROL

In a machine having two or more spindles (such as a 2-path lathe), this function controls more than one spindle synchronously.
When a workpiece is switched between two spindles during spindle rotation, or when the spindle speeds of two spindles are accelerated or decelerated while the spindles are holding a workpiece, the spindles can be rotated at the same speed. (Spindle speed synchronization)
When a non-standard workpiece is switched, the rotations of the two spindles can be made in phase with each other (matching of angular displacement). (Phase synchronization)

7.16 SIMPLE SPINDLE SYNCHRONOUS CONTROL (M SERIES)

M

Spindles can be synchronized with each other for control.
Two spindles can be combined, one as a master spindle and the other as a slave spindle, so that Cs contouring control, rigid tapping, and normal spindle rotation control can also be used with the slave spindle under control by the master spindle.
This function can be used with serial spindles.

Unlike spindle synchronous control, simple spindle synchronous control does not guarantee spindle operation synchronization between a master spindle and slave spindle. Moreover, applicable spindle functions differ.

The major differences from spindle synchronous control are indicated below.

Function		Simple spindle synchronous control	Spindle synchronous control
Combination with other spindle functions	Spindle rotation control	∆(Same speed command is used.)	O(Synchronization / phase matching is possible.)
	Cs contour control	O	X
	Rigid tapping	O	X
Parking function		O	X

7.17 MULTI SPINDLE CONTROL

In addition to the first spindle in a path, the second spindle can be controlled by an S command from the CNC.
Spindle commands are specified using a single S command as conventionally done. A spindle is selected depending on the signal from the PMC or the address P command.

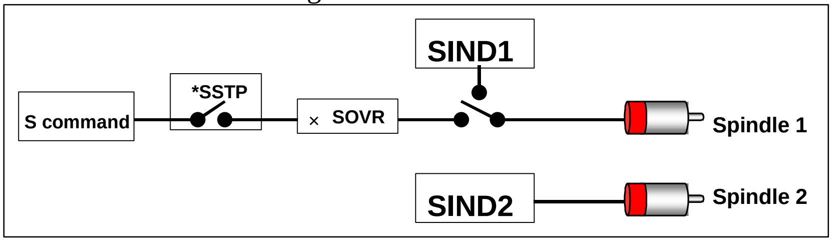
Gear change between two stages can be made for the second spindle in the same manner as for the first spindle.
Parameter setting makes it possible to set a maximum spindle speed for each spindle and clamp the speed of each spindle at the corresponding maximum spindle speed.
The position coder interfaces for the second spindle can be selected and added.
The first and second position coders are selected by signals from the PMC.

There are three types of multi spindle control methods: a method that can use the SIND (spindle motor speed control by the PMC) function only for the first spindle (TYPE-A), a method that can use the SIND function for the two spindles independently (TYPE-B), and a method that is equivalent to TYPE-B and can use address P to select the spindle to be controlled by the S command.
These methods can be selected by parameter setting.

An S command is sent as a speed command to each spindle selected, using a spindle selection signals. Each spindle rotates at the specified speed. If a spindle is not sent a spindle selection signal, it continues to rotate at its previous speed. This allows the spindles to rotate at different speeds at the same time.
Signals are provided to stop the corresponding spindles, so unnecessary spindles can be kept stopped.

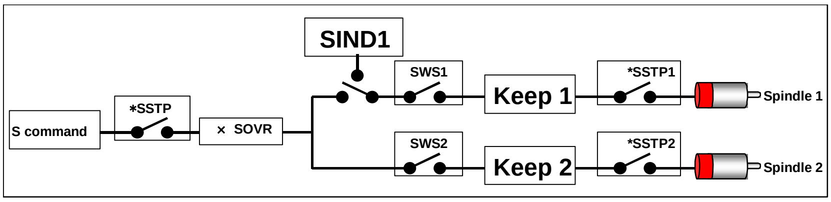
- When multi spindle control is disabled

Only the first spindle can be controlled using the S command even if there are more than one spindle.



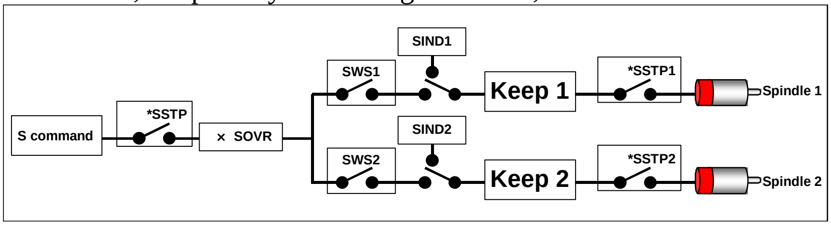
- Multi-spindle control (TYPE-A)

When the first spindle is selected with the SWS1 signal, the SIND signal is used to determine whether the spindle analog voltage is controlled by the PMC or CNC; then signals R01I to R12I are used to set that spindle’s analog voltage. These signals do not affect the second spindle.
The PMC-based polarity (rotation direction) control signals SGN and SSIN will function for any spindle selected by SWS1 and SWS2.



- Multi-spindle control (TYPE-B)

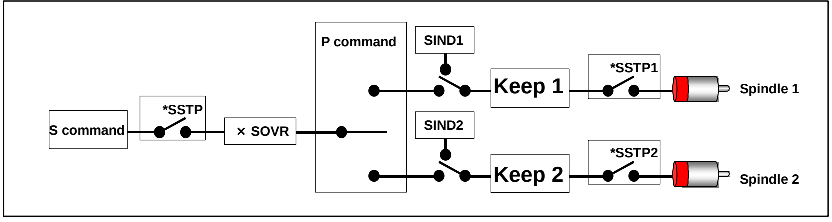
Each spindle has its own SIND, SSIN and SGN signals.
Each of these signals functions regardless of selection state of the spindle selection signals (SWS1 and SWS2).
When either the spindle selection signals (SWS1 and SWS2) or the SIND signal for the first, second, third, or fourth spindle is set to "1", the polarity control signals SSIN, SGN will function.



- Multi spindle control: when a spindle is selected by address P

This control method is basically the same as TYPE-B. The first and second spindles each have their own SIND, SSIN, and SGN signals. A spindle is selected by the P command instead of the spindle selection signals (SWS1 and SWS2). The relationship between the P code and the selected spindle is set in parameter.

Polarity (rotation direction) control signals SSIN and SGN for each spindle are valid only for the spindle selected by the P command or for the spindle of which SIND signal is 1.



In case of 2-path control, the P code becomes valid in all paths. For example, suppose that the P code to select the first spindle of path 2 is set to 21. When the following is specified in path 1:

S1000 P21;

Then, 1000 is specified for the first spindle in path 2. Therefore, the same P code cannot be used even in different paths.

7.18 SPINDLE POSITIONING (T SERIES)

T

The workpiece mounted on the spindle can be positioned at a certain angle by moving the spindle attached to the spindle motor by the certain angle. This function is called the spindle positioning function. During turning, use of this function allows machining such as drilling at any position on the circumference of the workpiece.

The spindle position is detected by the position coder 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).

- Movement command

There are the following two method of positioning the spindle in the spindle positioning mode.

1. Arbitrary angle positioning by an axis address
   G00 C_ ;(C_ is an arbitrary axis address.)
   The spindle is positioned to the commanded position by rapid traverse. Absolute (G90) and incremental (G91) command. Decimal point input is also available.
2. Semi-fixed angle positioning by a specific M code (set by a parameter)
   The positioning angle is specified by an M code.
   Set the M codes that can be specified and their corresponding positioning angles in advance using a parameter.
   Also set the rotation direction using a parameter.

- Increment system

Least input increment : 0.001deg (IS-B)

Detection unit : $\frac{360 \times N}{4096}$ deg

N represents the gear ratio of the spindle to position coder. (1, 2, 4, 8, . . .)

7.19 RIGID TAPPING

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

P=F/S
where P : Pitch of screw of tapper (mm, inch)
F : Feed rate of drilling axis (mm/min, inch/min)
S : Spindle speed (min^{-1})

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

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

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

7.20 SPINDLE SPEED FLUCTUATION DETECTION (T SERIES)

T

With this function, an overheat alarm is raised and the spindle speed fluctuation detection alarm signal SPAL is issued when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.

7.21 Cs CONTOUR CONTROL AXIS COORDINATE ESTABLISHMENT

Shifting a serial spindle from spindle rotation control to Cs contour control results in its current position being lost.
This function is intended to establish the current position without making a reference position return. This is done by setting the Cs axis coordinate establishment request signal to ‘1’.

NOTE

Once the power has been turned on, this function remains enabled until the power is turned off after the return of the Cs contour axis to the reference position.

7.22 SPINDLE CONTROL WITH SERVO MOTOR

The function for spindle control with servo motor allows a servo motor to be used for executing spindle speed commands and spindle functions such as rigid tapping.

(1) Spindle control with servo motor
Velocity control can be performed by using a speed command (S command) with a servo motor set as the tool rotation axis. No reference position return is necessary to switch between rotation and positioning commands.

(2) Spindle indexing
With the spindle indexing function, a stop position can be programmed to stop a rotating axis at the specified position. There are two types of spindle indexing. The first type allows the next-block command to be executed before spindle indexing is finished. The second type allows the next block to be executed only after spindle indexing is completed.
With the first type, it is possible to issue commands to axes other than the axis for which a spindle indexing command has been issued, before the next command is issued to the axis. Before the next command is issued to the axis for which a spindle indexing command has been issued, whether spindle indexing is completed or not can be checked by programming or by using a signal. Using this function can reduce the wait time. In addition, an axis can be stopped at a specified point by issuing a spindle indexing command to the axis when the spindle is rotating.

(3) Axis movement
When bit 0 (PCE) of parameter No. 11006 is 1, if axis movement (G00/G01) is specified for a servo motor spindle in the same way as in a normal controlled axis, position control can be performed.

(4) Rigid tapping with servo motor
Rigid tapping can be carried out by regarding a servo motor spindle as a rotation axis.

(5) Threading, feed per rotation feed, and constant surface speed control
Threading, feed per rotation feed, and constant surface speed control can be performed using a servo motor spindle as a spindle.

(6) Spindle output control with PMC
The rotation speed and polarity can be controlled by PMC.

Notes

(1) This function is optional.

(2) For this function, it is necessary to enable spindle serial output.
For a servo motor spindle, set the type of a spindle motor to a serial spindle (bit 0 (A/S) pf parameter No.3716 = 1). If there is a normal spindle in the path where the servo motor spindle is present, enable multi-spindle control.

(3) How a servo motor spindle is treated depends on whether position control such as axis movement (G00/G01) is enabled. When bit 0 (PCE) of parameter No. 11006 is 1, position control is enabled. In this case, the servo motor spindle is assumed to be a controlled axis and the number of controlled axes is incremented.
When bit 0 (PCE) of parameter No. 11006 is 0, position control is disabled. In this case, the servo motor spindle is not assumed to be a controlled axis and the number of controlled axes is not incremented.

Maximum number of controlled axes

Maximum number of controlled axes in an entire system	Position control	Controlled axis type	Maximum number of servo motors
1-path system 4 axes	Disabled	Servo axis ＋ Servo motor spindle	5
		Only servo axis	4
	Enabled	Servo axis ＋ Servo motor spindle	4
		Only servo axis	4
2-path system 8 axes	Disabled	Servo axis ＋ Servo motor spindle	9
		Only servo axis	8
	Enabled	Servo axis ＋ Servo motor spindle	8
		Only servo axis	8

(4) A servo motor spindle used by this function is counted as a controlled spindle.

Maximum number of spindles

Maximum number of spindles in an entire system	Spindle type	Maximum number of spindles
1-path system 2 axes	Spindle ＋ Servo motor spindle	2
	Only spindle	2
2-path system 3 axes	Spindle ＋ Servo motor spindle	3
	Only spindle	3

(5) Only one servo motor spindle can be set in a system. If more than one servo motor spindle is set, alarm PW1110 is issued and the emergency stop state is entered.

(6) When performing rigid tapping with a servo motor, enable rigid tapping (bit 3 (NRG) of parameter No. 8135 is 0).

(7) When bit 0 (PCE) of parameter No. 11006 is 1, position control is enabled and a servo motor spindle can be set as a PMC controlled axis.
If a servo motor spindle is set as a PMC controlled axis when position control is disabled (bit 0 (PCE) of parameter No. 11006 is 0), alarm DS2003 is issued.

Spindle motors and supported functions

Spindle-related function	Conventional spindle control	Spindle control with servo motor
Feed per revolution	○	○
Threading	○	○
Polygon machining	○	×(*1)
Spindle speed fluctuation detection	○	×
Spindle synchronous control	○	×
Spindle simple synchronous control	○	×
Polygon machining with two spindles	○	×
Spindle orientation Multi-point orientation Spindle output switching	○	×
Inter-path spindle control	○	○
Constant surface speed control	○	○
Multi-spindle control	○	○
Rigid tapping	○	○
Spindle output control with PMC	○	○
Actual spindle speed output	○	○
Spindle indexing	×	○

NOTE
Servo motor can be used as spindle. (*1)

7.23 SPINDLE REVOLUTION NUMBER HISTORY FUNCTION

This function counts the spindle revolution number and displays the total revolution number as diagnostic information.
Total revolution number data as diagnostic information can be read with the PMC window function, and can be used for spindle unit life management on the PMC ladder.

7.24 POSITION CODER SELECTION BY ADDRESS P

When the spindle is selected by address P in multi-spindle control (MPP(No.3703#3)=1), position coder feedback is automatically selected by address P for feed per revolution, thread cutting, and so on.

8 TOOL FUNCTION / TOOL COMPENSATION FUNCTION

Chapter 8, "TOOL FUNCTION / TOOL COMPENSATION FUNCTION", consists of the following sections:

8.1 TOOL FUNCTION

A tool can be selected by specifying a tool number of up to eight digits numeric value following address T.

When a T code is specified, the code signal corresponding to the tool number and the strobe signal are sent to the machine to select a tool. This code signal is maintained until another T code is specified.

Just one T code can be specified per block. The maximum number of digits can be specified by parameter setting.

T

In T series, part of the numeric following the T code is used also as a tool offset number specifying a value such as a tool offset value.

An offset number may be specified by the lowest one digit, lowest two digits, or lowest three digits of the numeric following the T code as shown below.

The number of the digits of the offset number can be selected by parameter setting.

- **Specifying an offset number with the lowest one digit**

T○○○○○○○ ☆
↑ ↑
Tool number Offset number

- **Specifying an offset number with the lowest two digits**

T○○○○○○ ☆☆
↑ ↑
Tool number Offset number

- **Specifying an offset number with the lowest three digits**

T○○○○○ ☆☆☆
↑ ↑
Tool number Offset number

Specifying an offset number means selecting the corresponding offset value and also means starting tool offset.
When 0 is specified as the offset number, tool offset is canceled.

8.2 TOOL OFFSET PAIRS

M

The number of tool offset pairs used in the entire system can be selected from the list shown below.

Table 8.2 (a) Tool offset pairs

Bit 5 (NDO) of parameter No.8136	Number of tool offset pairs
1	32
0	400

T

- 0*i* Mate -TD, 0*i* -TD with 1-path system

The number of tool offset pairs used in the entire system can be selected from the list shown below.

Table 8.2 (b) Tool offset pairs

		Bit 5 (NDO) of parameter No.8136	
		0	1
Tool offset pairs 99-pairs option	disable	64	32
	enable	99	99

- 0*i* -TD with 2-path system

The number of tool offset pairs used in the entire system can be selected from the list shown below.
For a 2-path system, the number of tool offset pairs used in each path can be set by the corresponding parameter. The number of tool offset pairs used in the entire system must not exceed the number selected in the following table.

Table 8.2 (b) Tool offset pairs

		Bit 5 (NDO) of parameter No.8136	
		0	1
Tool offset pairs 200-pairs option	disable	128	32
	enable	200	200

NOTE

The maximum number of tool offset pairs that can be used in a path is 200.
In this case, the number of tool offset pairs that can be used in the other path is 0.

8.3 TOOL COMPENSATION MEMORY

M

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

Explanation

- Tool compensation memory A

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

Setting example of tool compensation memory A

Offset number	Compensation value (geometry + wear)	Common to D and H codes
001	10.000	For D code
002	20.000	For D code
003	100.000	For H code
:	:	:

- Tool compensation memory C

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

Setting example of tool compensation memory C

Offset number	D code		H code	
	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation
001	10.000	0.100	100.000	0.100
002	20.000	0.200	200.000	0.300
:	:	:	:	:

T

- Tool offset

When the tool offset function is used, data that can be set in tool compensation memory is the compensation values for the X- and Z-axes.

- Tool nose radius compensation

When the tool nose radius compensation function is provided, compensation value R and imaginary tool nose direction T are added to the tool compensation memory setting items.

- Y-axis offset

When Y-axis offset is provided, the Y-axis offset setting screen is added. Data that can be set in Y-axis offset memory is the compensation value for the Y-axis.

- Interference check for each path

A contact inhibition area for the interference check for each path can be set for each tool number. The data to be set for the contact inhibition area is X, Z, I, and K coordinates. These data items are not tool compensation values, but the number of sets varies depending on the number of tool compensation pairs.

- Tool geometry / wear compensation

The tool compensation memory configuration differs depending on whether the tool geometry and wear compensation functions are provided or not. The following data items in tool compensation memory are affected:

1. X- and Z-axis compensation values in tool offset
2. Compensation value R when tool nose radius compensation is provided
3. Y-axis compensation value when Y-axis offset is provided

When the tool geometry and wear compensation functions are not provided, there is no distinction between geometry compensation memory and wear compensation memory. Therefore, a sum of the geometry compensation value and wear compensation value is set in compensation memory.

When the tool geometry and wear compensation functions are provided, geometry compensation memory and wear compensation memory are prepared separately. Therefore, geometry compensation values and wear compensation values can be set separately.

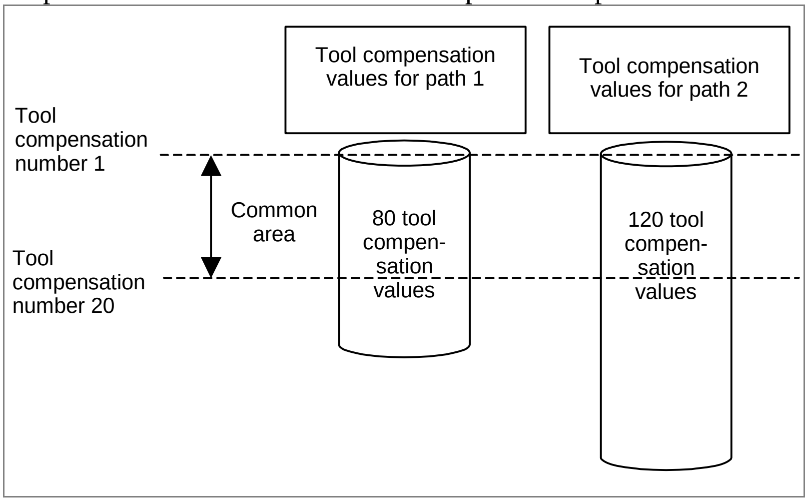
8.4 COMMON COMPENSATION MEMORY BETWEEN EACH PATH (T SERIES)

T

In 2-path systems, part or all of tool compensation memory provided for each path can be shared among multiple paths by parameter setting.

Example)

Sharing tool compensation numbers 1 to 20 between path 1 and path 2

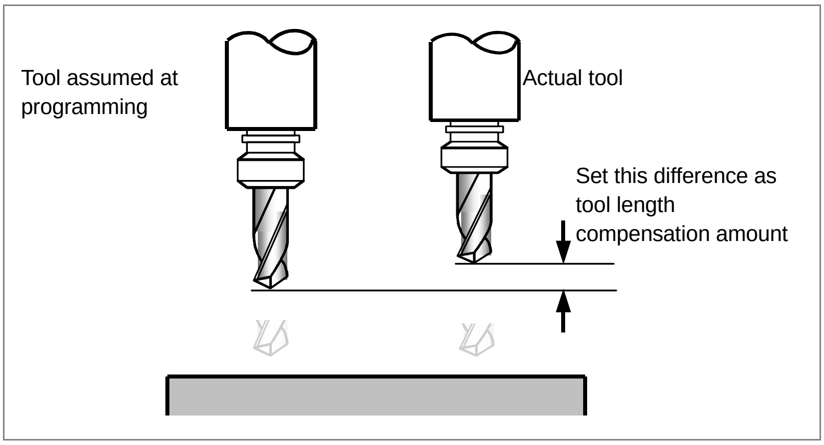


8.5 TOOL LENGTH COMPENSATION (M SERIES)

M

When the difference between the tool length assumed at the time of programming and the tool length of the tool actually used for machining is set in offset memory, the difference in tool length can be corrected without modifying the program.

G43 and G44 specify the offset direction, and a number following the tool length compensation specification address (H code) specifies the tool length compensation amount set in the offset memory.



Format

Type	Format	Description
Tool length compensation A	**G43 Z_ H_ ;** **G44 Z_ H_ ;**	G43 : Positive offset G44 : Negative offset G17 : XY plane selection G18 : ZX plane selection G19 : YZ plane selection α : Address of a specified axis H : Address for specifying the tool length compensation value X, Y, Z : Offset move command
Tool length compensation B	**G17 G43 Z_ H_ ;** **G17 G44 Z_ H_ ;** **G18 G43 Y_ H_ ;** **G18 G44 Y_ H_ ;** **G19 G43 X_ H_ ;** **G19 G44 X_ H_ ;**	
Tool length compensation C	**G43 α_H_ ;** **G44 α_H_ ;**	
Tool length compensation cancel	**G49 ; or H0 ;**	

Explanation

One of the following three methods is available, depending on the type of axis that can be subject to tool length compensation:

- **Tool length compensation A**
  Compensates for the difference in tool length along the basic Z-axis.
- **Tool length compensation B**
  Compensates for the difference in tool length in the direction normal to a selected plane.
- **Tool length compensation C**
  Compensates for the difference in tool length along a specified axis.

One of the above methods can be selected by parameter setting.

- Direction of the offset

When G43 is specified, the tool length compensation value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

When the specification of an axis is omitted, a movement is made by the tool length compensation value.

G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

- Specification of the tool length compensation value

The tool length compensation value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

A tool length compensation value is to be set in the offset memory corresponding to an offset number.

- Specification of tool length compensation along two or more axes

Tool length compensation B can be executed along two or more axes when the axes are specified in two or more blocks.

By setting the parameter, cutter compensation C can also can be executed along two or more axes when the axes are specified in two or more blocks.

- Tool length compensation cancel

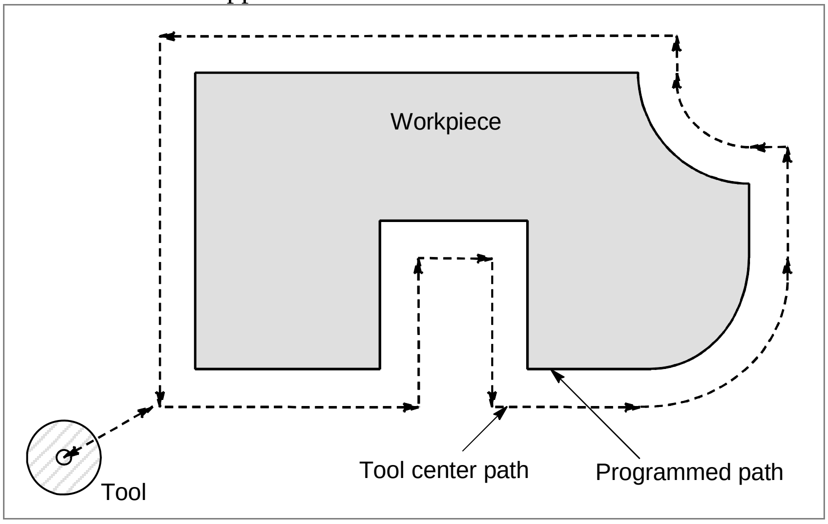
To cancel tool length compensation, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

8.6 TOOL OFFSET

M

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value.

The tool offset function can also be applied to an additional axis.



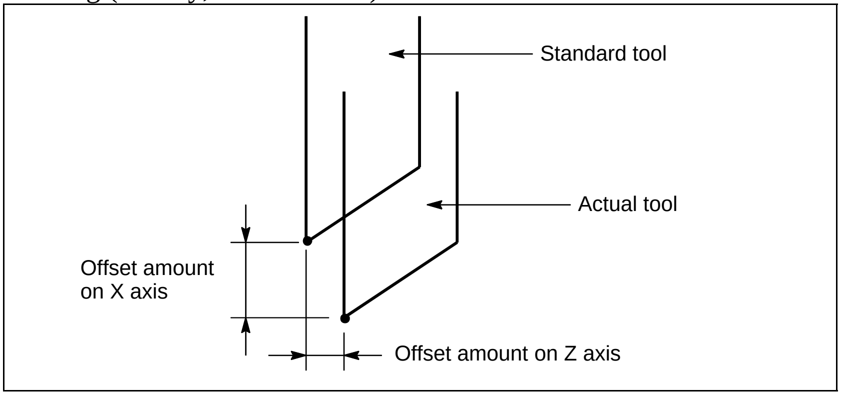
Format

G45 IP_ D_ ; Increase the travel distance by the tool offset value
G46 IP_ D_ ; Decrease the travel distance by the tool offset value
G47 IP_ D_ ; Increase the travel distance by twice the tool offset value
G48 IP_ D_ ; Decrease the travel distance by twice the tool offset value

G45 to 48 : One-shot G code for increasing or decreasing the travel distance
IP_ : Command for moving the tool
D Code for specifying the tool offset value

T

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).



8.7 Y-AXIS OFFSET (T SERIES)

T

When the Y axis, one of the basic three axes, is used with a T series, this function performs Y axis offset. If the tool geometry / wear offset are enabled, both tool geometry offset and tool wear offset are effective to Y axis offset.

If the parameter is set to the number of an axis that uses Y-axis offset, Y-axis offset can be enabled for an arbitrary axis other than the Y-axis of the basic three axes.

NOTE

1. When Y-axis offset is used, the Y-axis must always be a linear axis.
2. Y-axis offset does not allow direct input of tool offset values.

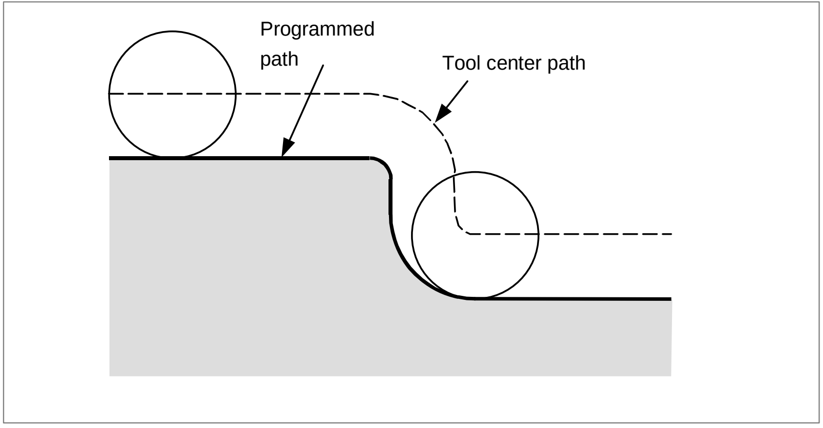
8.8 CUTTER OR TOOL NOSE RADIUS COMPENSATION

M

- Cutter compensation

Use of this function can offset a programmed tool path by the tool radius set in the CNC when machining is performed.

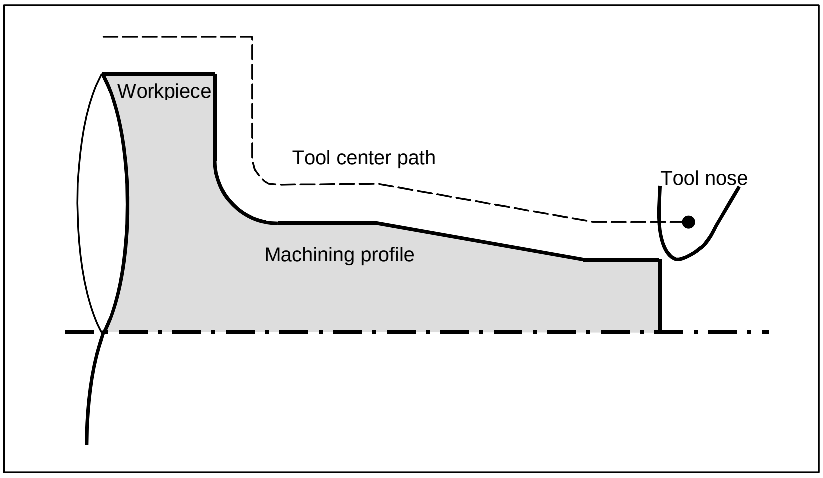
When the radius of the tool to be used for machining is measured and set as the offset value in the CNC, the tool moves along the offset path to cut a programmed profile. Therefore, even when the tool diameter changes, you must only change the offset value and need not modify the program.



T

- Tool nose radius compensation

Use of this function can offset a programmed tool path by the tool nose radius set in the CNC when machining is performed. The radius of the tool nose to be used for actual machining is measured and set as the offset value in the CNC, the tool moves along the offset path to cut the programmed profile. Therefore, even when the tool nose radius changes, you must only change the offset value and need not modify the program.



To obtain the actual offset tool path, the CNC internally calculates intersections of a straight line and a straight line, an arc and an arc, and a straight line and an arc automatically. The programmer only has to program a machining profile, therefore the programming can be done very easily.

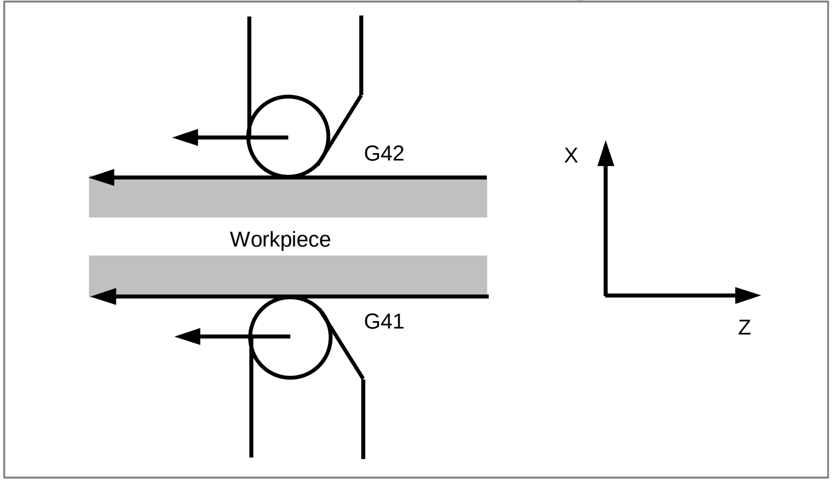
Format

G40 : Cutter or tool nose radius compensation cancel
G41 : Cutter or tool nose radius compensation : left
G42 : Cutter or tool nose radius compensation : right

Explanation

- Cutter or tool nose radius compensation cancel

G41 and G42 are commands to place the system in cutter or tool nose radius compensation mode. G41 causes offset on the left side of the tool advance direction, and G42 causes offset on the right side of the tool advance direction. G40 cancels cutter or tool nose radius compensation.



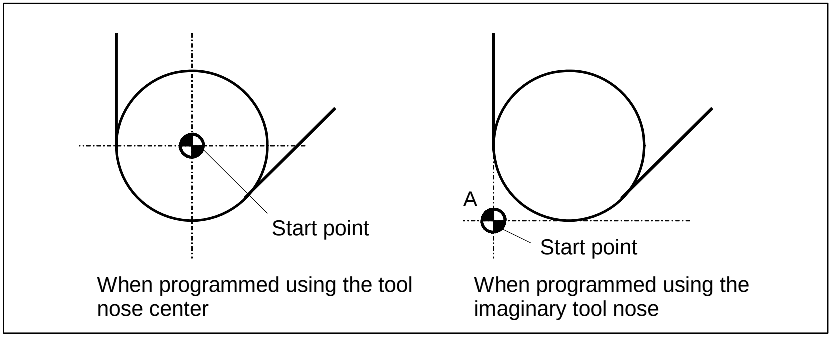
T

- Imaginary tool nose (tool nose radius compensation)

The tool nose at position A in following figure does not actually exist.

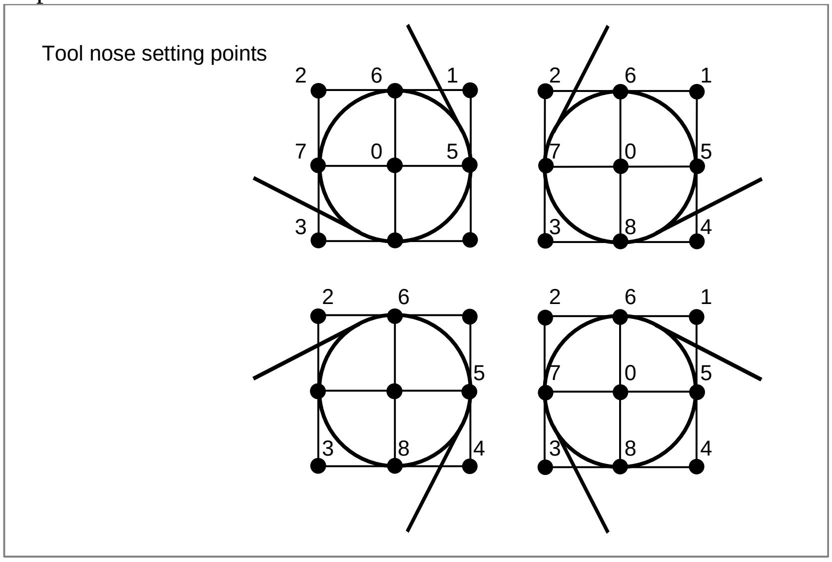
The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start point than the imaginary tool nose.

Also when imaginary tool nose is used, the tool nose radius need not be considered in programming. Programming can be performed in the same way as when a tool having no tool nose radius is used.



The position relationship when the tool is set to the start point is shown in following figure.

The start point or reference position in the tool nose can be set in offset memory in the same way as the tool nose radius compensation value.



- Specifying a compensation value and imaginary tool nose position

A compensation value and imaginary tool nose position can be set in tool compensation memory.

By specifying an offset number in part of a T code, the tool nose radius compensation values and imaginary tool nose position in the tool compensation memory area corresponding to the offset number are selected.

- Plane selection (G17, G18, G19)

Cutter or tool nose radius compensation can be applied not only to the XY, ZX, and YZ planes but also axes parallel to X, Y, and Z.

The target plane for cutter or tool nose radius compensation is selected by specifying G17, G18, or G19.

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

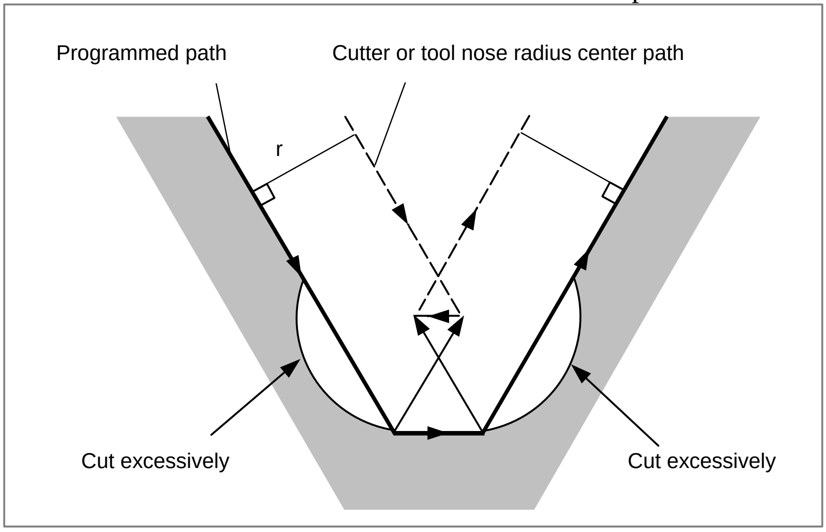
An additional axis can be set as an axis parallel to the X-, Y-, or Z-axis by parameter setting.

- Corner circular interpolation (G39)

By specifying G39 in offset mode during cutter or tool nose radius compensation, corner circular interpolation can be performed. The radius of the corner circular interpolation equals the compensation value.

- Interference check

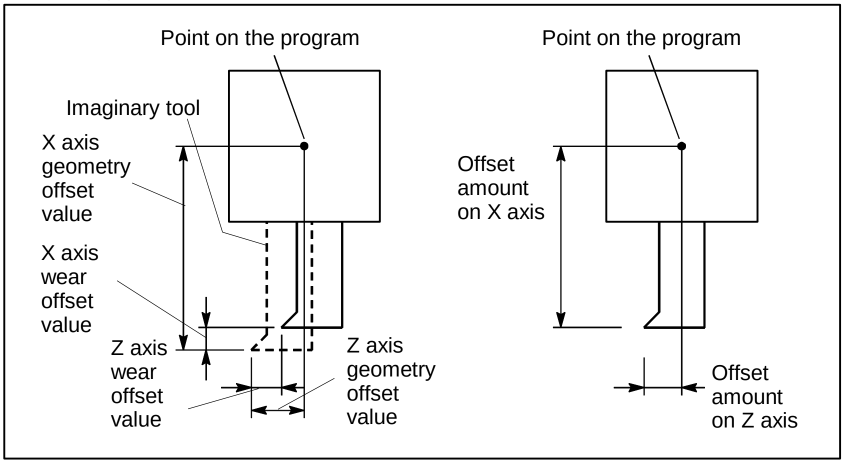
A state in which the tool cuts a workpiece excessively is called "interference". A check is made to see whether interference occurs as a result of cutter or tool nose radius compensation.



8.9 TOOL GEOMETRY OFFSET AND TOOL WEAR OFFSET (T SERIES)

T

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear. The tool geometry offset value and tool wear offset value can be set individually. When these values are not distinguished from each other, the total of the values is set as the tool offset value.



If tool geometry offset and tool wear offset are distinguished from each other

If tool geometry offset and tool wear offset are not distinguished from each other

8.10 TOOL OFFSET VALUE COUNTER INPUT (T SERIES)

T

A value displayed as a relative coordinate position can be set as an offset value in tool compensation memory by using a soft key.

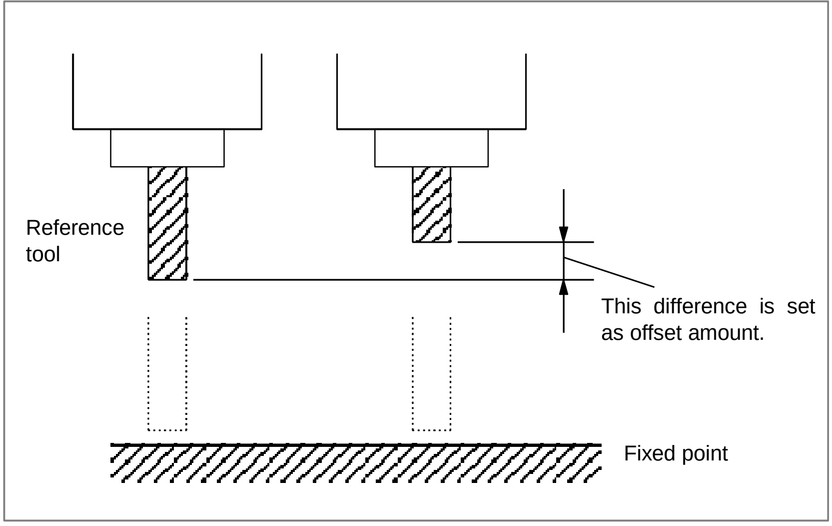
First, display the offset value screen. This screen also displays the relative coordinate position. Next, select a reference tool, and manually set it at the fixed point on the machine. Reset the relative coordinates displayed on the screen to zero. Then, manually set the tool to be measured at the same fixed point on the machine. At this time, the relative coordinate position shows the difference between the reference tool and the tool to be measured, and this value can be set as the offset value.

8.11 TOOL LENGTH MEASUREMENT (M SERIES)

M

The value displayed as a relative position can be set in the offset memory as an offset value by a soft key.

Switch to the offset value display screen on the screen. Relative positions are also displayed on this screen. Then select the reference tool and set it at the fixed point on the machine by manual operation. Reset the displayed relative position to zero. Set the tool for measurement at the same fixed point on the machine by manual operation. 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.



8.12 AUTOMATIC TOOL LENGTH MEASUREMENT (M SERIES) / AUTOMATIC TOOL OFFSET (T SERIES)

8.12.1 Automatic Tool Length Measurement (M Series)

M

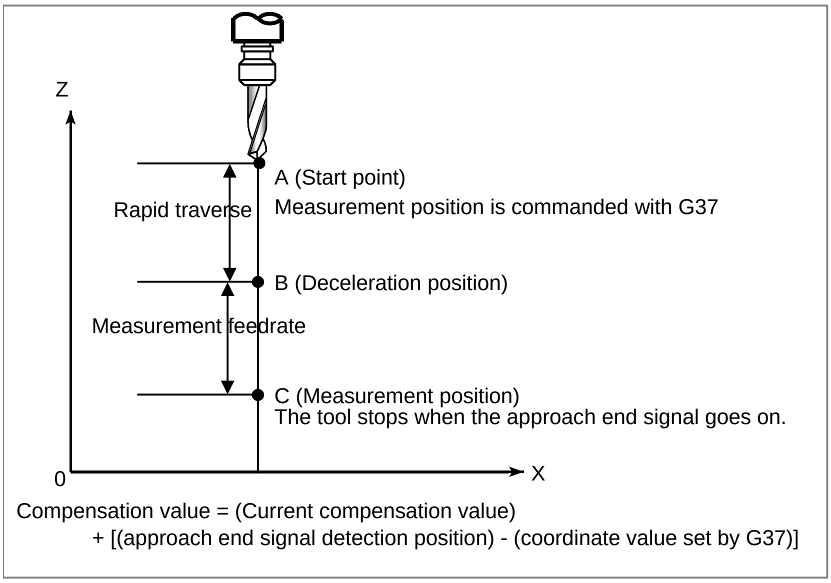
Difference between the coordinate value of tool when tool end has reached the measuring position and coordinate value of the measuring position is automatically measured, calculated, and added to the currently set tool offset amount by CNC system. The machine must be equipped with measuring devices,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

Format

G92 IP_ ;	Sets the workpiece coordinate system. (It can be set with G54 to G59.)
Hxx ;	Specifies an offset number for tool length compensation.
G90 G37 IP_ ;	Absolute programming

G37 is valid only in the block in which it is specified.
IP_ indicates the X-, Y-, Z-, or fourth axis.

Explanation



When G37 is specified, the tool moves from the start point to deceleration position by rapid traverse, then from the deceleration position the tool decelerates to a parameter-set measurement federate and moves until the approach end signal is issued from the measuring device. That is, the tool movement stops when the tool tip has reached the measurement position.

The difference between the coordinate value of the tool when it reaches the measurement position and the coordinate value specified in G37 is added to the tool length compensation value currently used.

8.12.2 Automatic Tool Offset (T Series)

T

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 touch sensor, so that a signal is sent when the tool end has reached the measuring position.

Format

G36 X_ ; or G37 Z_ ;

G36 and G37 are effective only in the block in which these commands are specified.
X_, Z_ : Absolute programming

Explanation

Execution of above command moves the tool at the rapid traverse rate toward the measurement position, lowers the feedrate halfway, then continues to move it until the approach end signal from the measuring instrument is issued.

When the tool tip reaches the measurement position, the measuring instrument outputs the measurement position reach signal to the CNC which stops the tool.

The difference between the coordinate value of the tool when the measurement position is reached and the coordinate value specified by G36 or G37 is added to the currently used tool offset value.

8.13 DIRECT INPUT OF TOOL OFFSET VALUE MEASURED (T SERIES) / DIRECT INPUT OF COORDINATE SYSTEM SHIFT (M SERIES)

T

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 workpiece 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 workpiece 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.

When offset and shift values are set, a movement can be made along the axis for which the values are set to retract the tool by using the position record signal.

8.14 DIRECT INPUT OF TOOL OFFSET VALUE MEASURED B (T SERIES)

T

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 workpiece coordinate system shift amount automatically.

8.15 TOOL LIFE MANAGEMENT

Tools are classified into several groups, and a tool life (use count or use duration) is specified for each group in advance. Each time a tool is used, its life is counted, and when the tool life expires, a new tool that is sequenced next within the same group is selected automatically. With this function, the tool life can be managed while machining is being performed continuously. Data for tool life management consists of tool group numbers, tool life values, tool numbers, and codes for specifying a tool offset value. These data items are registered in the CNC.

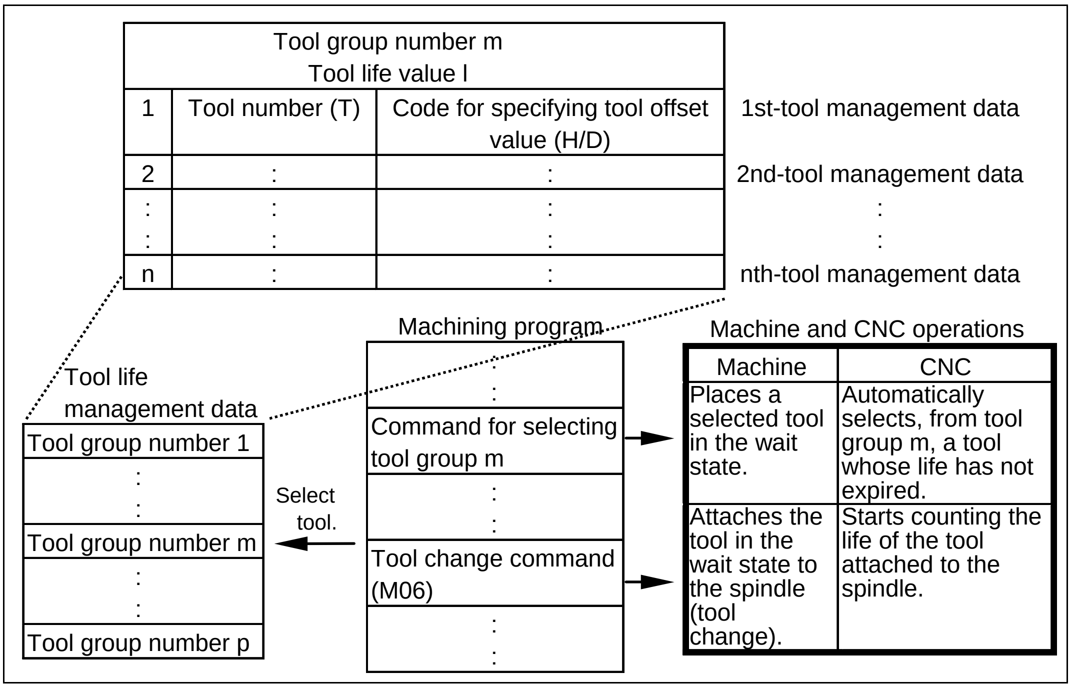


Fig. 8.15 (a) Tool selection from machining program

M

A group is selected by a T code, and tool life counting is started by the M06 command.

T

A group is selected, tool compensation is specified, and counting of tool life is started by only a T code (turret method).

- Maximum number of tool life management groups and 2-path system

The maximum number of tool life management groups that can be used in the entire CNC system is 256. For each path, set a maximum number of groups to be used in parameter No. 6813.
The maximum number of groups must be a multiple of the minimum number of groups (eight groups). A setting of 0 indicates 128 groups.

9 ACCURACY COMPENSASION FUNCTION

Chapter 9, "ACCURACY COMPENSASION FUNCTION", consists of the following sections:

9.1 BACKLASH COMPENSATION

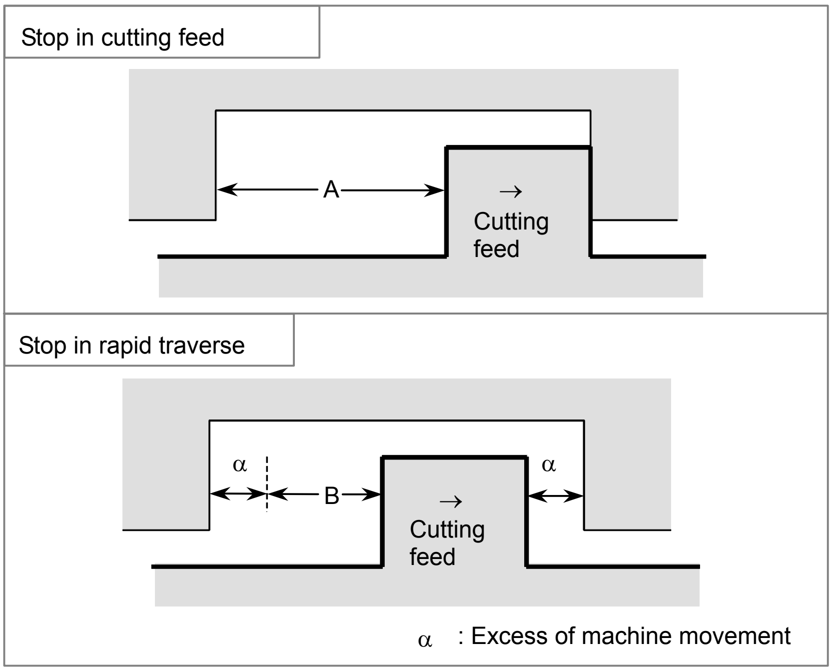
Function for compensating for lost motion on the machine. Set a compensation value in parameter, in detection units from 0 to ±9999 pulses for each axis.

9.2 BACKLASH COMPENSATION FOR EACH RAPID TRAVERSE AND CUTTING FEED

Since different backlash compensation values can be used for cutting feed and rapid traverse, the machining precision is improved.
Let the measured backlash amount at the time of cutting feed be A and the measured backlash amount at rapid traverse be B. Then, the output backlash compensation value varies according to the change in feed (cutting feed and rapid traverse) and the change in movement direction as listed in the following table:

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

- $\alpha = (A-B) / 2$
- Signs (+ and –) indicate directions.



9.3 SMOOTH BACKLASH COMPENSATION

In ordinary backlash compensation, backlash compensation pulses are all output at a position where the axis movement direction is inverted.

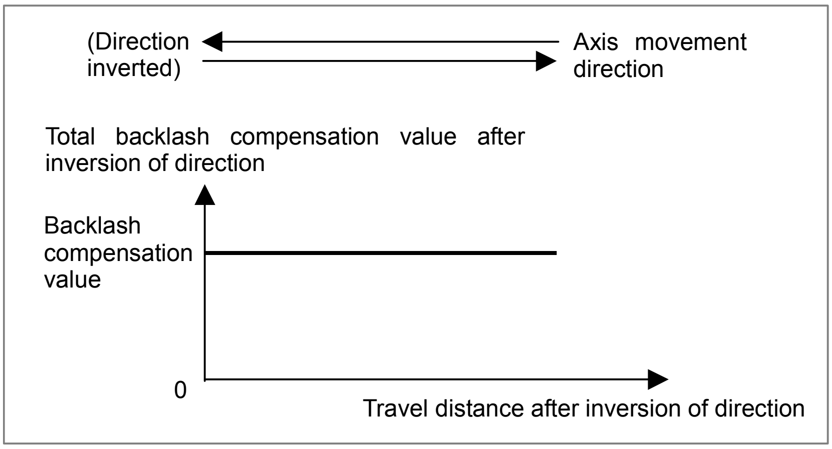
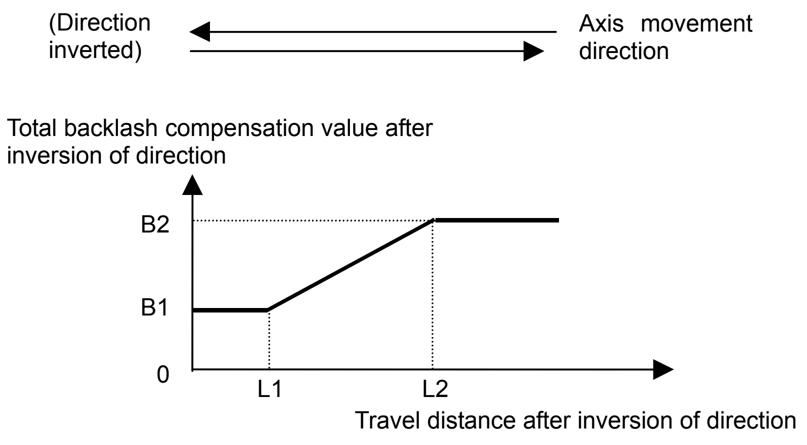


Fig. 9.3 (a) Ordinary backlash compensation

In smooth backlash compensation, backlash compensation pulses are output according to the distance from the position where the axis movement direction is inverted, so fine backlash compensation can be performed according to the machine characteristics.



B2 : Backlash compensation value
B1 : Compensation value in the 1st smooth backlash compensation stage
L1 : Distance for starting the 2nd smooth backlash compensation stage
L2 : Distance for ending the 2nd smooth backlash compensation stage

Fig. 9.3 (b) Smooth backlash compensation

9.4 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).

The total number of compensation points is as follows:

Total compensation points = 1024points

Distribution to each axis can be determined arbitrarily (by parameter setting). The number of compensation pulses at each position is as follows:

Compensation pulses = (-7 to +7) × (magnification)

The compensation pulse unit is the same as the detection unit.
The magnification is as follows:

Magnification: 0 to 100, parameter setting (for each axis)

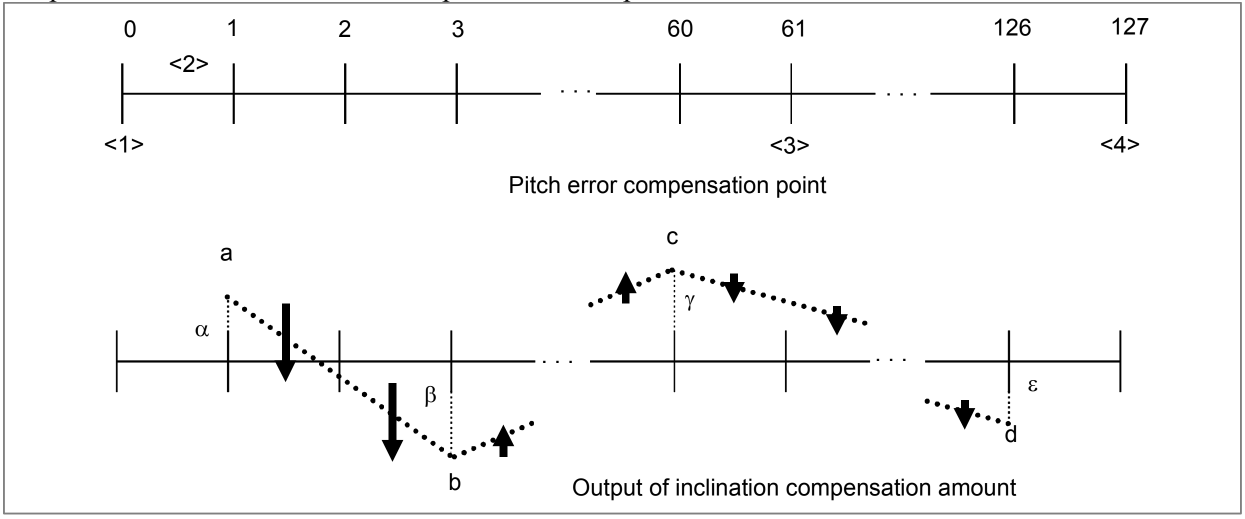
9.5 BI-DIRECTIONAL PITCH ERROR COMPENSATION

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

9.6 INCLINATION COMPENSATION

By compensating for those errors in tools such as feed screws that depend on the position of the machine system in detection units, machining precision can be improved and mechanical life can be prolonged. Compensation is performed along an approximate straight line formed with a parameter-specified compensation point and a compensation amount related to it.

Three approximate straight lines are formed with four parameter-specified compensation points and compensation amounts related to the respective compensation points. Inclination compensation is carried out along these approximate straight lines at pitch error compensation intervals. The inclination compensation amount is added to the pitch error compensation amount.



To perform inclination compensation, stored pitch error compensation must be set for the axis subject to compensation.

(1) Number of the most distant pitch error compensation point on the - side
(2) Pitch error compensation point interval
(3) Number of the pitch error compensation point of the reference position
(4) Number of the most distant pitch error compensation point on the + side

Inclination compensation parameters must be set.

a,b,c,d : Compensation point numbers.
α,β,γ,ε : Compensation amounts at compensation points a, b, c, and d

In above figure, a, b, c, and d are 1, 3, 60, and 126, respectively.

Unlike stored pitch error compensation, whose amount is set up for an individual compensation point, an amount of inclination compensation is calculated for individual compensation points by setting up four typical points and compensation amounts for them.

Example)

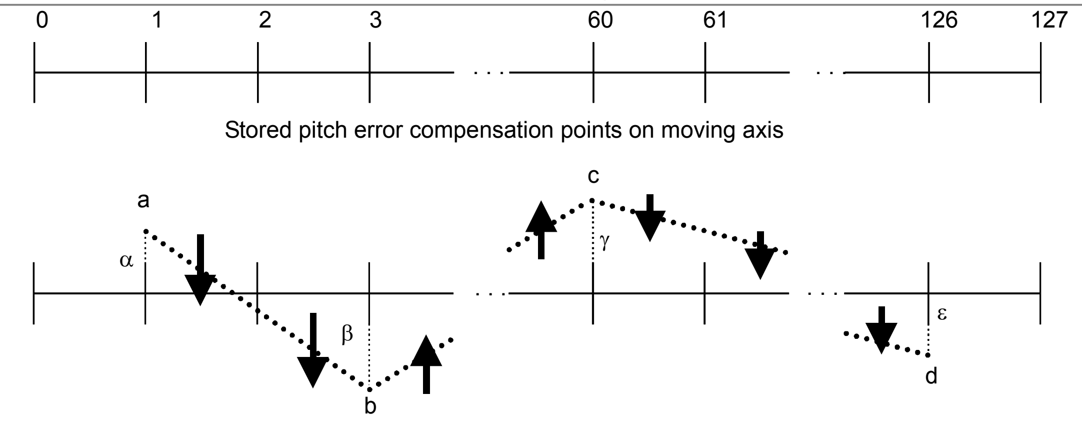
In above figure, the compensation amounts at the individual compensation points located between points a and b are (β-α)/(b-a).

9.7 SIMPLE STRAIGHTNESS COMPENSATION (M SERIES)

M

For a machine tool with a long stroke, deviations in straightness between axes may affect the machining accuracy. For this reason, when an axis moves, other axes are compensated in detection units to improve straightness. This improvement results in better machining accuracy. As a moving axis (set in a parameter) moves, compensation is applied to the compensation axis (set in a parameter) during pitch error compensation for the moving axis.

- Pitch error compensation points on moving axis and compensation value



Output timing of stored pitch error compensation of moving axis and output compensation pulses

a , b , c , d: Compensation position numbers of the moving axis (This number is originally a pitch error compensation position number.)

α, β, γ, ε : Compensation amount for the compensation position number (Amount of compensation for compensation axis)

From the current position of the moving axis, the compensation amount for the corresponding compensation axis is calculated. Compensation for the compensation axis is performed at the timing of the output of pitch error compensation of the moving axis.

In the above figure, while the moving axis moves from compensation point a to compensation point b, the compensation amount (β-α)/(b-a) is output for the compensation axis at the output timing of pitch error compensation of the moving axis.

NOTE

1 The simple straightness compensation function can be used after a moving axis and its compensation axis have returned to the reference position.

2 To add the simple straightness compensation function option, the stored pitch error compensation is enabled.

3 Simple straightness compensation data is superposed on stored pitch error compensation data and output. Simple straightness compensation is performed at pitch error compensation intervals.

4 Simple straightness compensation does not allow the moving axis to be used as a compensation axis. To implement such compensation, use inclination compensation.

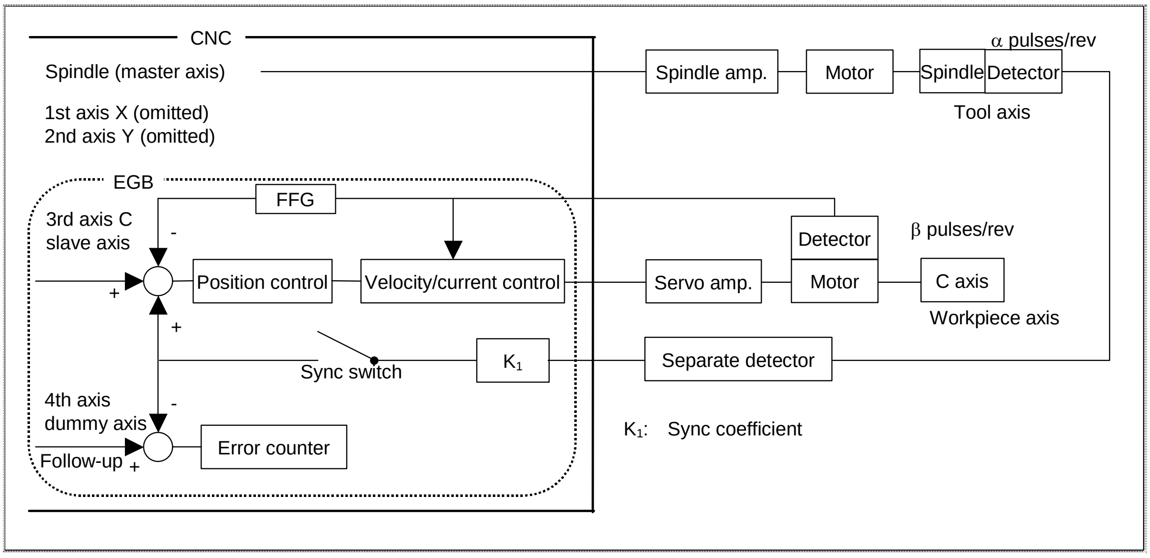
10 ELECTRONIC GEAR BOX (M SERIES)

Chapter 10, "ELECTRONIC GEAR BOX", consists of the following sections:

10.1 ELECTRONIC GEAR BOX (M SERIES)

M

This function synchronizes the revolutions of the workpiece axis connected to the servo motor with the revolutions of the tool axis (grinding stone/hob) connected to the spindle motor so as to machine (grind/cut) gears as in the hobbing machine function. The synchronization ratio can be specified by a program. Synchronization of the tool axis and the workpiece axis by this function is directly controlled by a digital servo, so the workpiece axis can track changes in the speed of the tool axis with no error, thereby achieving high-precision machining of gears. In the following descriptions, the electric gear box is called the EGB.



Format

	Parameter EFX(No.7731#0)=0	Parameter EFX(No.7731#0)=1
Start of synchronization	**G81 T__ (L__) (Q__ P__) ;**	**G81.4 T__ (L__) (Q__ P__) ;**
Cancellation of synchronization	**G80 ;**	**G80.4 ;**

T(or R) : Number of teeth (Specifiable range: 1 to 1000)

L : Number of hob threads (Specifiable range: -200 to +200)
The sign of L determines the direction of rotation for the workpiece axis.
When L is positive, the direction of rotation for the workpiece axis is positive (+ direction).
When L is negative, the direction of rotation for the workpiece axis is negative (- direction).
When L is 0, it follows the setting of bit 3 (LZR) of parameter No.7701.
If L is not specified, the number of hob threads is assumed 1.

Q : Module or diametral pitch
Specify a module in the case of metric input.
(Unit: 0.00001mm, Specifiable range: 0.01 to 25.0mm)
Specify a diametral pitch in the case of inch input.
(Unit: 0.00001inch^{-1}, Specifiable range: 0.01 to 254.0 inch^{-1})

P : Gear helix angle
(Unit: 0.0001deg, Specifiable range: -90.0 to 90.0deg)

* When specifying Q and P, the user can use a decimal point.

- Synchronization coefficient

A synchronization coefficient is internally represented using a fraction (Kn/Kd) to eliminate an error. The formula below is used for calculation.

$$\text{Synchronization coefficient} = \frac{K_n}{K_d} = \frac{L}{T} \times \frac{\beta}{\alpha}$$

where

L : Number of hob threads

T : Number of teeth

α : Number of pulses of the position detector per rotation about the master axis

β : Number of pulses of the position detector per rotation about the slave axis

Kn / Kd is a value resulting from reducing the right side of the above formula, but the result of reduction is subject to the following restrictions:

$-2147483648 \leq K_n \leq 2147483647$

$1 \leq K_d \leq 65535$

When this restriction is not satisfied, the alarm is issued when G81 is specified.

11 EDITING OPERATION

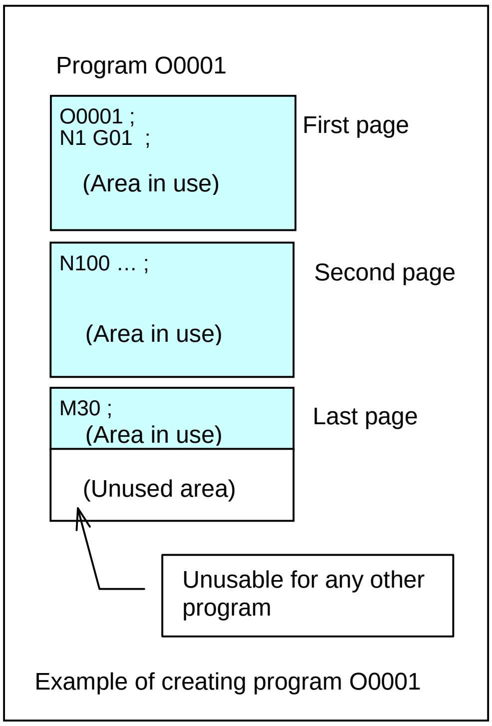
Chapter 11, "EDITING OPERATION", consists of the following sections:

11.1 PART PROGRAM STORAGE SIZE / NUMBER OF REGISTERABLE PROGRAMS

The following table lists the combinations of program storage sizes and the total number of registrable programs.

Part program storage size	**Number of registerable programs**	**0*i*-D**		**0*i* Mate-D**	
		M	**T**	**M**	**T**
320Kbyte	400	○B	○B	－	－
512Kbyte	400	○A	○A	○	○
1Mbyte	800	－	*2	－	－
2Mbyte	400	☆	－	－	－

○: Standard package (A/B) *2: 2-path system ☆: Optional



Example of creating program O0001

NOTE

1. The program storage size means the maximum size of a program if the program is the one and only program registered.
2. If more than one program is registered, the total size of registerable programs reduces for the following reason.
   The Series 0*i*-D/0*i* Mate-D manage programs in page units. The unit of program storage is managed also in page units. When a program is created, as many pages as necessary to store the program are secured, and the program is stored on these pages. Generally, the last program storage page has an unused area (left figure). This unused area cannot be used to store any other program. For the sake of program management, it is regarded as an area in use.
   The Series 0*i*-C uses a similar way of management, but the unit of pages in it differs from that in the Series 0*i*-D/0*i* Mate-D. So, if more than one program is registered in the Series 0*i*-D/0*i* Mate-D, the total program size of registerable programs in the Series 0*i*-D/0*i* Mate-D differs from that in the Series 0*i*-C.

11.2 PROGRAM EDITING

Following program editing operations are possible.

1. Creating new programs
2. Deleting programs
   - Deleting one program
   - Deleting specified programs at a time
   - Deleting all programs at a time
3. Editing a word
   - Inserting a word or block
   - Overwriting a word
   - Replacing a word or address (selection or global replacement)
   - Deleting a word or block
   - Copying, deleting, and pasting a specified range (on a word-by-word basis)
   - Copying and moving a program
4. Search
   - Searching for a program
   - Searching for a word or address
   - Repeating search
5. Rearranging program memory contents

An NC program on program memory is first stored in a continuous storage area. Editing the NC program divides the NC program storage area into several separate blocks, and as editing is repeated, these blocks are further divided into smaller blocks. When the NC program is divided into small blocks, the memory operation processing speed may decrease, or the program memory space used may increase excessively. So, program memory must be rearranged to store the program in a continuous area again.

11.3 PROGRAM PROTECT

The following functions are provided to protect programs:

1 Protection by parameter
By using a password parameter and a keyword parameter, this function provides a lock for a parameter that protects programs with program Nos. O9000 to O9999.
When locked (the password ≠ the keyword), the parameter for protection cannot be set to 0. In this case, program protection for programs with program Nos. O9000 to O9999 cannot be released unless the keyword is set correctly.
The values set in both the password and keyword parameters are not indicated. The lock is released by setting the keyword parameter to the same value as the value set in advance in the password parameter. When 0 is indicated in the password parameter, the password parameter is not yet set.

2 Protection of data at eight levels
Eight protection levels can be set to specify whether a change/external output is allowed. See "PROTECTION OF DATA AT EIGHT LEVELS" in Chapter 12, "SETTING AND DISPLAY".

11.4 PLAYBACK

In the TEACH IN JOG or TEACH IN HANDLE mode, you can create a program while inserting the coordinate of the current position along each axis in the absolute coordinate system when the tool is moved by manual operation into the program.
You can input the words other than axis names in the same way as in the EDIT mode.

11.5 BACKGROUND EDITING

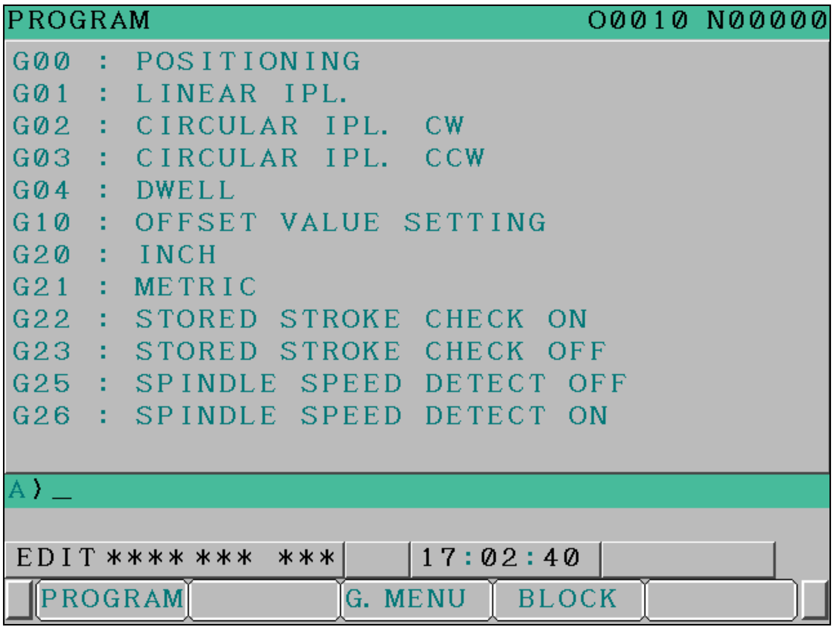
While a program is being executed, another program is edited. Such an edit operation is called background editing (BG editing). Background editing can be performed in exactly the same way as ordinary editing (foreground editing).

11.6 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

Guidance on the following items can be displayed together with illustration on the screen to assist CNC format programming.

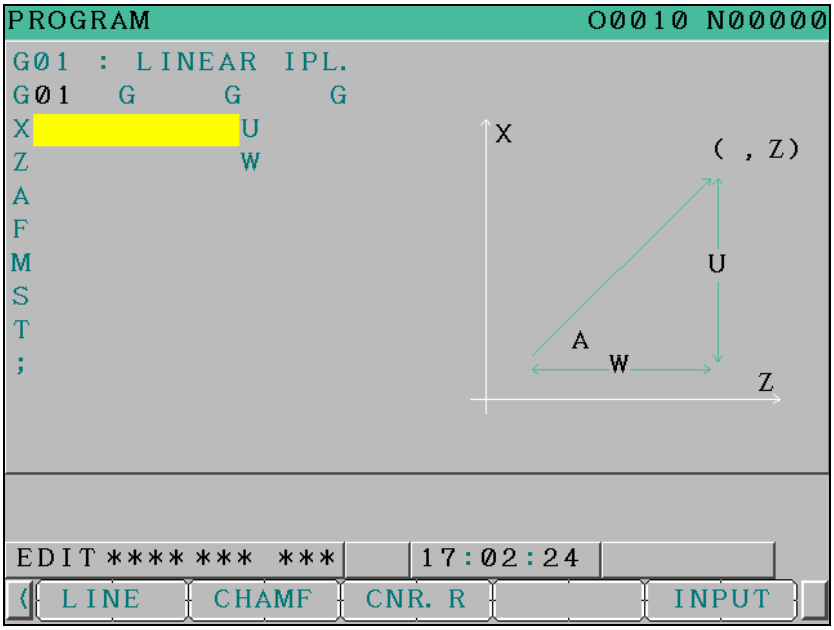
- List of G codes
- One block of the standard format for G codes

It is possible to create a program by inputting the necessary data with reference to the guidance.



```
PROGRAM                                   O0010 N00000
G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL.  CW
G03 : CIRCULAR IPL.  CCW
G04 : DWELL
G10 : OFFSET VALUE SETTING
G20 : INCH
G21 : METRIC
G22 : STORED STROKE CHECK ON
G23 : STORED STROKE CHECK OFF
G25 : SPINDLE SPEED DETECT OFF
G26 : SPINDLE SPEED DETECT ON

A)_

EDIT**** ***  ***        17:02:40
```

↓ When G01 is selected



```
PROGRAM                                   O0010 N00000
G01 : LINEAR IPL.
G01   G     G     G
X              U
Z              W
A
F
M
S
T
;

EDIT**** ***  ***        17:02:24
```

11.7 MEMORY CARD PROGRAM OPERATION/EDITING

A program held in a program storage file (named "FANUCPRG.BIN") on the memory card can be selected as a main program to perform memory operation.
Moreover, the contents of a program storage file can be referenced using the program directory screen, and a program held in a program storage can be edited using the program editing screen.

A program storage file can be created using a memory card program tool on a commercially available personal computer. To use a created program storage file, the file is written to a memory card prepared in the FAT format.

The maximum number of program storage files that can be written to a memory card is 63.

12 SETTING AND DISPLAY

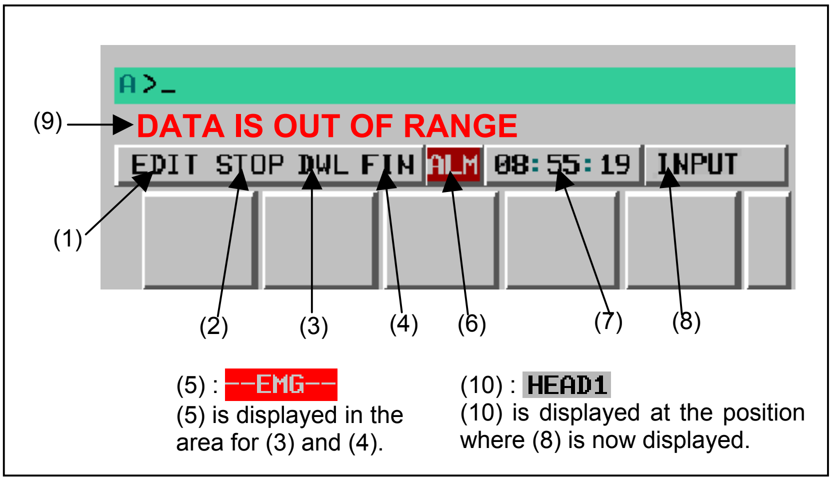
In the subsequent explanation in this chapter, the screen descriptions are provided based on a 10.4" LCD of the 12-soft-key type.

Chapter 12, "SETTING AND DISPLAY", consists of the following sections:

12.1 STATUS DISPLAY

The current mode, automatic operation state, alarm state, and program editing state are displayed on the bottom right on the screen allowing the operator to readily understand the operation condition of the system.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed. This prevents invalid data setting and input/output errors.

Explanation



DATA IS OUT OF RANGE

EDIT STOP DWL FIN ALM 08:55:19 INPUT

(5) is displayed in the area for (3) and (4).

(10) is displayed at the position where (8) is now displayed.

Fig. 12.1 (a) Positions of status display items

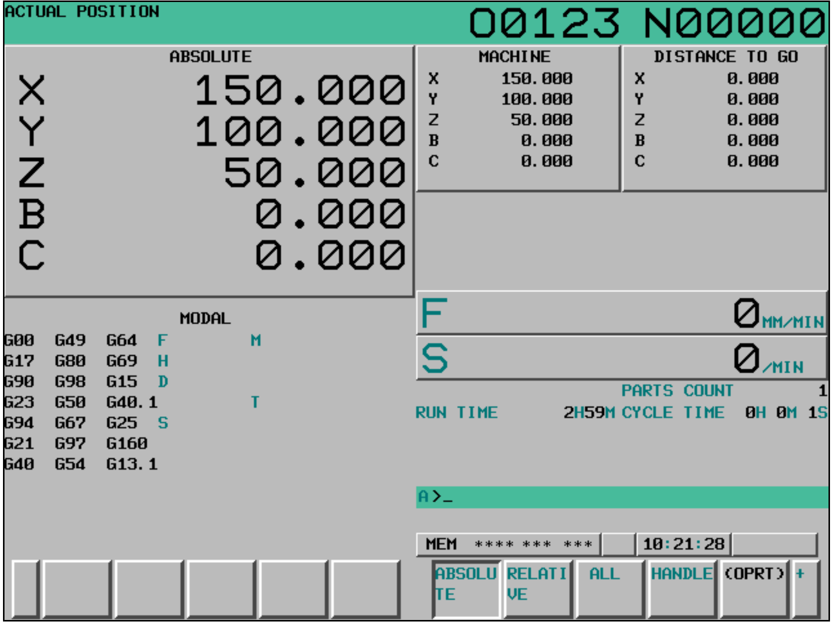
(1) Current mode
(2) Automatic operation status
(3) Axis moving status/dwell status
(4) State in which an auxiliary function is being executed
(5) Emergency stop or reset status
(6) Alarm status
  ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
  BAT : Indicates that the voltage of the lithium battery (the backup battery of the CNC) has decreased. (Blinks in reversed display.)
  APC : Indicates that the voltage of the backup battery of the absolute pulse coder has decreased. (Blinks in reversed display.)
  FAN : Indicates that the rotation speed of the fan has decreased. (Blinks in reversed display.)
  Space : Indicates a state other than the above.
(7) Current time
(8) Program editing status
(9) Warning for data setting or input/output operation
(10) Path name

12.2 CLOCK FUNCTION

Time is displayed in the hour/minute/second format on each display screen. Some screens allow display of the year, month, and day.
The custom macro system variable can be used to read the time. On PMC side, a window command can be used to know the current time.

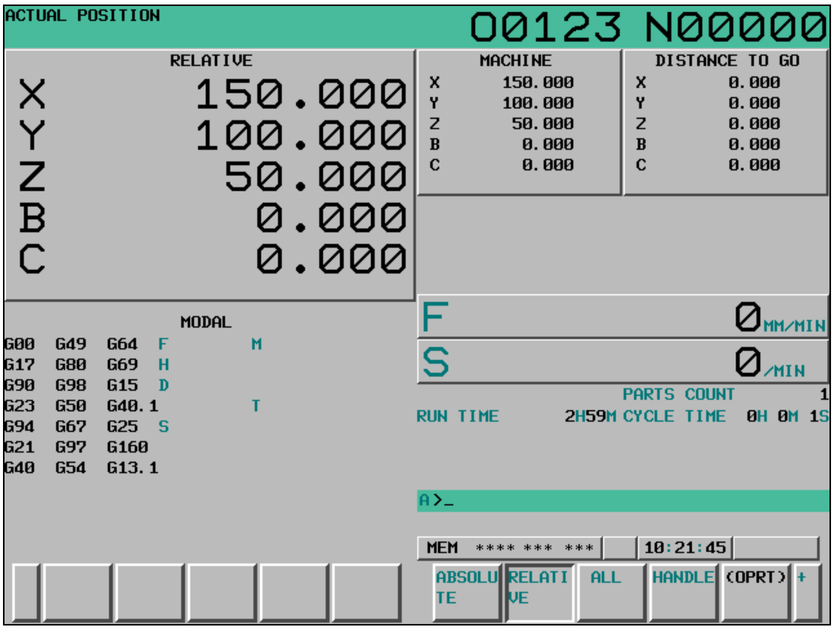
12.3 CURRENT POSITION DISPLAY

The current position and the remaining distance in the relative, workpiece, and machine coordinate systems are displayed.



```
ACTUAL POSITION                          O0123 N00000
           ABSOLUTE        MACHINE          DISTANCE TO GO
X       150.000          X   150.000      X       0.000
Y       100.000          Y   100.000      Y       0.000
Z        50.000          Z    50.000      Z       0.000
B         0.000          B     0.000      B       0.000
C         0.000          C     0.000      C       0.000

             MODAL                   F                  0 MM/MIN
G00  G49  G64  F       M             S                  0 /MIN
G17  G80  G69  H                                 PARTS COUNT     1
G90  G98  G15  D                     RUN TIME     2H59M CYCLE TIME  0H 0M 1S
G23  G50  G40.1        T
G94  G67  G25  S
G21  G97  G160
G40  G54  G13.1
                                     A>_

                                     MEM  **** *** ***    10:21:28
```

Fig. 12.3 (a) Current position (absolute coordinate) screen



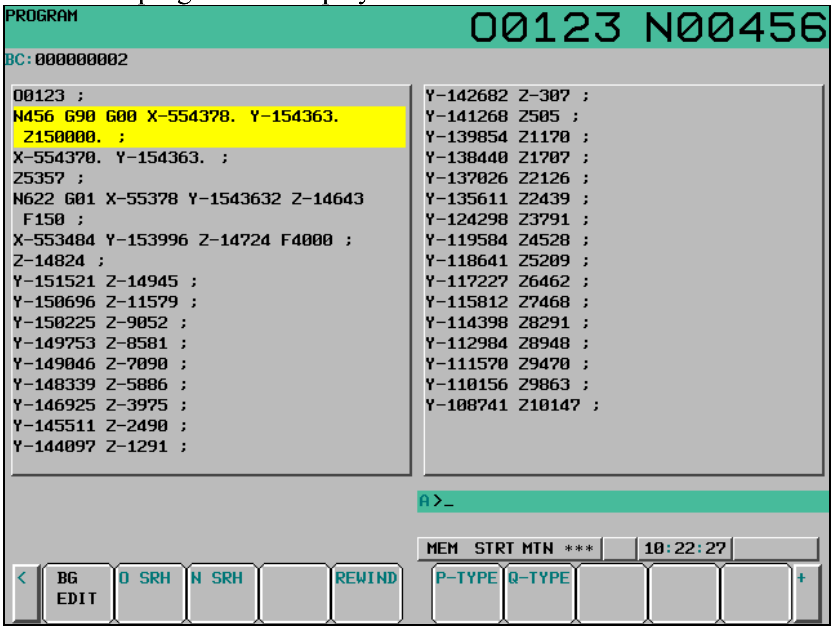
```
ACTUAL POSITION                          O0123 N00000
           RELATIVE        MACHINE          DISTANCE TO GO
X       150.000          X   150.000      X       0.000
Y       100.000          Y   100.000      Y       0.000
Z        50.000          Z    50.000      Z       0.000
B         0.000          B     0.000      B       0.000
C         0.000          C     0.000      C       0.000

             MODAL                   F                  0 MM/MIN
G00  G49  G64  F       M             S                  0 /MIN
G17  G80  G69  H                                 PARTS COUNT     1
G90  G98  G15  D                     RUN TIME     2H59M CYCLE TIME  0H 0M 1S
G23  G50  G40.1        T
G94  G67  G25  S
G21  G97  G160
G40  G54  G13.1
                                     A>_

                                     MEM  **** *** ***    10:21:45
```

Fig. 12.3 (b) Current position (relative coordinate) screen

12.4 PROGRAM DISPLAY

The contents of a program being edited or executed are displayed.

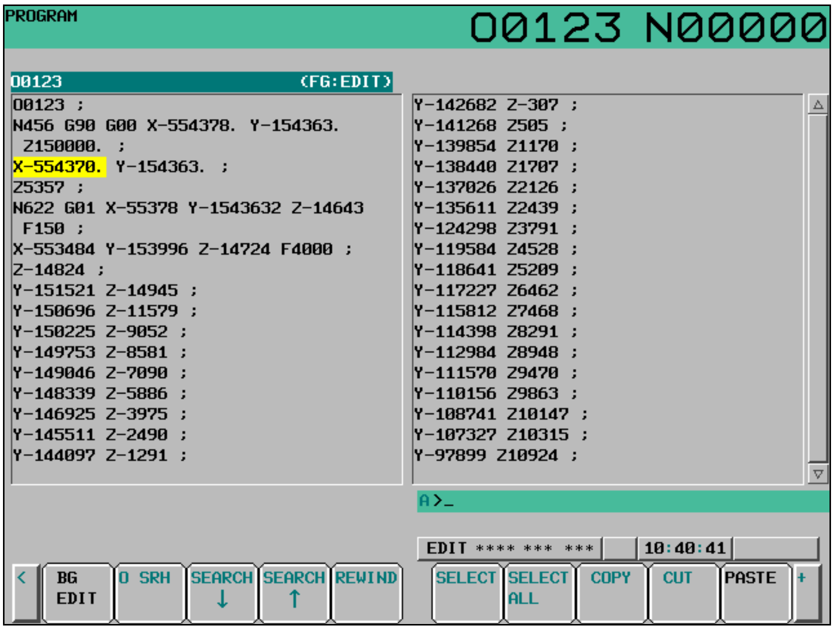
In addition, a list of the file names of programs registered in program memory is displayed. On the list screen, the amount of memory used, the amount of remaining memory, and the size, comment statement, and update date/time of each program are displayed.



```
O0123 ;
N456 G90 G00 X-554378. Y-154363.
 Z150000. ;
X-554370. Y-154363. ;
Z5357 ;
N622 G01 X-55378 Y-1543632 Z-14643
 F150 ;
X-553484 Y-153996 Z-14724 F4000 ;
Z-14824 ;
Y-151521 Z-14945 ;
Y-150696 Z-11579 ;
Y-150225 Z-9052 ;
Y-149753 Z-8581 ;
Y-149046 Z-7090 ;
Y-148339 Z-5886 ;
Y-146925 Z-3975 ;
Y-145511 Z-2490 ;
Y-144097 Z-1291 ;
```

```
Y-142682 Z-307 ;
Y-141268 Z505 ;
Y-139854 Z1170 ;
Y-138440 Z1707 ;
Y-137026 Z2126 ;
Y-135611 Z2439 ;
Y-124298 Z3791 ;
Y-119584 Z4528 ;
Y-118641 Z5209 ;
Y-117227 Z6462 ;
Y-115812 Z7468 ;
Y-114398 Z8291 ;
Y-112984 Z8948 ;
Y-111570 Z9470 ;
Y-110156 Z9863 ;
Y-108741 Z10147 ;
```

Fig. 12.4 (a) Screen displaying a program being executed

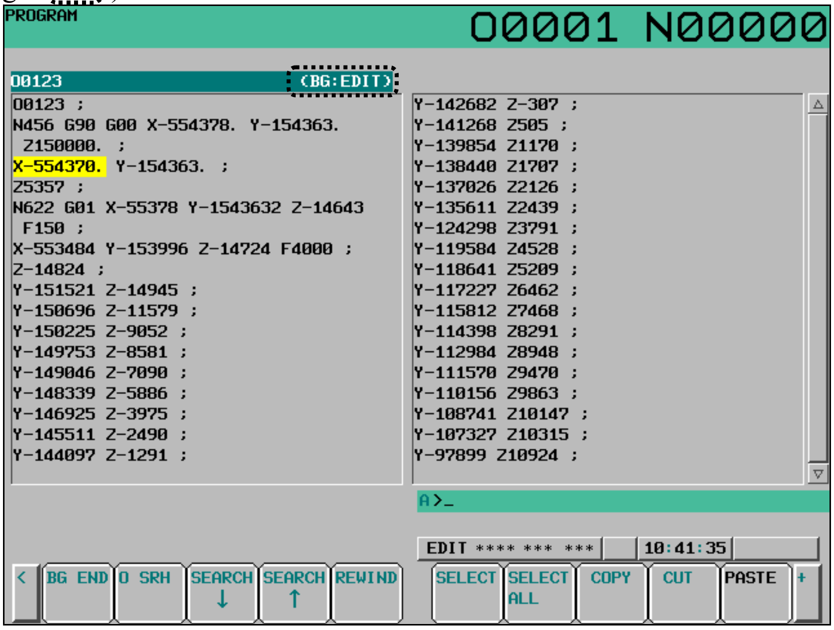


```
O0123 ;
N456 G90 G00 X-554378. Y-154363.
 Z150000. ;
X-554370. Y-154363. ;
Z5357 ;
N622 G01 X-55378 Y-1543632 Z-14643
 F150 ;
X-553484 Y-153996 Z-14724 F4000 ;
Z-14824 ;
Y-151521 Z-14945 ;
Y-150696 Z-11579 ;
Y-150225 Z-9052 ;
Y-149753 Z-8581 ;
Y-149046 Z-7090 ;
Y-148339 Z-5886 ;
Y-146925 Z-3975 ;
Y-145511 Z-2490 ;
Y-144097 Z-1291 ;
```

```
Y-142682 Z-307 ;
Y-141268 Z505 ;
Y-139854 Z1170 ;
Y-138440 Z1707 ;
Y-137026 Z2126 ;
Y-135611 Z2439 ;
Y-124298 Z3791 ;
Y-119584 Z4528 ;
Y-118641 Z5209 ;
Y-117227 Z6462 ;
Y-115812 Z7468 ;
Y-114398 Z8291 ;
Y-112984 Z8948 ;
Y-111570 Z9470 ;
Y-110156 Z9863 ;
Y-108741 Z10147 ;
Y-107327 Z10315 ;
Y-97899 Z10924 ;
```

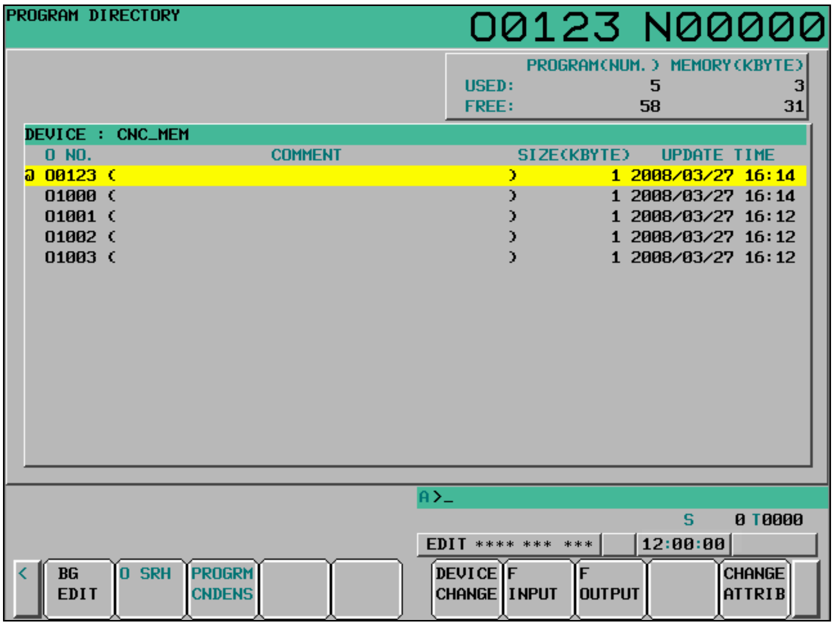
Fig. 12.4 (b) Screen displaying a program being edited

During background editing, "(BG:EDIT)" is displayed at the right of the program name. (Part enclosed by a dotted rectangle ⬚)



```
PROGRAM                                   O0001 N00000

O0123                      (BG:EDIT)
O0123 ;                                  Y-142682 Z-307 ;
N456 G90 G00 X-554378. Y-154363.         Y-141268 Z505 ;
 Z150000. ;                              Y-139854 Z1170 ;
X-554370. Y-154363. ;                    Y-138440 Z1707 ;
Z5357 ;                                  Y-137026 Z2126 ;
N622 G01 X-55378 Y-1543632 Z-14643       Y-135611 Z2439 ;
 F150 ;                                  Y-124298 Z3791 ;
X-553484 Y-153996 Z-14724 F4000 ;        Y-119584 Z4528 ;
Z-14824 ;                                Y-118641 Z5209 ;
Y-151521 Z-14945 ;                       Y-117227 Z6462 ;
Y-150696 Z-11579 ;                       Y-115812 Z7468 ;
Y-150225 Z-9052 ;                        Y-114398 Z8291 ;
Y-149753 Z-8581 ;                        Y-112984 Z8948 ;
Y-149046 Z-7090 ;                        Y-111570 Z9470 ;
Y-148339 Z-5886 ;                        Y-110156 Z9863 ;
Y-146925 Z-3975 ;                        Y-108741 Z10147 ;
Y-145511 Z-2490 ;                        Y-107327 Z10315 ;
Y-144097 Z-1291 ;                        Y-97899 Z10924 ;

                                         A>_

                                         EDIT **** *** ***    10:41:35
```

Fig. 12.4 (c) Program display screen displayed during background editing

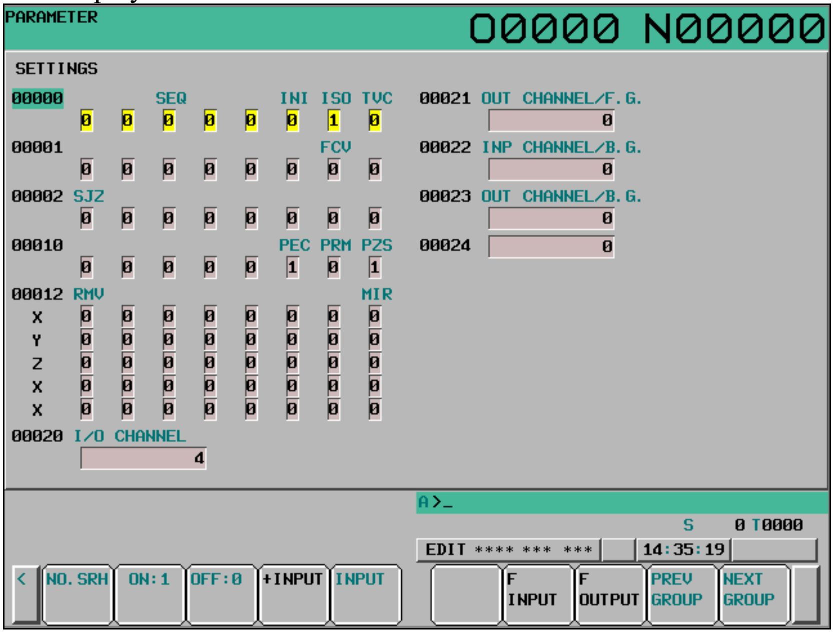


```
PROGRAM DIRECTORY                          O0123 N00000
                                  PROGRAM(NUM.) MEMORY(KBYTE)
                              USED:           5            3
                              FREE:          58           31
 DEVICE : CNC_MEM
   O NO.              COMMENT               SIZE(KBYTE)   UPDATE TIME
 @ O0123 (                                )     1 2008/03/27 16:14
   O1000 (                                )     1 2008/03/27 16:14
   O1001 (                                )     1 2008/03/27 16:12
   O1002 (                                )     1 2008/03/27 16:12
   O1003 (                                )     1 2008/03/27 16:12

                                  A>_
                                                      S     0 T0000
                                  EDIT **** *** ***    12:00:00
```

Fig. 12.4 (d) Program folder screen

12.5 PARAMETER SETTING AND DISPLAY

Parameter settings are displayed.



```
PARAMETER                                   O0000 N00000
SETTINGS
00000           SEQ          INI ISO TVC   00021 OUT CHANNEL/F.G.
         0   0   0   0   0   0   1   0           [          0]
00001                            FCV       00022 INP CHANNEL/B.G.
         0   0   0   0   0   0   0   0           [          0]
00002 SJZ                                  00023 OUT CHANNEL/B.G.
         0   0   0   0   0   0   0   0           [          0]
00010                        PEC PRM PZS   00024 [          0]
         0   0   0   0   0   1   0   1
00012 RMV                            MIR
  X      0   0   0   0   0   0   0   0
  Y      0   0   0   0   0   0   0   0
  Z      0   0   0   0   0   0   0   0
  X      0   0   0   0   0   0   0   0
  X      0   0   0   0   0   0   0   0
00020 I/O CHANNEL
       [          4]
                                         A>_
                                                      S      0 T0000
                                         EDIT **** *** ***   14:35:19
```

Fig. 12.5 (a) Parameter screen

12.6 ALARM DISPLAY

The currently issued alarm number and a brief description of the alarm are displayed.

```
ALARM MESSAGE                               O0123 N00000
PS0010 IMPROPER G-CODE

                                         A>_
                                         MEM STOP *** *** ALM 10:44:15
```

Fig. 12.6 (a) Alarm screen

12.7 ALARM HISTORY DISPLAY

Information about up to 50 alarms including the latest alarm that have been issued in the CNC is stored and displayed on the screen.

The following information is displayed for each alarm:

(1) Date and time of alarm issuance
(2) Alarm type
(3) Alarm number
(4) Alarm message (This item may not appear for some alarms.)

```
ALARM HISTORY                              O0001 N00000
                                         TOTAL ALARM:  50
0001 2008/02/26 10:44:36
PS0010 IMPROPER G-CODE
0002 2008/02/26 10:44:03
PS0010 IMPROPER G-CODE
0003 2008/02/26 10:32:40
SW0100 PARAMETER ENABLE SWITCH ON
0004 2008/02/26 10:26:36
SR1966 FILE NOT FOUND (MEMORY CARD)
0005 2008/02/26 10:20:10
PS0224 ZERO RETURN NOT FINISHED
0006 2008/02/26 10:18:57
SW0100 PARAMETER ENABLE SWITCH ON
0007 2008/02/26 10:18:04
PW0000 POWER MUST BE OFF
0008 2008/02/26 10:17:42
SW0100 PARAMETER ENABLE SWITCH ON

                                        A>_
                                        REF **** *** ***   10:46:05
```

Fig. 12.7 (a) Alarm history screen

12.8 OPERATOR MESSAGE HISTORY DISPLAY

A history of operator messages can be stored. The stored history can be referenced on the operator message history screen.

12.9 OPERATION HISTORY DISPLAY

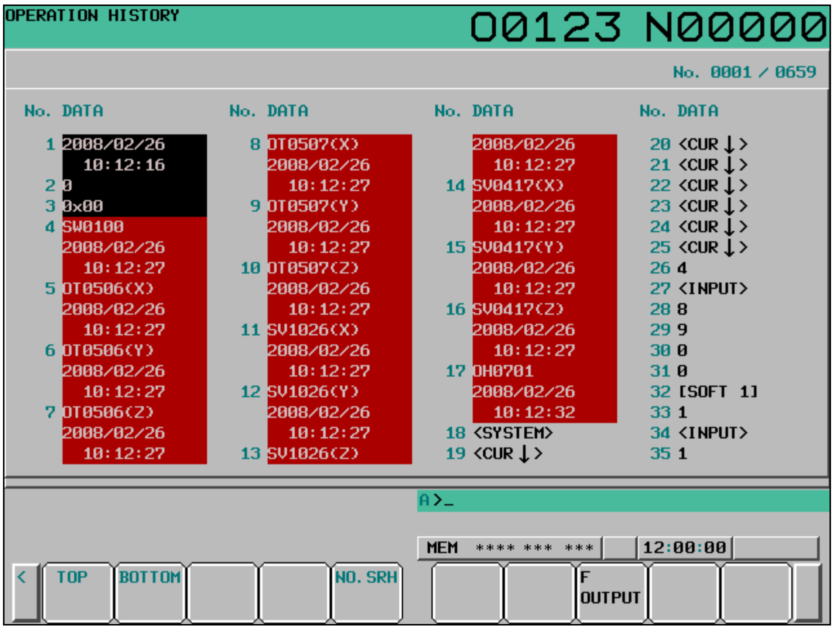
A history of events such as operations performed by the operator, alarms issued, and external operator messages can be recorded and referenced, and history data can be output.

The following data is recorded as history data:

(1) MDI key operations performed by the operator
(2) On/off status transition of I/O signals (X, Y, G, and F)
(3) Alarms issued
(4) Time stamp (date and time)
(5) External operator message

Recorded data can be referenced on the operation history screen.

In addition to display of history data, search, erasure, and output to external I/O devices can be performed for history data. While this screen is being displayed, no history data is recorded.

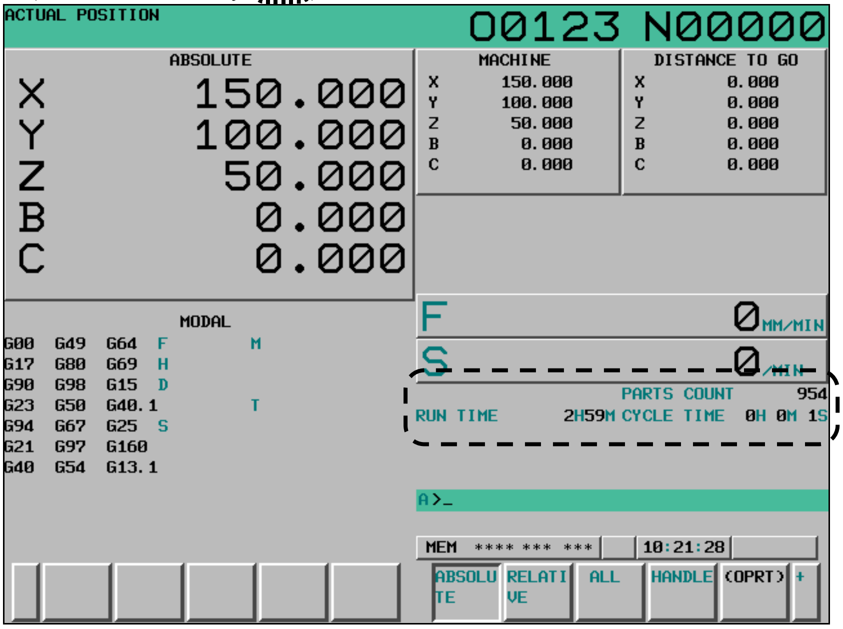


```
OPERATION HISTORY                                   O0123 N00000
                                                    No. 0001 / 0659
No. DATA           No. DATA           No. DATA           No. DATA
 1 2008/02/26       8 OT0507(X)          2008/02/26      20 <CUR↓>
      10:12:16        2008/02/26            10:12:27     21 <CUR↓>
 2 0                     10:12:27     14 SV0417(X)       22 <CUR↓>
 3 0x00             9 OT0507(Y)          2008/02/26      23 <CUR↓>
 4 SW0100             2008/02/26            10:12:27     24 <CUR↓>
   2008/02/26            10:12:27     15 SV0417(Y)       25 <CUR↓>
      10:12:27     10 OT0507(Z)          2008/02/26      26 4
 5 OT0506(X)          2008/02/26            10:12:27     27 <INPUT>
   2008/02/26            10:12:27     16 SV0417(Z)       28 8
      10:12:27     11 SV1026(X)          2008/02/26      29 9
 6 OT0506(Y)          2008/02/26            10:12:27     30 0
   2008/02/26            10:12:27     17 OH0701          31 0
      10:12:27     12 SV1026(Y)          2008/02/26      32 [SOFT 1]
 7 OT0506(Z)          2008/02/26            10:12:32     33 1
   2008/02/26            10:12:27     18 <SYSTEM>        34 <INPUT>
      10:12:27     13 SV1026(Z)       19 <CUR↓>          35 1

                                      A>_
                                      MEM **** *** ***   12:00:00
```

Fig. 12.9 (a) Operation history screen

12.10 RUN HOUR AND PARTS COUNT DISPLAY

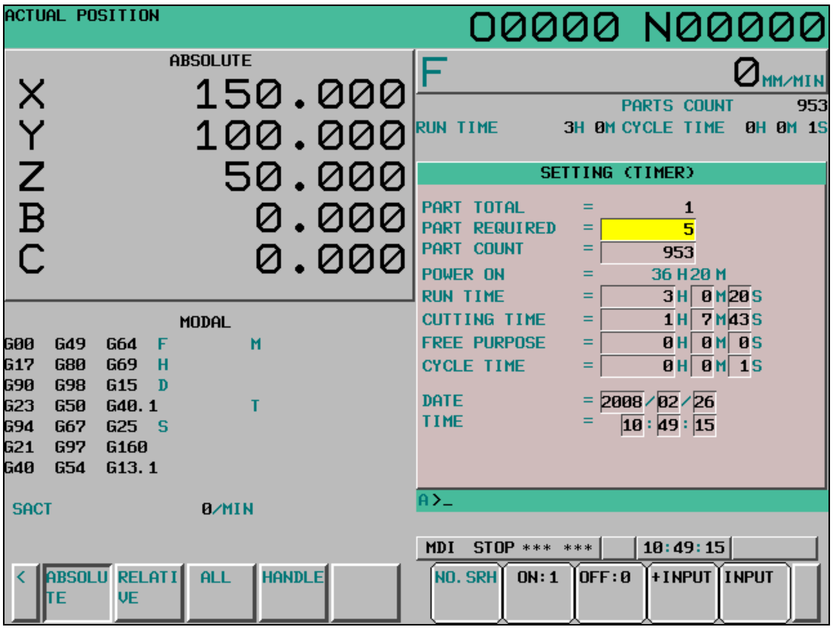
On the current position screen, operation times (run time and cycle time) and the number of machined parts are displayed. (Area indicated by ⬚)



```
ACTUAL POSITION                                     O0123 N00000
            ABSOLUTE                 MACHINE          DISTANCE TO GO
X        150.000              X    150.000         X       0.000
Y        100.000              Y    100.000         Y       0.000
Z         50.000              Z     50.000         Z       0.000
B          0.000              B      0.000         B       0.000
C          0.000              C      0.000         C       0.000

              MODAL                F                        0 MM/MIN
G00  G49  G64  F        M
G17  G80  G69  H                   S                        0 /MIN
G90  G98  G15  D                                 PARTS COUNT     954
G23  G50  G40.1         T          RUN TIME   2H59M CYCLE TIME  0H 0M 1S
G94  G67  G25  S
G21  G97  G160
G40  G54  G13.1
                                   A>_
                                   MEM **** *** ***   10:21:28
```

Fig. 12.10 (a) Current position screen

The cumulative value of automatic operation time, the cumulative value of cutting time, and timer values that can be set freely can be modified and preset by MDI.

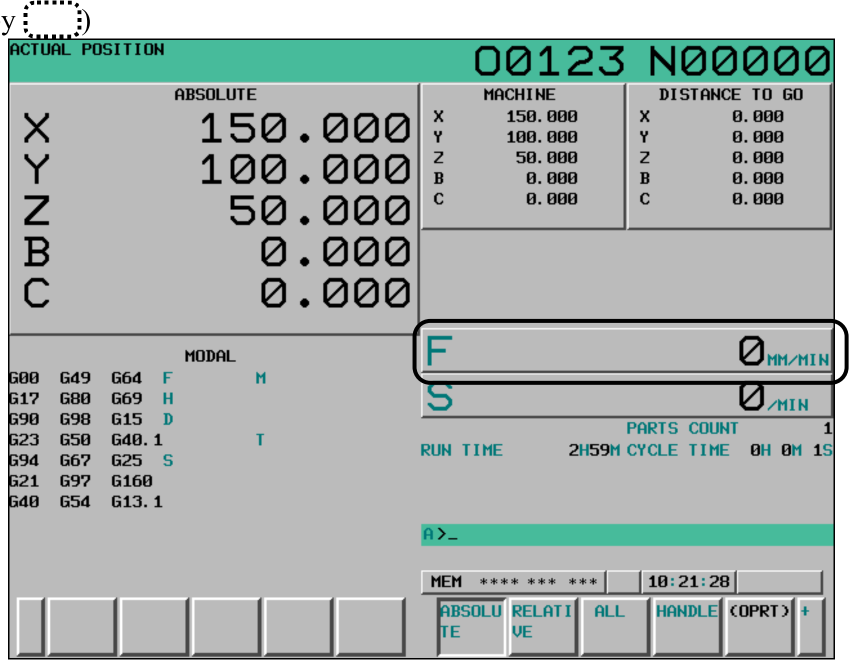


```
ACTUAL POSITION                         O0000 N00000
        ABSOLUTE                 F                  0 MM/MIN
X       150.000                         PARTS COUNT        953
Y       100.000                  RUN TIME    3H 0M CYCLE TIME   0H 0M 1S
Z        50.000                         SETTING (TIMER)
B         0.000                  PART TOTAL      =        1
C         0.000                  PART REQUIRED   =        5
                                 PART COUNT      =      953
        MODAL                    POWER ON        =   36H20M
G00 G49 G64  F    M              RUN TIME        =    3H  0M20S
G17 G80 G69  H                   CUTTING TIME    =    1H  7M43S
G90 G98 G15  D                   FREE PURPOSE    =    0H  0M 0S
G23 G50 G40.1     T              CYCLE TIME      =    0H  0M 1S
G94 G67 G25  S
G21 G97 G160                     DATE            = 2008/02/26
G40 G54 G13.1                    TIME            =   10:49:15

SACT            0/MIN            A>_
                                 MDI  STOP *** ***  10:49:15
```

Fig. 12.10 (b) Setting (timer) screen

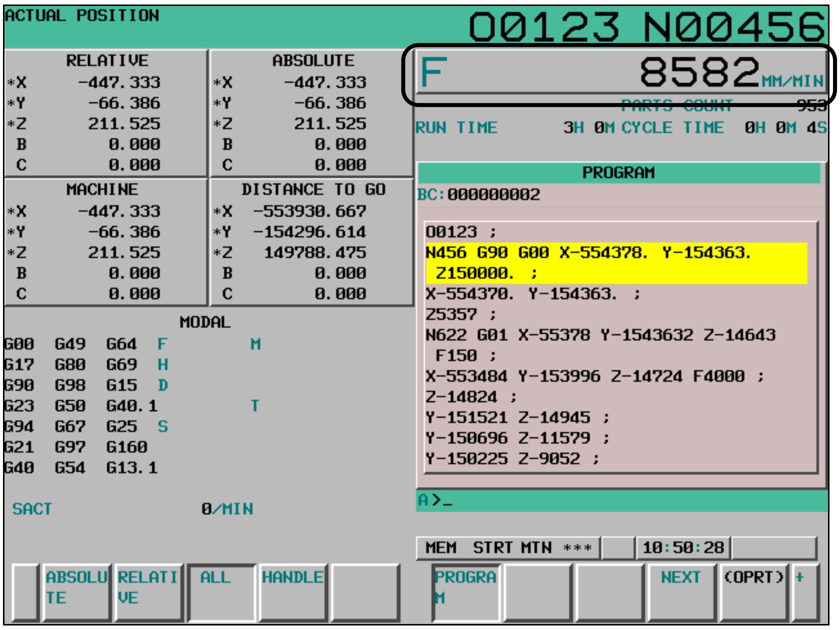
12.11 ACTUAL CUTTING FEEDRATE DISPLAY

The actual machine federate per minute can be indicated on the current position screen and program check screen by parameter setting.
(Area indicated by ⬚)



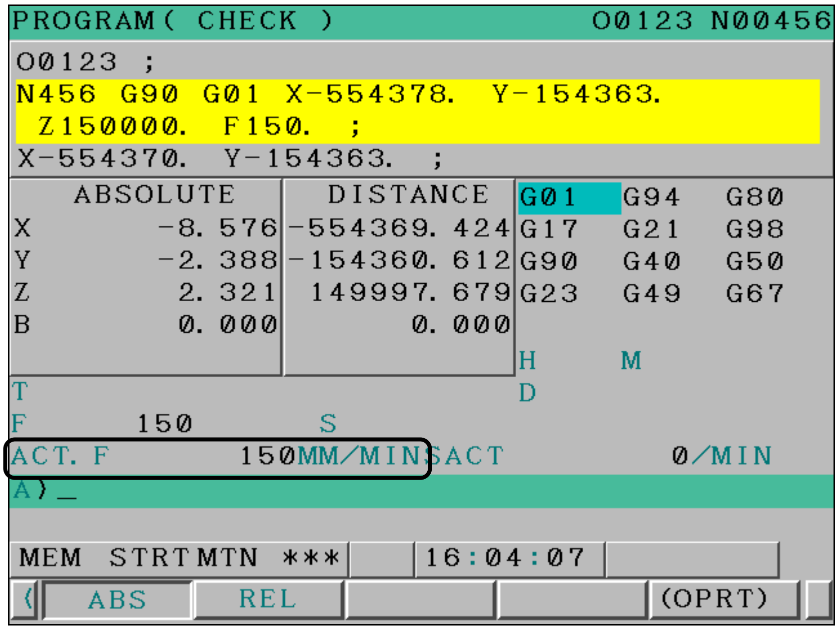
```
ACTUAL POSITION                         O0123 N00000
        ABSOLUTE                   MACHINE          DISTANCE TO GO
X       150.000                  X  150.000         X     0.000
Y       100.000                  Y  100.000         Y     0.000
Z        50.000                  Z   50.000         Z     0.000
B         0.000                  B    0.000         B     0.000
C         0.000                  C    0.000         C     0.000

        MODAL                    F                  0 MM/MIN
G00 G49 G64  F    M              S                  0 /MIN
G17 G80 G69  H                          PARTS COUNT          1
G90 G98 G15  D                   RUN TIME   2H59M CYCLE TIME   0H 0M 1S
G23 G50 G40.1     T
G94 G67 G25  S
G21 G97 G160
G40 G54 G13.1                    A>_
                                 MEM **** *** ***   10:21:28
```

Fig. 12.11 (a) Current position screen



```
ACTUAL POSITION                              O0123 N00456
       RELATIVE              ABSOLUTE         F          8582MM/MIN
*X     -447.333      *X     -447.333                  PARTS COUNT    953
*Y      -66.386      *Y      -66.386          RUN TIME      3H 0M CYCLE TIME   0H 0M 4S
*Z      211.525      *Z      211.525
 B        0.000       B        0.000                   PROGRAM
 C        0.000       C        0.000          BC:000000002
       MACHINE         DISTANCE TO GO         O0123 ;
*X     -447.333      *X  -553930.667          N456 G90 G00 X-554378. Y-154363.
*Y      -66.386      *Y  -154296.614           Z150000. ;
*Z      211.525      *Z   149788.475          X-554370. Y-154363. ;
 B        0.000       B        0.000          Z5357 ;
 C        0.000       C        0.000          N622 G01 X-55378 Y-1543632 Z-14643
                 MODAL                         F150 ;
G00  G49  G64  F        M                     X-553484 Y-153996 Z-14724 F4000 ;
G17  G80  G69  H                              Z-14824 ;
G90  G98  G15  D                              Y-151521 Z-14945 ;
G23  G50  G40.1         T                     Y-150696 Z-11579 ;
G94  G67  G25  S                              Y-150225 Z-9052 ;
G21  G97  G160
G40  G54  G13.1                               A>_

SACT                 0/MIN                    MEM  STRT MTN ***     10:50:28
```

Fig. 12.11 (b) Program screen (10.4-inch LCD)



```
PROGRAM( CHECK )                               O0123 N00456
O0123 ;
N456 G90 G01 X-554378. Y-154363.
 Z150000. F150. ;
X-554370. Y-154363. ;
     ABSOLUTE        DISTANCE      G01   G94   G80
X        -8.576  -554369.424       G17   G21   G98
Y        -2.388  -154360.612       G90   G40   G50
Z         2.321   149997.679       G23   G49   G67
B         0.000        0.000
                                   H     M
T                                  D
F        150              S
ACT. F          150MM/MIN SACT                0/MIN
A)_

MEM  STRT MTN ***       16:04:07
```

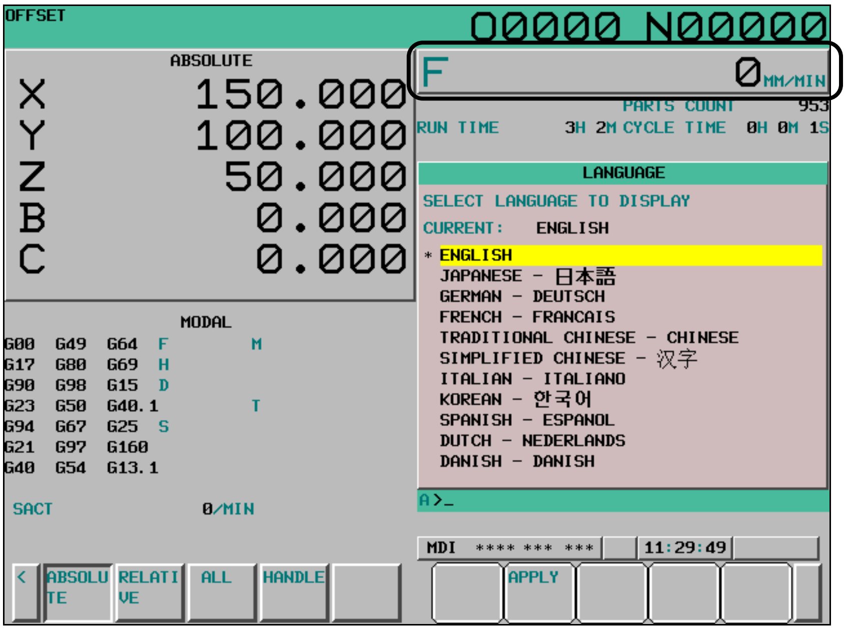
Fig. 12.11 (c) Program check screen (8.4-inch LCD)

The fractional portion of the actual feedrate (ACT. F) can also be displayed.
The number of digits of the fractional portion is shown below.

- Feed per minute: 0 to 3 digits (depends on the parameter setting.)

It is also possible to display the feed amount per revolution in the case of feed per revolution. In this case, the number of digits of the fractional portion is shown below.

- Feed per revolution (millimeter input): 2 digits
- Feed per revolution (inch input): 3 digits



X 150.000
Y 100.000
Z 50.000
B 0.000
C 0.000

```
G00  G49  G64   F        M
G17  G80  G69   H
G90  G98  G15   D
G23  G50  G40.1          T
G94  G67  G25   S
G21  G97  G160
G40  G54  G13.1
SACT         0/MIN
```

F 0 MM/MIN

PARTS COUNT 953

RUN TIME 3H 2M CYCLE TIME 0H 0M 1S

LANGUAGE

SELECT LANGUAGE TO DISPLAY

CURRENT: ENGLISH

- * ENGLISH
- JAPANESE - 日本語
- GERMAN - DEUTSCH
- FRENCH - FRANCAIS
- TRADITIONAL CHINESE - CHINESE
- SIMPLIFIED CHINESE - 汉字
- ITALIAN - ITALIANO
- KOREAN - 한국어
- SPANISH - ESPANOL
- DUTCH - NEDERLANDS
- DANISH - DANISH

A>_

Fig. 12.11(d) Program screen (display example for the 10.4-inch display unit)

12.12 DISPLAY OF SPINDLE SPEED AND T CODE AT ALL SCREENS

The spindle speed calculated from feedback pulses from the position coder mounted on the spindle and a T code specified by program execution can always displayed by parameter setting.

12.13 DIRECTORY DISPLAY OF FLOPPY CASSETTE

A list of file names in an external I/O device can be displayed on the screen (directory display).
File names that can be displayed in the directory are up to 17 characters long.

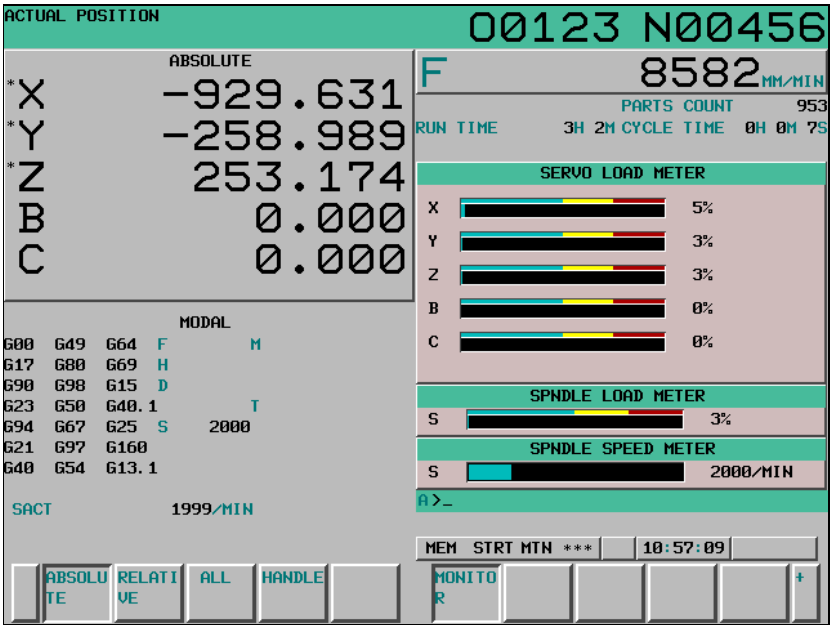
When a part program stored in program memory is written to an external I/O device, a program number can be assigned as the file name. When data other than part programs is written, a name specific to the data can be assigned.

12.14 OPTIONAL PATH NAME DISPLAY

An arbitrary character string can be displayed as a path name indicated in the status display field by parameter setting. Up to seven characters including numerical, alphabetical, and katakana characters and symbols can be displayed.

12.15 OPERATING MONITOR SCREEN

Load values (torque values) of spindle and servo motors are displayed in the bar graph form on the screen. In the bar graph display, the latest sampling values are indicated. By setting the rated load value of the motor corresponding to each load meter in a parameter in advance, the load meter shows 100% when the load value equals the rated load value.



```
ACTUAL POSITION                 O0123 N00456
        ABSOLUTE             F          8582MM/MIN
*X      -929.631                   PARTS COUNT    953
*Y      -258.989             RUN TIME    3H 2M CYCLE TIME  0H 0M 7S
*Z       253.174                    SERVO LOAD METER
 B         0.000             X                    5%
 C         0.000             Y                    3%
                             Z                    3%
             MODAL           B                    0%
G00  G49  G64   F        M   C                    0%
G17  G80  G69   H                   SPNDLE LOAD METER
G90  G98  G15   D            S                       3%
G23  G50  G40.1          T          SPNDLE SPEED METER
G94  G67  G25   S    2000    S                       2000/MIN
G21  G97  G160                A>_
G40  G54  G13.1
 SACT            1999/MIN
                              MEM  STRT MTN ***    10:57:09
```

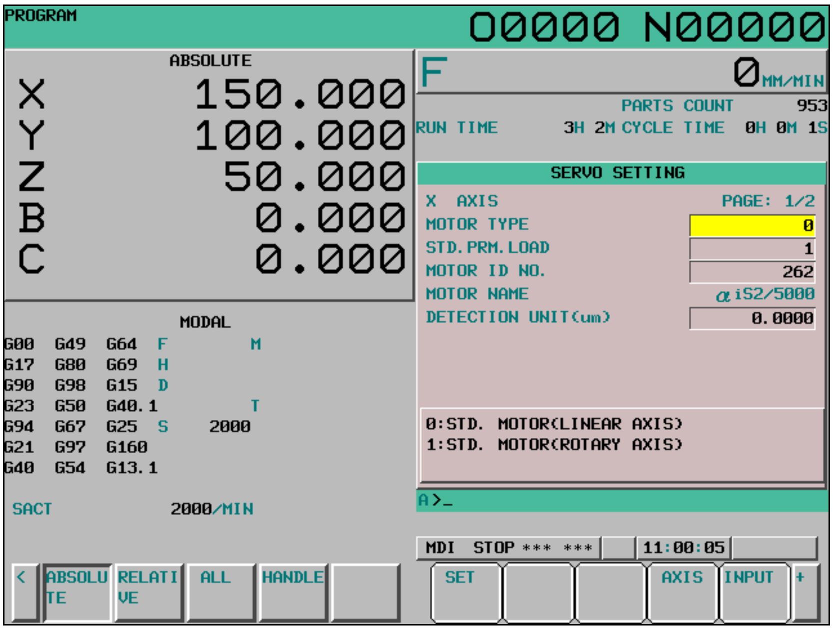
Fig. 12.15 (a) Operating monitor screen

12.16 SERVO SETTING SCREEN

Necessary parameters for basic servo motor tuning and the status monitor are displayed for each axis.

12.16.1 Servo Setting Screen

This screen summarizes necessary parameters for initial standard setting of servo motors. Parameter setting is also possible on this screen.

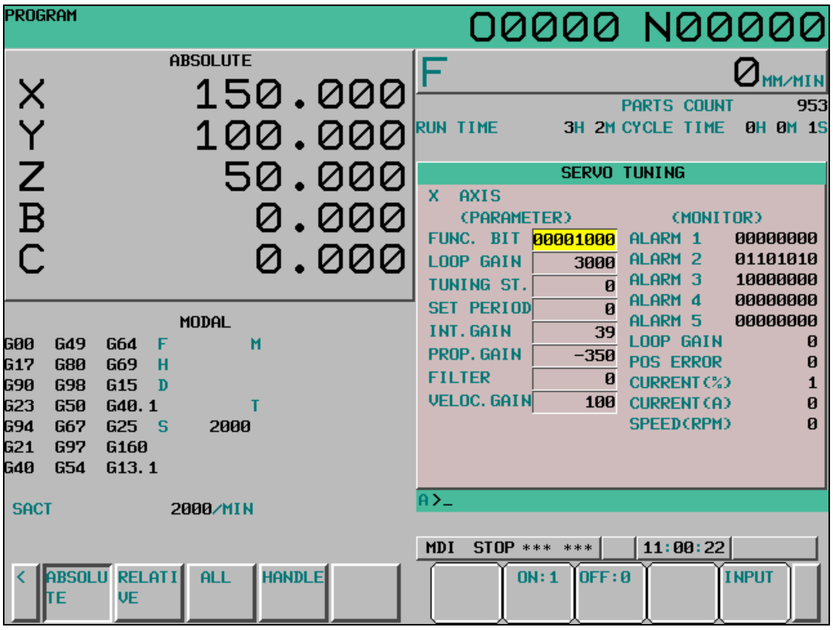


```
PROGRAM                                    O0000 N00000
            ABSOLUTE              F                 0MM/MIN
X          150.000                          PARTS COUNT    953
Y          100.000                RUN TIME   3H 2M CYCLE TIME  0H 0M 1S
Z           50.000                            SERVO SETTING
B            0.000                X  AXIS                 PAGE: 1/2
C            0.000                MOTOR TYPE                      0
                                  STD.PRM.LOAD                    1
             MODAL                MOTOR ID NO.                  262
G00  G49  G64  F       M          MOTOR NAME             αiS2/5000
G17  G80  G69  H                  DETECTION UNIT(um)         0.0000
G90  G98  G15  D
G23  G50  G40.1        T
G94  G67  G25  S    2000          0:STD. MOTOR(LINEAR AXIS)
G21  G97  G160                    1:STD. MOTOR(ROTARY AXIS)
G40  G54  G13.1
                                  A>_
SACT            2000/MIN
                                  MDI  STOP *** ***     11:00:05
```

Fig. 12.16.1 (a) Servo parameter setting screen

12.16.2 Servo Motor Tuning Screen

This screen summarizes necessary parameters for basic servo motor tuning and the status monitor for each axis.



```
PROGRAM                                    O0000 N00000
            ABSOLUTE              F                 0MM/MIN
X          150.000                          PARTS COUNT    953
Y          100.000                RUN TIME   3H 2M CYCLE TIME  0H 0M 1S
Z           50.000                            SERVO TUNING
B            0.000                X  AXIS
C            0.000                  (PARAMETER)          (MONITOR)
                                  FUNC. BIT  00001000  ALARM 1    00000000
             MODAL                LOOP GAIN      3000  ALARM 2    01101010
G00  G49  G64  F       M          TUNING ST.        0  ALARM 3    10000000
G17  G80  G69  H                  SET PERIOD        0  ALARM 4    00000000
G90  G98  G15  D                  INT.GAIN         39  ALARM 5    00000000
G23  G50  G40.1        T          PROP.GAIN      -350  LOOP GAIN         0
G94  G67  G25  S    2000          FILTER            0  POS ERROR         0
G21  G97  G160                    VELOC.GAIN      100  CURRENT(%)        1
G40  G54  G13.1                                        CURRENT(A)        0
                                                       SPEED(RPM)        0
                                  A>_
SACT            2000/MIN
                                  MDI  STOP *** ***     11:00:22
```

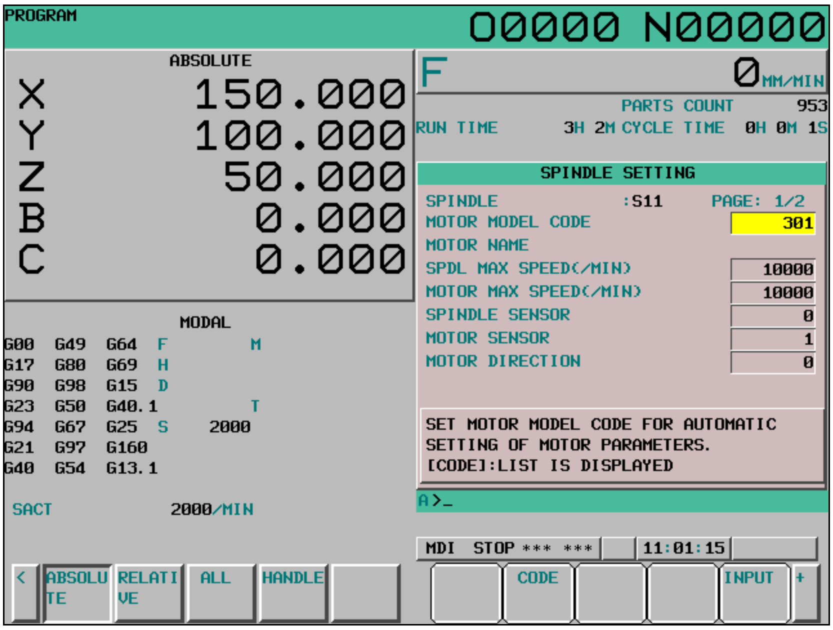
Fig. 12.16.2 (a) Servo motor tuning screen

12.17 SPINDLE SETTING SCREEN

Spindle-related parameters are set and displayed. Data other than parameters can also be displayed. Spindle setting, spindle tuning, and spindle monitor screens are provided.

12.17.1 Spindle Setting Screen

This screen summarizes necessary parameters for initial spindle setting.
Parameters can also be set.

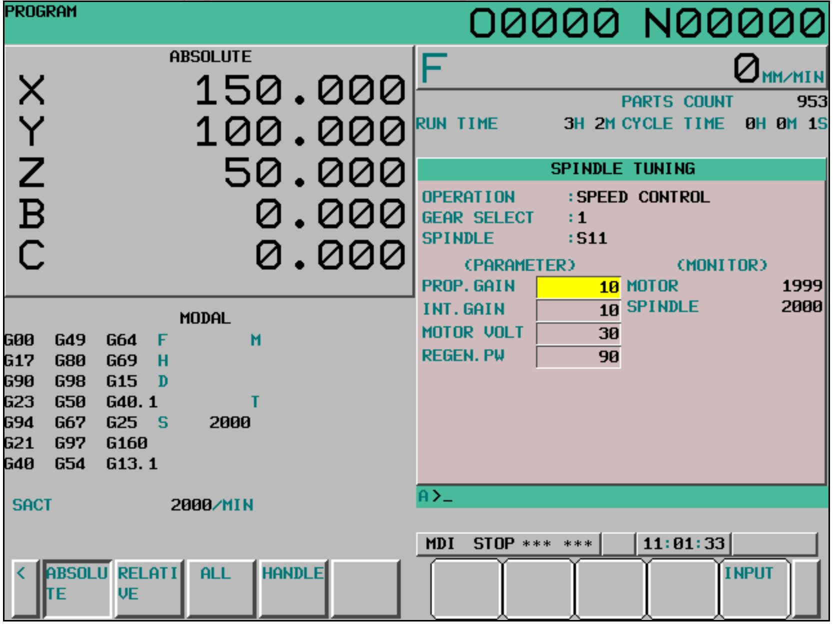


```
PROGRAM                                  O0000 N00000
            ABSOLUTE              F                 0MM/MIN
X          150.000                          PARTS COUNT    953
Y          100.000                RUN TIME     3H 2M CYCLE TIME  0H 0M 1S
Z           50.000                           SPINDLE SETTING
B            0.000                SPINDLE              :S11     PAGE: 1/2
C            0.000                MOTOR MODEL CODE                   301
                                  MOTOR NAME
            MODAL                 SPDL MAX SPEED(/MIN)             10000
G00  G49  G64   F        M        MOTOR MAX SPEED(/MIN)            10000
G17  G80  G69   H                 SPINDLE SENSOR                       0
G90  G98  G15   D                 MOTOR SENSOR                         1
G23  G50  G40.1          T        MOTOR DIRECTION                      0
G94  G67  G25   S   2000
G21  G97  G160                    SET MOTOR MODEL CODE FOR AUTOMATIC
G40  G54  G13.1                   SETTING OF MOTOR PARAMETERS.
                                  [CODE]:LIST IS DISPLAYED
SACT             2000/MIN         A>_
                                  MDI  STOP *** ***   11:01:15
```

Fig. 12.17.1 (a) Spindle setting screen

12.17.2 Spindle Tuning Screen

This screen summarizes necessary parameters for basic spindle tuning and the status monitor.

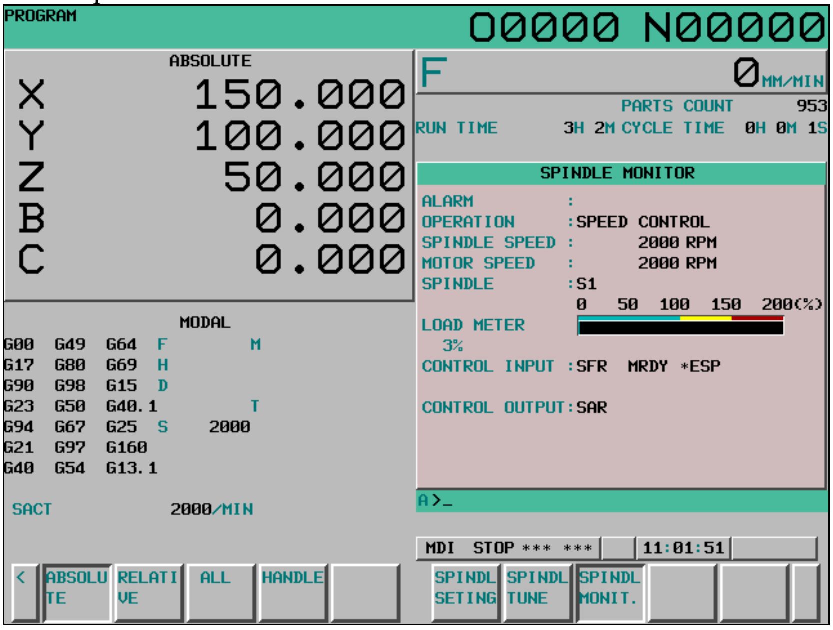


```
PROGRAM                                  O0000 N00000
            ABSOLUTE              F                 0MM/MIN
X          150.000                          PARTS COUNT    953
Y          100.000                RUN TIME     3H 2M CYCLE TIME  0H 0M 1S
Z           50.000                           SPINDLE TUNING
B            0.000                OPERATION    :SPEED CONTROL
C            0.000                GEAR SELECT  :1
                                  SPINDLE      :S11
            MODAL                    (PARAMETER)          (MONITOR)
G00  G49  G64   F        M        PROP.GAIN        10 MOTOR          1999
G17  G80  G69   H                 INT.GAIN         10 SPINDLE        2000
G90  G98  G15   D                 MOTOR VOLT       30
G23  G50  G40.1          T        REGEN.PW         90
G94  G67  G25   S   2000
G21  G97  G160
G40  G54  G13.1
SACT             2000/MIN         A>_
                                  MDI  STOP *** ***   11:01:33
```

Fig. 12.17.2 (a) Spindle tuning screen

12.17.3 Spindle Monitor Screen

This screen summarizes spindle information.

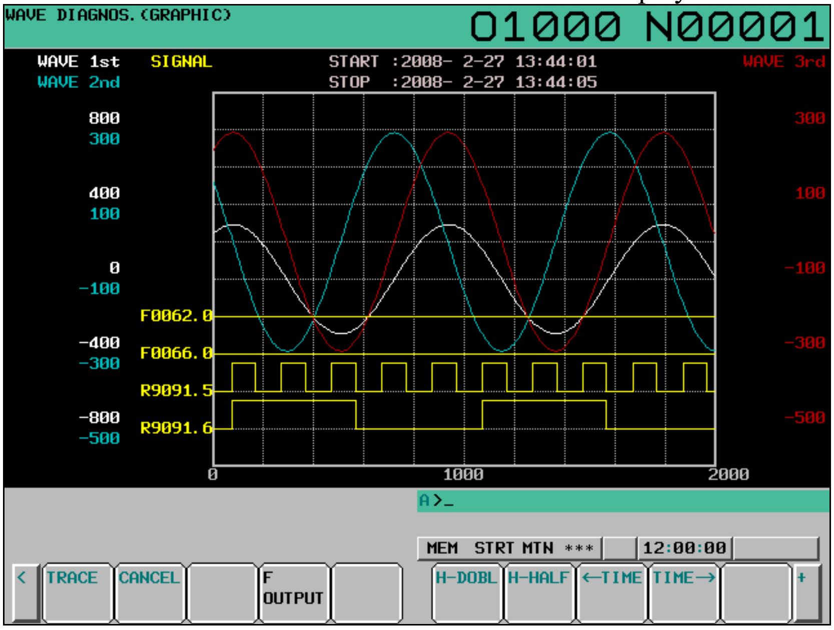


```
PROGRAM                                   O0000 N00000
         ABSOLUTE              F                   0MM/MIN
X        150.000                      PARTS COUNT      953
Y        100.000               RUN TIME    3H 2M CYCLE TIME  0H 0M 1S
Z         50.000                        SPINDLE MONITOR
B          0.000               ALARM          :
C          0.000               OPERATION      :SPEED CONTROL
                               SPINDLE SPEED  :     2000 RPM
                               MOTOR SPEED    :     2000 RPM
                               SPINDLE        :S1
            MODAL                              0   50  100  150  200(%)
G00  G49  G64  F         M     LOAD METER
G17  G80  G69  H                 3%
G90  G98  G15  D               CONTROL INPUT  :SFR  MRDY  *ESP
G23  G50  G40.1          T
G94  G67  G25  S    2000       CONTROL OUTPUT:SAR
G21  G97  G160
G40  G54  G13.1

SACT          2000/MIN         A>_
                               MDI  STOP *** ***    11:01:51
```

Fig. 12.17.3 (a) Spindle monitor screen

12.18 SERVO WAVEFORM DISPLAY

Servo data (including the error amount, torque amount, and the number of distributed pulses) and the waveforms of signals transferred between the CNC and PMC can be displayed.



```
WAVE DIAGNOS.(GRAPHIC)                    O1000 N00001
WAVE 1st   SIGNAL        START :2008- 2-27 13:44:01      WAVE 3rd
WAVE 2nd                 STOP  :2008- 2-27 13:44:05
A>_
                               MEM  STRT MTN ***    12:00:00
```

Fig. 12.18 (a) Waveform diagnosis screen

12.19 MAINTENANCE INFORMATION SCREEN

On the maintenance information screen, a history of maintenance operations can be recorded.
The following operations can be performed:

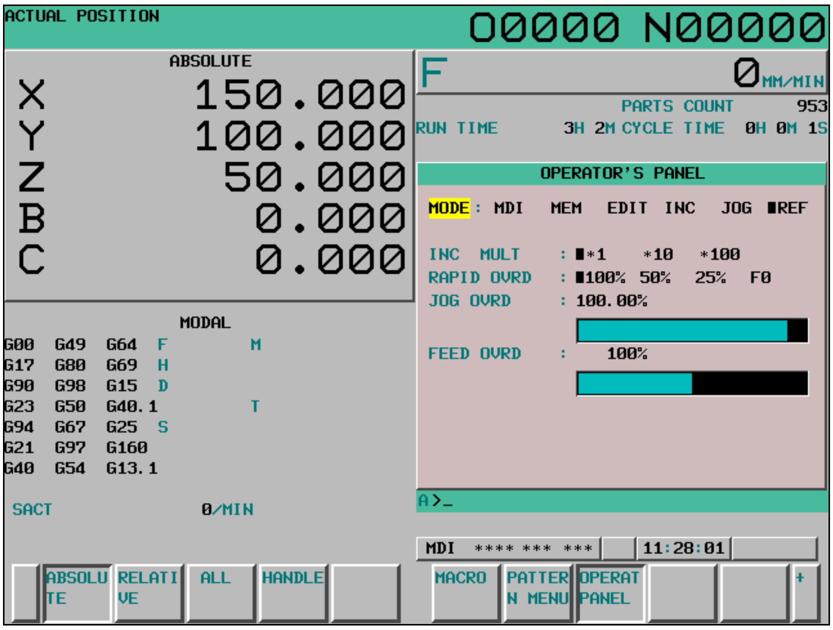
- Input of alphabetical characters by MDI (En-size kana characters can be input only in Japanese display mode.)
- Reference of the record screen by line-by-line scroll
- Input/output of maintenance information after editing
- Write and save in FROM
- Display of em-size code (Shift-JIS) (the only input is F input.)

12.20 SOFTWARE OPERATOR'S PANEL

The functions of switches on the machine operator's panel are made available by using the MDI panel. This means that functions such as a mode selection and selection of jog feedrate override can be made by operating the MDI panel, so corresponding switches on the machine operator's panel need not be used.
This function is enabled only when the software operator's panel screen is displayed. The user can make selection operations by moving the cursor by using the cursor keys while checking the screen. The following functions can be performed:

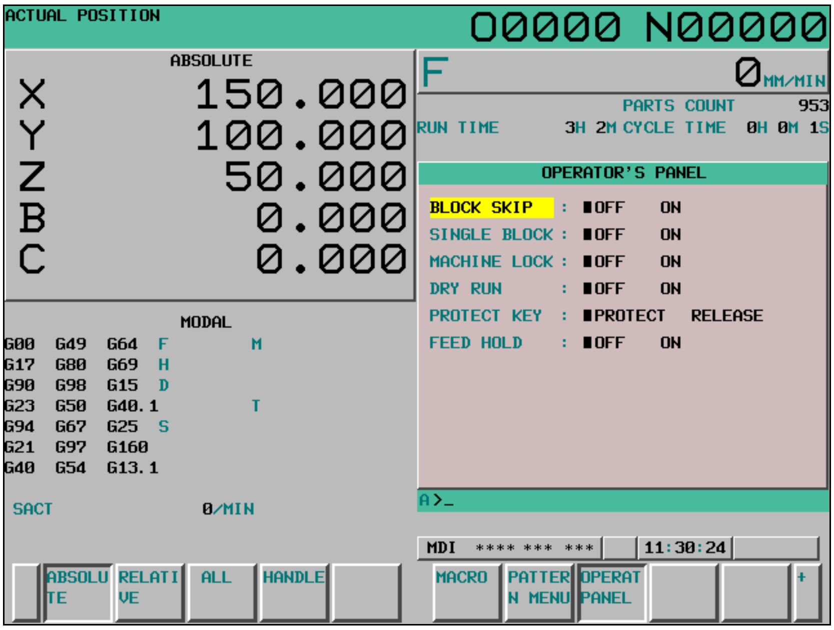
A : Mode selection
B : Selection of jog feed axis, manual rapid traverse
C : Selection of manual pulse generator feed axis, selection of manual pulse magnification
D : Jog federate, federate override, rapid traverse override
E : Optional block skip, single block, machine lock, dry run
F : Protect key
G : Feed hold
H : General purpose switch

A parameter that enables and disables MDI operations is provided for each of operation groups A to G indicated above.



```
ACTUAL POSITION                     O0000 N00000
            ABSOLUTE             F              0 MM/MIN
X         150.000                     PARTS COUNT      953
Y         100.000                RUN TIME     3H 2M CYCLE TIME  0H 0M 1S
Z          50.000                         OPERATOR'S PANEL
B           0.000                MODE : MDI  MEM  EDIT  INC  JOG  ■REF
C           0.000
                                 INC  MULT   : ■*1   *10   *100
            MODAL                RAPID OVRD  : ■100%  50%  25%  F0
G00  G49  G64   F     M          JOG OVRD    : 100.00%
G17  G80  G69   H
G90  G98  G15   D                FEED OVRD   :   100%
G23  G50  G40.1       T
G94  G67  G25   S
G21  G97  G160
G40  G54  G13.1
                                 A>_
SACT          0/MIN
                                 MDI **** *** ***     11:28:01
```

Fig. 12.20 (a) Software operator's panel screen (1/2)



BLOCK SKIP : ▮OFF ON
SINGLE BLOCK : ▮OFF ON
MACHINE LOCK : ▮OFF ON
DRY RUN : ▮OFF ON
PROTECT KEY : ▮PROTECT RELEASE
FEED HOLD : ▮OFF ON

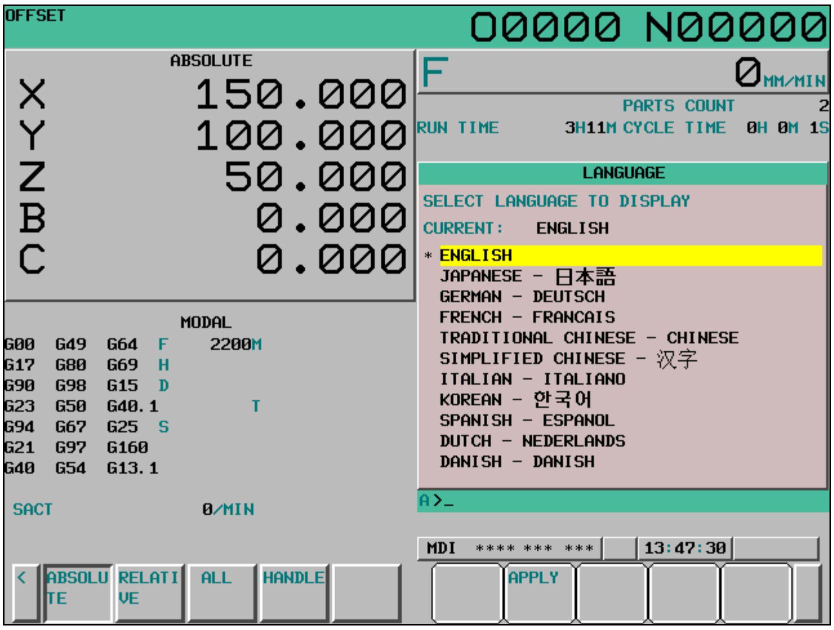
Fig. 12.20 (b) Software operator's panel screen (2/2)

12.21 SOFTWARE OPERATOR'S PANEL GENERAL PURPOSE SWITCH

This function allows the user to assign arbitrary signals to general purpose switches 1 to 16 on the software operator's panel.

12.22 MULTI-LANGUAGE DISPLAY

The following 18 languages are prepared as the display languages. English, Japanese, German, French, Spanish, Italian, Chinese (traditional), Chinese (simplified), Korean, Portuguese, Dutch, Danish, Swedish, Hungarian, Czech, Polish, Russian, and Turkish.
When a desired language is selected and set on the language screen, the display language can be changed immediately.



SELECT LANGUAGE TO DISPLAY
CURRENT: ENGLISH
* ENGLISH
JAPANESE - 日本語
GERMAN - DEUTSCH
FRENCH - FRANCAIS
TRADITIONAL CHINESE - CHINESE
SIMPLIFIED CHINESE - 汉字
ITALIAN - ITALIANO
KOREAN - 한국어
SPANISH - ESPANOL
DUTCH - NEDERLANDS
DANISH - DANISH

Fig. 12.22 (a) Language screen

12.23 DATA PROTECTION KEY

To protect various types of data in the CNC, data protection keys can be provided for the machine. Depending on the type of the data to be protected, the following four input signals are provided:

- KEY1
  Enables input of tool offset values and workpiece origin offset values.
- KEY2
  Enables input of settings.
- KEY3
  Enables program registration and editing.
- KEY4
  Enables input of PMC data.

12.24 PROTECTION OF DATA AT EIGHT LEVELS

Eight operation levels can be set for CNC and PMC operations, and eight protect levels can be set for various types of CNC and PMC data.

When CNC and PMC data is modified or output externally, the operation level and protect level are compared to determine whether to allow the modification or external output.

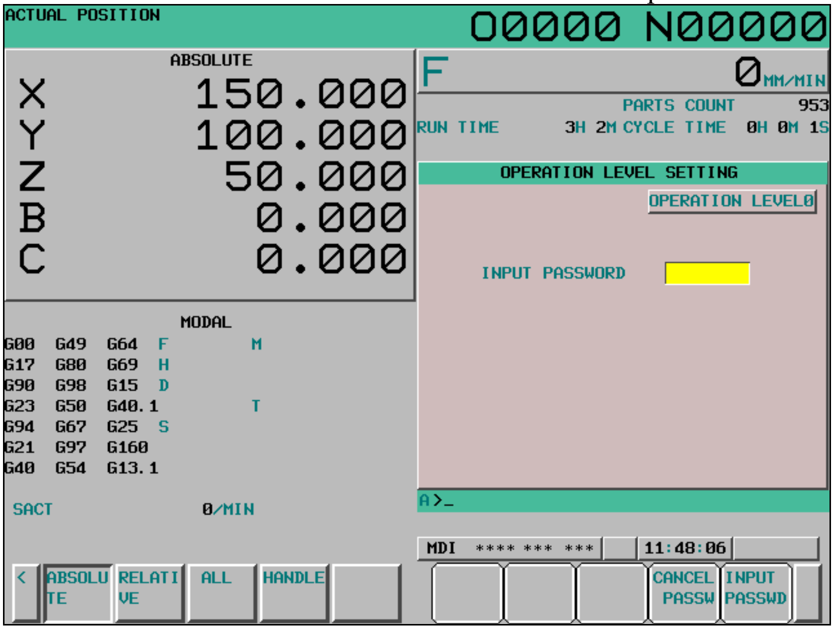
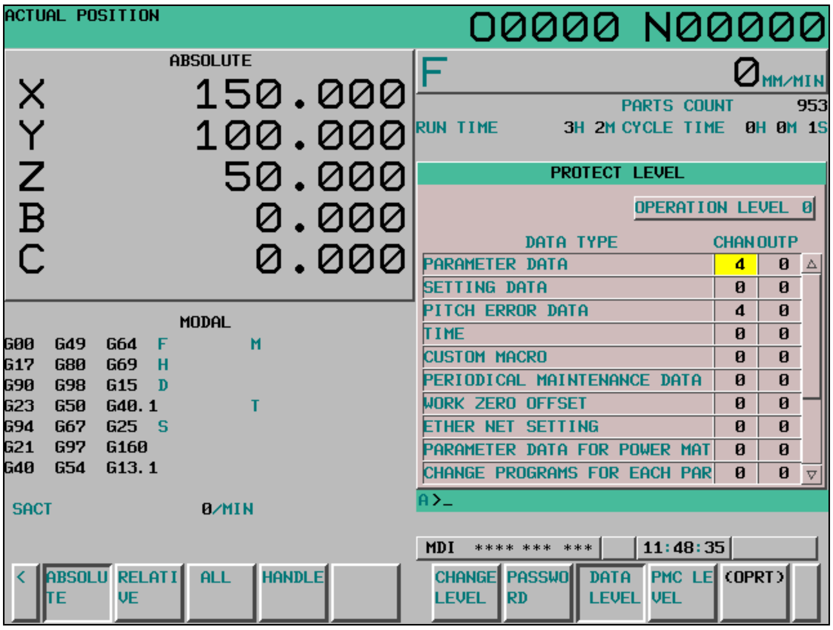


Fig. 12.24 (a) Operation level setting screen



```
X   150.000
Y   100.000
Z    50.000
B     0.000
C     0.000
```

F 0 MM/MIN
PARTS COUNT 953
RUN TIME 3H 2M CYCLE TIME 0H 0M 1S

OPERATION LEVEL 0

DATA TYPE	CHAN	OUTP
PARAMETER DATA	4	0
SETTING DATA	0	0
PITCH ERROR DATA	4	0
TIME	0	0
CUSTOM MACRO	0	0
PERIODICAL MAINTENANCE DATA	0	0
WORK ZERO OFFSET	0	0
ETHER NET SETTING	0	0
PARAMETER DATA FOR POWER MAT	0	0
CHANGE PROGRAMS FOR EACH PAR	0	0

```
G00  G49  G64   F    M
G17  G80  G69   H
G90  G98  G15   D
G23  G50  G40.1      T
G94  G67  G25   S
G21  G97  G160
G40  G54  G13.1
```

SACT 0/MIN

MDI **** *** *** 11:48:35

Fig. 12.24 (b) Protect level setting screen

12.25 ERASE CRT SCREEN DISPLAY

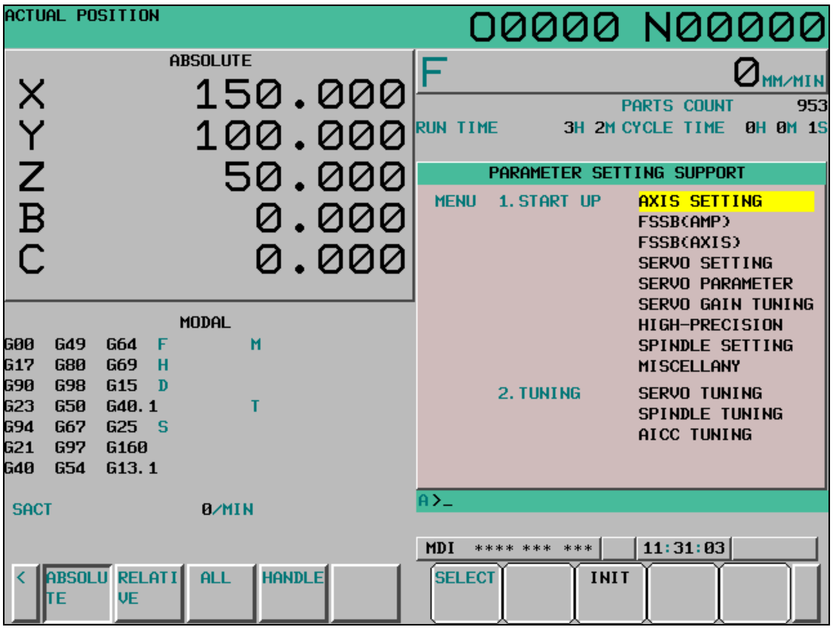
If the same characters are left displayed in the same position of the screen for an extended time, the service life of the display will be shortened. To prevent this, the CNC screen display can be erased.
The screen display can be erased by pressing keys (pressing the CAN key and the function key at the same time) (screen erasure). The screen display can also be erased automatically when no key operation has been performed for a parameter-set time (automatic screen erasure).

12.26 PARAMETER SETTING SUPPORT SCREEN

The parameter setting support screen allows parameter setting and tuning for the following purposes:
1 Displaying a list of the least required parameters for machine startup so that the machine can be started easily
2 Displaying a servo tuning screen, spindle tuning screen, and machining parameter tuning screen to allow smooth tuning

On this screen, the following items can be set and tuned:
Support of startup
- Axis setting
- FSSB (amplifier)
- FSSB (axis)
- Servo setting
- Servo parameter
- Servo gain tuning
- High-precision setting
- Spindle setting
- Miscellany

Support of tuning
- Servo tuning
- Spindle tuning
- AICC tuning

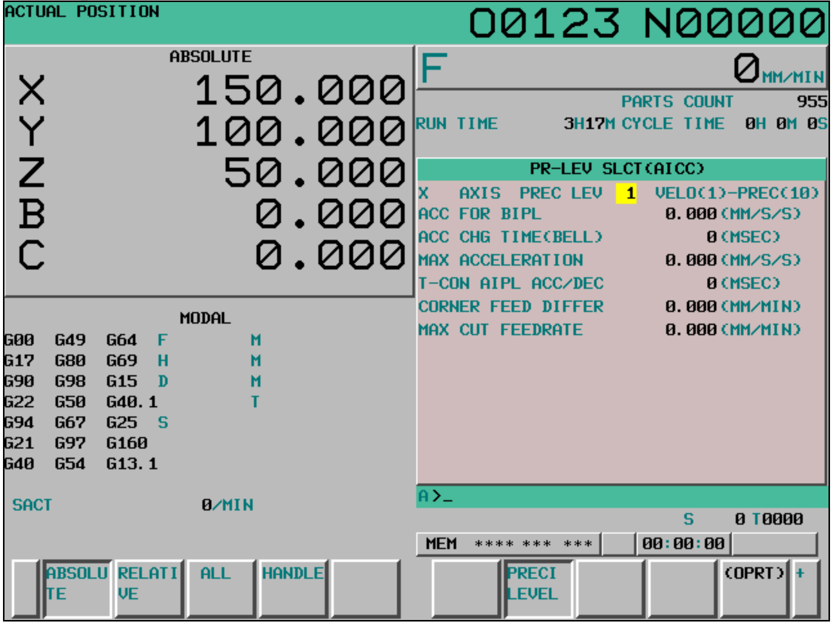


```
ACTUAL POSITION                    O0000 N00000
          ABSOLUTE            F              0MM/MIN
X    150.000                          PARTS COUNT    953
Y    100.000                  RUN TIME    3H 2M CYCLE TIME  0H 0M 1S
Z     50.000                      PARAMETER SETTING SUPPORT
B      0.000                  MENU  1.START UP   AXIS SETTING
C      0.000                                     FSSB(AMP)
                                                 FSSB(AXIS)
           MODAL                                 SERVO SETTING
G00  G49  G64  F       M                         SERVO PARAMETER
G17  G80  G69  H                                 SERVO GAIN TUNING
G90  G98  G15  D                                 HIGH-PRECISION
G23  G50  G40.1        T                         SPINDLE SETTING
G94  G67  G25  S                                 MISCELLANY
G21  G97  G160                      2.TUNING     SERVO TUNING
G40  G54  G13.1                                  SPINDLE TUNING
                                                 AICC TUNING
SACT             0/MIN        A>_
                              MDI **** *** ***    11:31:03
```

Fig. 12.26 (a) Menu screen for parameter setting support

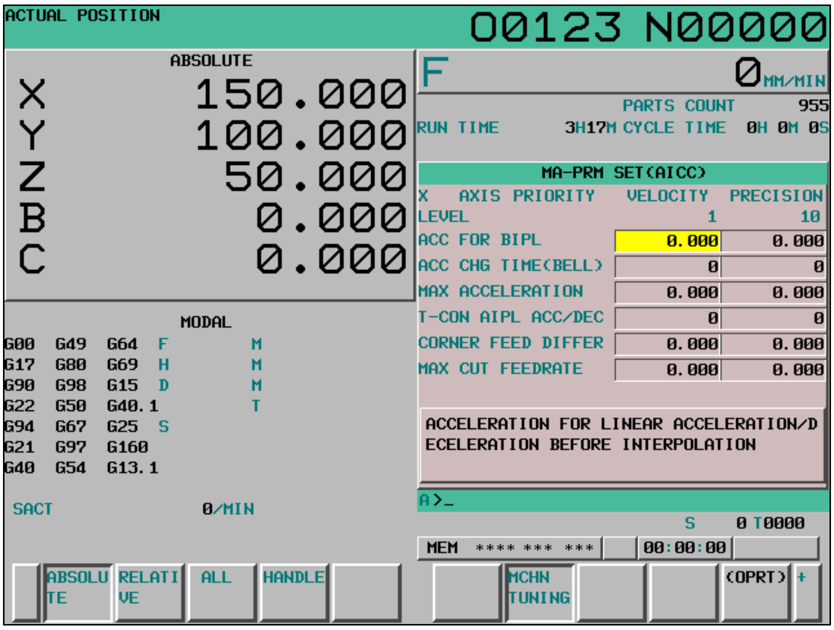
12.27 MACHINING CONDITION SELECTING FUNCTION

In advanced preview control, AI advanced preview control, or AI contour control, when speed-oriented and precision-oriented parameter sets are set, and a precision level is set according to the machining condition such as rough machining or finish machining on the precision level selection screen or with a programmed command, parameters that meet the condition can be calculated automatically for machining.



```
ACTUAL POSITION                    O0123 N00000
          ABSOLUTE            F              0MM/MIN
X    150.000                          PARTS COUNT    955
Y    100.000                  RUN TIME    3H17M CYCLE TIME  0H 0M 0S
Z     50.000                         PR-LEV SLCT(AICC)
B      0.000                  X   AXIS  PREC LEV  1  VELO(1)-PREC(10)
C      0.000                  ACC FOR BIPL          0.000(MM/S/S)
                              ACC CHG TIME(BELL)        0(MSEC)
           MODAL              MAX ACCELERATION      0.000(MM/S/S)
G00  G49  G64  F       M      T-CON AIPL ACC/DEC        0(MSEC)
G17  G80  G69  H       M      CORNER FEED DIFFER    0.000(MM/MIN)
G90  G98  G15  D       M      MAX CUT FEEDRATE      0.000(MM/MIN)
G22  G50  G40.1        T
G94  G67  G25  S
G21  G97  G160
G40  G54  G13.1
SACT             0/MIN        A>_
                                          S     0 T0000
                              MEM **** *** ***    00:00:00
```

Fig. 12.27 (a) Precision level selection screen



```
ACTUAL POSITION                      O0123 N00000
        ABSOLUTE           F                  0MM/MIN
X        150.000                  PARTS COUNT      955
Y        100.000           RUN TIME   3H17M CYCLE TIME  0H 0M 0S
Z         50.000                 MA-PRM SET(AICC)
B          0.000           X   AXIS PRIORITY  VELOCITY  PRECISION
C          0.000           LEVEL                     1         10
                           ACC FOR BIPL          0.000      0.000
         MODAL             ACC CHG TIME(BELL)        0          0
G00  G49  G64   F     M    MAX ACCELERATION      0.000      0.000
G17  G80  G69   H     M    T-CON AIPL ACC/DEC        0          0
G90  G98  G15   D     M    CORNER FEED DIFFER    0.000      0.000
G22  G50  G40.1       T    MAX CUT FEEDRATE      0.000      0.000
G94  G67  G25   S
G21  G97  G160             ACCELERATION FOR LINEAR ACCELERATION/D
G40  G54  G13.1            ECELERATION BEFORE INTERPOLATION

SACT            0/MIN      A>_
                                               S     0 T0000
                           MEM  **** *** ***  |00:00:00|
```

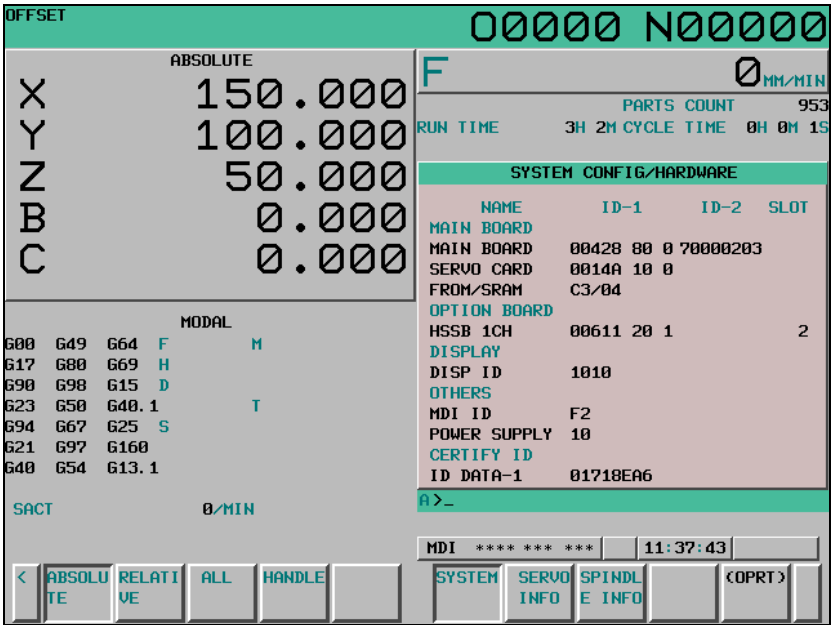
Fig. 12.27 (b) Machining parameter setting screen

12.28 SYSTEM CONFIGURATION SCREEN

This screen displays the necessary hardware and software configuration for CNC maintenance.
The system configuration screen shows the following:

- Hardware Configuration Screen
- Software Configuration Screen

12.28.1 Hardware Configuration Screen



```
OFFSET                               O0000 N00000
        ABSOLUTE           F                  0MM/MIN
X        150.000                  PARTS COUNT      953
Y        100.000           RUN TIME   3H 2M CYCLE TIME  0H 0M 1S
Z         50.000               SYSTEM CONFIG/HARDWARE
B          0.000              NAME         ID-1      ID-2   SLOT
C          0.000            MAIN BOARD
                            MAIN BOARD    00428 80 0 70000203
         MODAL              SERVO CARD    0014A 10 0
G00  G49  G64   F     M     FROM/SRAM     C3/04
G17  G80  G69   H           OPTION BOARD
G90  G98  G15   D           HSSB 1CH      00611 20 1            2
G23  G50  G40.1       T     DISPLAY
G94  G67  G25   S           DISP ID       1010
G21  G97  G160              OTHERS
G40  G54  G13.1             MDI ID        F2
                            POWER SUPPLY  10
                            CERTIFY ID
                            ID DATA-1     01718EA6
SACT            0/MIN      A>_

                           MDI  **** *** ***  |11:37:43|
```

Fig. 12.28.1 (a) Hardware configuration screen

1. NAME
   MAIN BOARD
   - Displays information on the main board and information on the cards on the main board.

OPTION BOARD

- Displays information on the board installed in the option slot.

DISPLAY

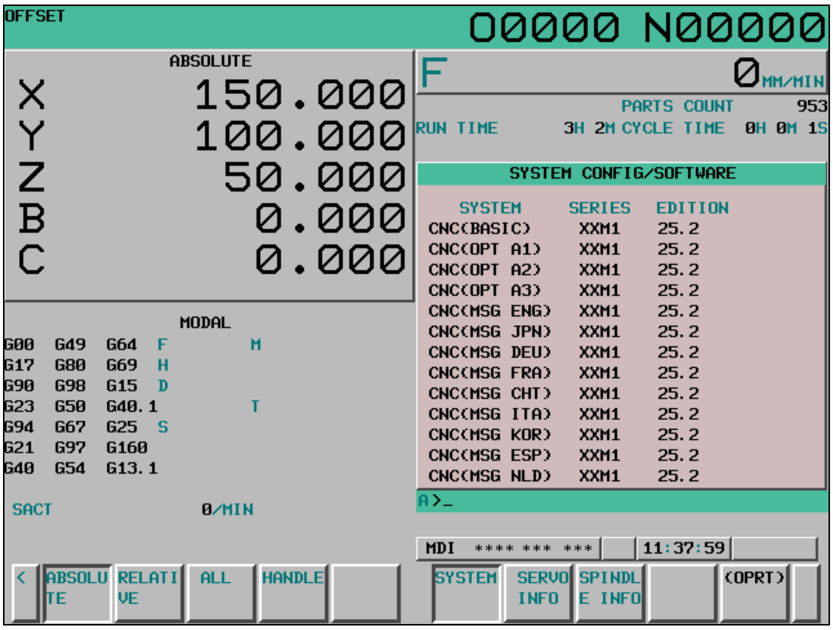
- Displays information on the display unit.

OTHERS

- Displays information on other components (such as an MDI and a power supply unit).

2. ID-1 / ID-2
   - Displays ID information.
3. SLOT
   - Displays the number of the slot in which the option board is inserted.

12.28.2 Software Configuration Screen



SYSTEM CONFIG/SOFTWARE

SYSTEM	SERIES	EDITION
CNC(BASIC)	XXM1	25.2
CNC(OPT A1)	XXM1	25.2
CNC(OPT A2)	XXM1	25.2
CNC(OPT A3)	XXM1	25.2
CNC(MSG ENG)	XXM1	25.2
CNC(MSG JPN)	XXM1	25.2
CNC(MSG DEU)	XXM1	25.2
CNC(MSG FRA)	XXM1	25.2
CNC(MSG CHT)	XXM1	25.2
CNC(MSG ITA)	XXM1	25.2
CNC(MSG KOR)	XXM1	25.2
CNC(MSG ESP)	XXM1	25.2
CNC(MSG NLD)	XXM1	25.2

Fig. 12.28.2 (a) Software configuration screen

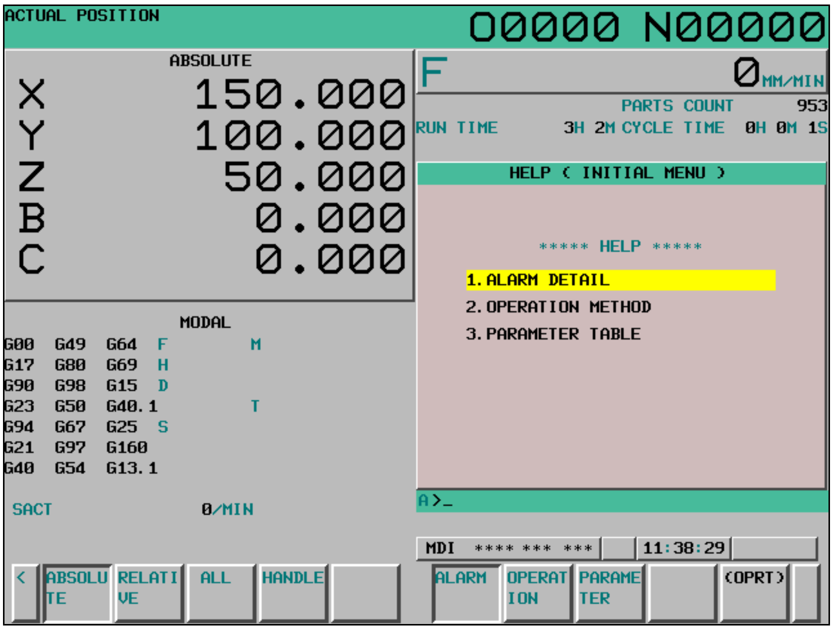
1. SYSTEM
   Software type
2. SERIES
   Software series
3. EDITION
   Software edition

12.29 HELP SCREEN

Detailed information about alarms issued in the CNC and how to operate the CNC is displayed on the screen. The initial menu screen, alarm detail screen, operation method screen, and parameter table screen are provided.

12.29.1 Initial Menu Screen

This screen shows a list of help functions. From the initial menu screen, you can display the alarm detail screen, operation method screen, and parameter table screen.

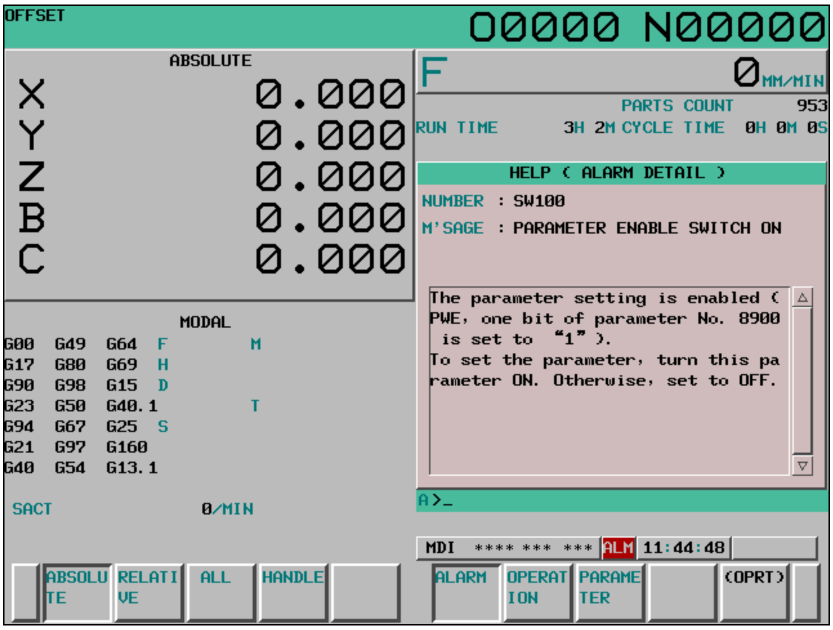


```
ACTUAL POSITION                          O0000 N00000
            ABSOLUTE                 F              0MM/MIN
X          150.000                       PARTS COUNT      953
Y          100.000                 RUN TIME   3H 2M CYCLE TIME  0H 0M 1S
Z           50.000                        HELP ( INITIAL MENU )
B            0.000
C            0.000                         ***** HELP *****
                                      1.ALARM DETAIL
              MODAL                   2.OPERATION METHOD
G00  G49  G64   F         M           3.PARAMETER TABLE
G17  G80  G69   H
G90  G98  G15   D
G23  G50  G40.1           T
G94  G67  G25   S
G21  G97  G160
G40  G54  G13.1
SACT               0/MIN               A>_
                                    MDI  **** *** ***    11:38:29
```

Fig. 12.29.1 Initial Menu Screen

12.29.2 Alarm Detail Screen

This screen provides detailed information about alarms as reference information. The causes of alarms and the methods of releasing them are displayed.

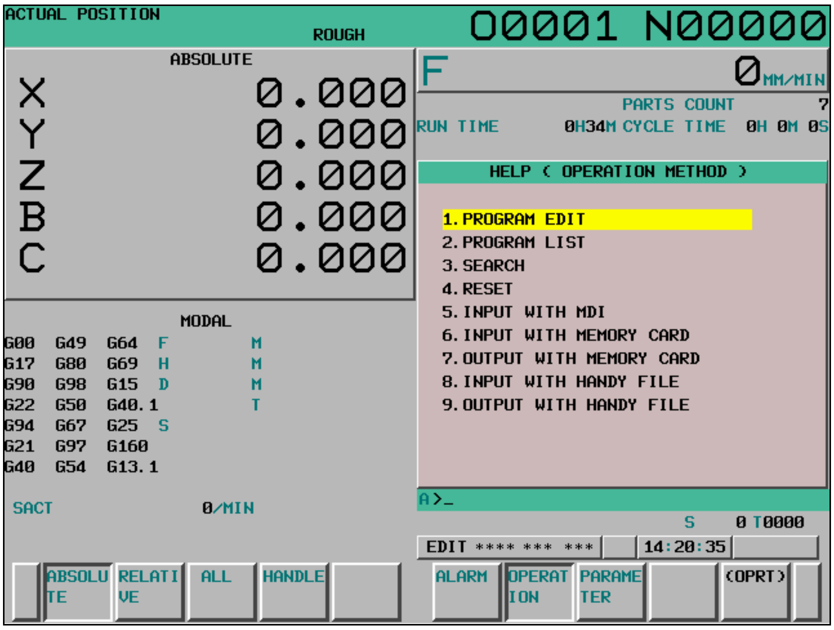


```
OFFSET                                   O0000 N00000
            ABSOLUTE                 F              0MM/MIN
X            0.000                       PARTS COUNT      953
Y            0.000                 RUN TIME   3H 2M CYCLE TIME  0H 0M 0S
Z            0.000                        HELP ( ALARM DETAIL )
B            0.000                 NUMBER  : SW100
C            0.000                 M'SAGE  : PARAMETER ENABLE SWITCH ON

              MODAL                 The parameter setting is enabled (
G00  G49  G64   F         M        PWE, one bit of parameter No. 8900
G17  G80  G69   H                   is set to  "1" ).
G90  G98  G15   D                  To set the parameter, turn this pa
G23  G50  G40.1           T        rameter ON. Otherwise, set to OFF.
G94  G67  G25   S
G21  G97  G160
G40  G54  G13.1
SACT               0/MIN               A>_
                                    MDI  **** *** *** ALM 11:44:48
```

Fig. 12.29.2 Alarm detail screen

12.29.3 Operation Method Screen

This screen provides how to operate the CNC.
On the operation method screen, the contents of operation items are listed.

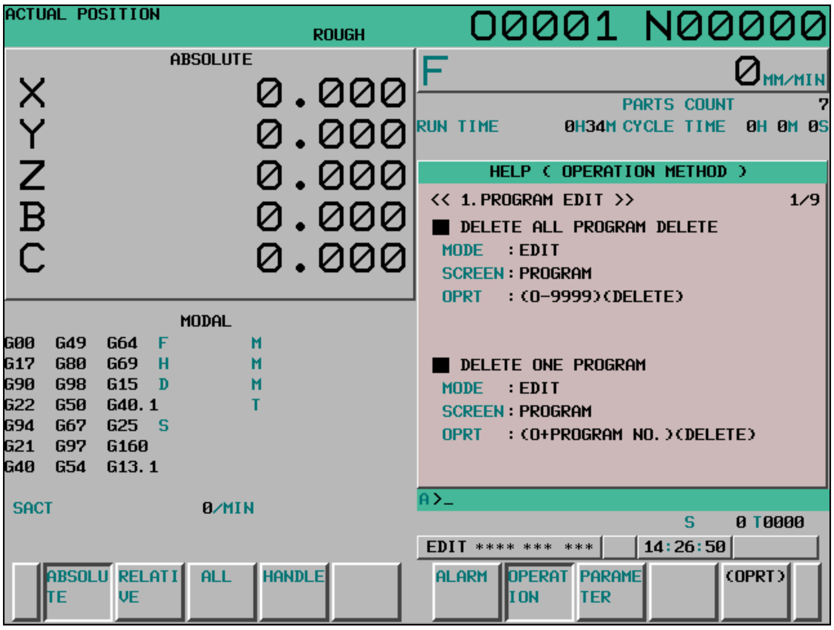


HELP (OPERATION METHOD)

1. PROGRAM EDIT
2. PROGRAM LIST
3. SEARCH
4. RESET
5. INPUT WITH MDI
6. INPUT WITH MEMORY CARD
7. OUTPUT WITH MEMORY CARD
8. INPUT WITH HANDY FILE
9. OUTPUT WITH HANDY FILE

Fig. 12.29.3 (a) Operation method screen

The operation method for an item selected on the operation method screen is displayed.



HELP (OPERATION METHOD)

<< 1. PROGRAM EDIT >> 1/9

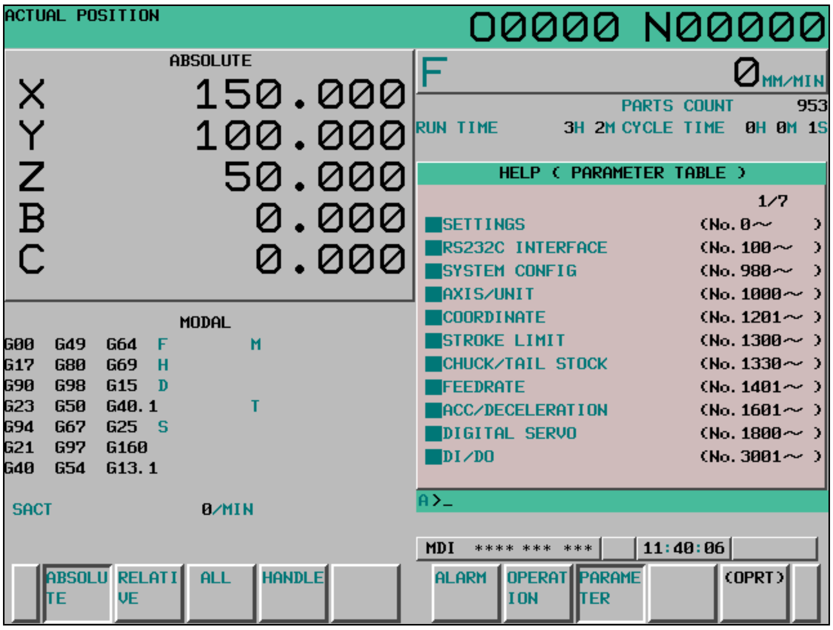
■ DELETE ALL PROGRAM DELETE
MODE : EDIT
SCREEN : PROGRAM
OPRT : (O-9999)(DELETE)

■ DELETE ONE PROGRAM
MODE : EDIT
SCREEN : PROGRAM
OPRT : (O+PROGRAM NO.)(DELETE)

Fig. 12.29.3 (b) Operation information screen

12.29.4 Parameter Table Screen

This screen is used to find the number of a system parameter to be set or referenced.
On the parameter table screen, the parameter numbers are listed for each function.



X 150.000
Y 100.000
Z 50.000
B 0.000
C 0.000

F 0 MM/MIN
PARTS COUNT 953
RUN TIME 3H 2M CYCLE TIME 0H 0M 1S

HELP (PARAMETER TABLE)

1/7

SETTINGS	(No. 0～)
RS232C INTERFACE	(No. 100～)
SYSTEM CONFIG	(No. 980～)
AXIS/UNIT	(No. 1000～)
COORDINATE	(No. 1201～)
STROKE LIMIT	(No. 1300～)
CHUCK/TAIL STOCK	(No. 1330～)
FEEDRATE	(No. 1401～)
ACC/DECELERATION	(No. 1601～)
DIGITAL SERVO	(No. 1800～)
DI/DO	(No. 3001～)

G00 G49 G64 F M
G17 G80 G69 H
G90 G98 G15 D
G23 G50 G40.1 T
G94 G67 G25 S
G21 G97 G160
G40 G54 G13.1

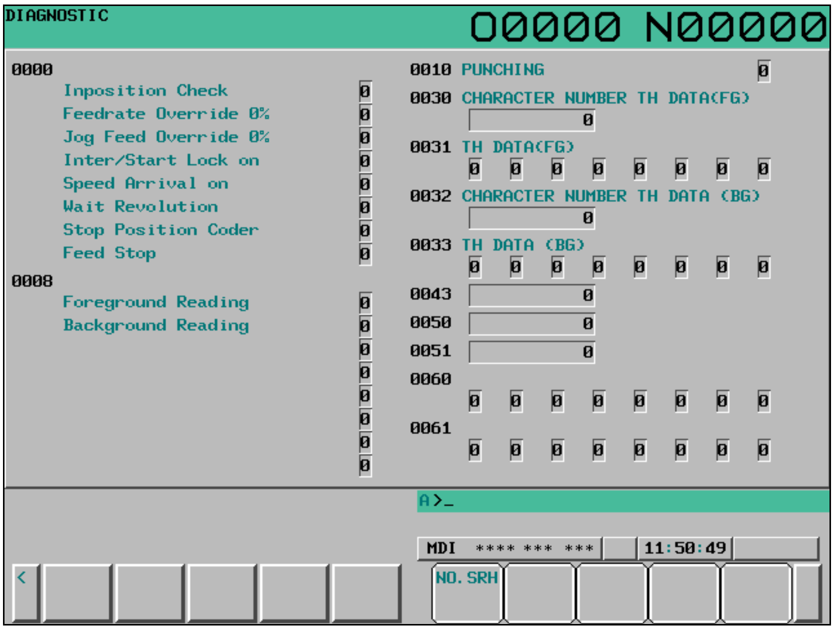
SACT 0/MIN

A>_

Fig. 12.29.4 Parameter table screen

12.30 SELF-DIAGNOSIS SCREEN

The current internal status of the system is indicated.



0000
Inposition Check 0
Feedrate Override 0% 0
Jog Feed Override 0% 0
Inter/Start Lock on 0
Speed Arrival on 0
Wait Revolution 0
Stop Position Coder 0
Feed Stop 0
0008
Foreground Reading 0
Background Reading 0
0
0
0
0
0
0
0

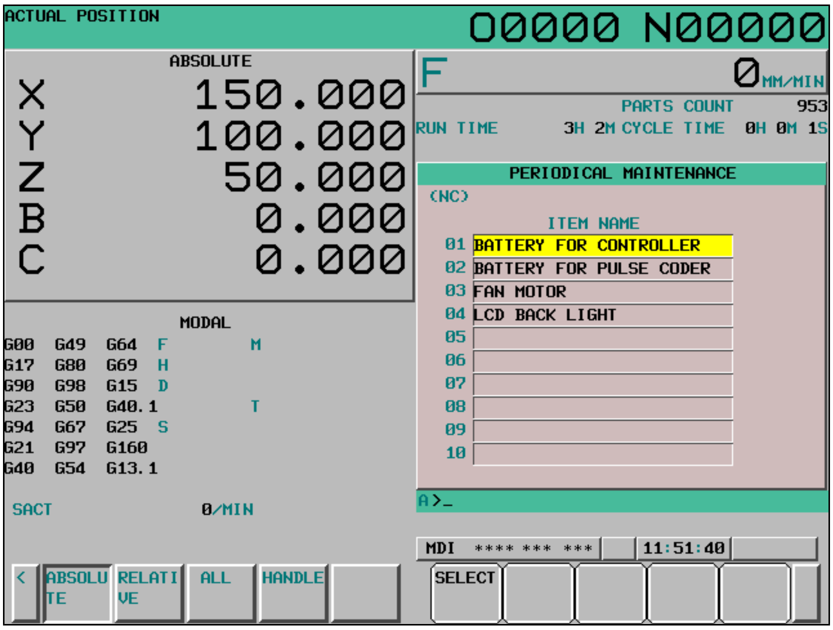
0010 PUNCHING 0
0030 CHARACTER NUMBER TH DATA(FG) 0
0031 TH DATA(FG) 0 0 0 0 0 0 0 0
0032 CHARACTER NUMBER TH DATA (BG) 0
0033 TH DATA (BG) 0 0 0 0 0 0 0 0
0043 0
0050 0
0051 0
0060 0 0 0 0 0 0 0 0
0061 0 0 0 0 0 0 0 0

A>_

Fig. 12.30 (a) Diagnosis screen

12.31 PERIODIC MAINTENANCE SCREEN

On the periodic maintenance screen, you can check whether the service lives of consumables that require periodic replacement (for example, the backup battery, the back light and touch panel of the LCD unit, etc.) have expired or not from displayed information such as the machine operation time.



```
ACTUAL POSITION                          O0000 N00000
           ABSOLUTE               F                  0MM/MIN
X         150.000                          PARTS COUNT     953
Y         100.000                 RUN TIME     3H 2M CYCLE TIME  0H 0M 1S
Z          50.000                       PERIODICAL MAINTENANCE
B           0.000                 <NC>
C           0.000                            ITEM NAME
                                   01 BATTERY FOR CONTROLLER
           MODAL                   02 BATTERY FOR PULSE CODER
G00  G49  G64   F        M         03 FAN MOTOR
G17  G80  G69   H                  04 LCD BACK LIGHT
G90  G98  G15   D                  05
G23  G50  G40.1          T         06
G94  G67  G25   S                  07
G21  G97  G160                     08
G40  G54  G13.1                    09
                                   10
SACT             0/MIN            A>_
                                  MDI **** *** ***   11:51:40
```

Fig. 12.31 (a) periodic maintenance screen

12.32 SERVO AND SPINDLE INFORMATION SCREENS

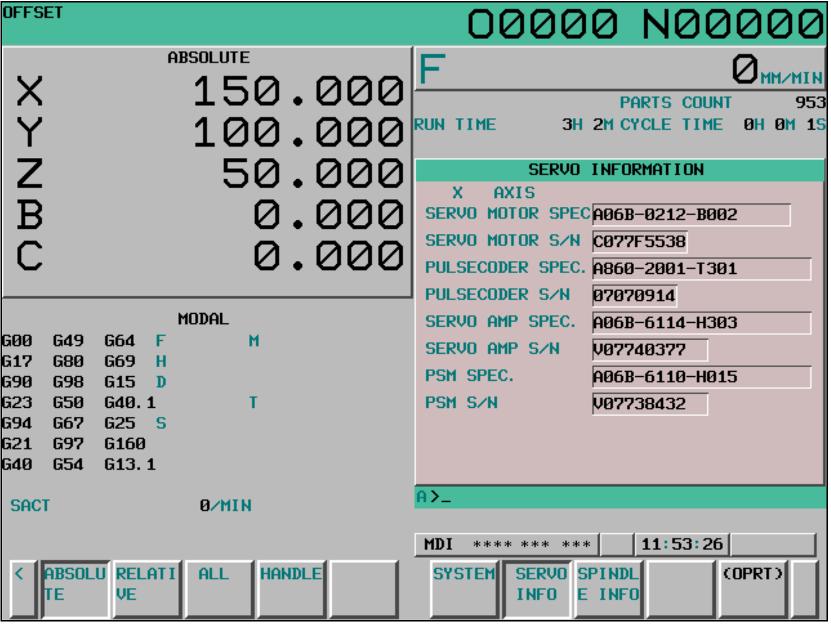
ID information owned by the connected units of servo and spindle systems is displayed on the CNC screen.

ID information is automatically read from each of the connected units during first startup of the CNC and then recorded. During the second or later startup, the ID information recorded during first startup can be compared with the ID information read this time on the screen to check whether the configuration of the connected units is changed. (If there is a difference between them, the alarm mark (*) appears.)

The recorded ID information can be edited. Therefore, the ID information of an unit that does not have ID information can be displayed. (However, the alarm mark (*) indicating a difference between these IDs appears.)

12.32.1 Servo Information Screen

- Displaying the servo information screen



SERVO INFORMATION

X AXIS

SERVO MOTOR SPEC A06B-0212-B002

SERVO MOTOR S/N C077F5538

PULSECODER SPEC. A860-2001-T301

PULSECODER S/N 07070914

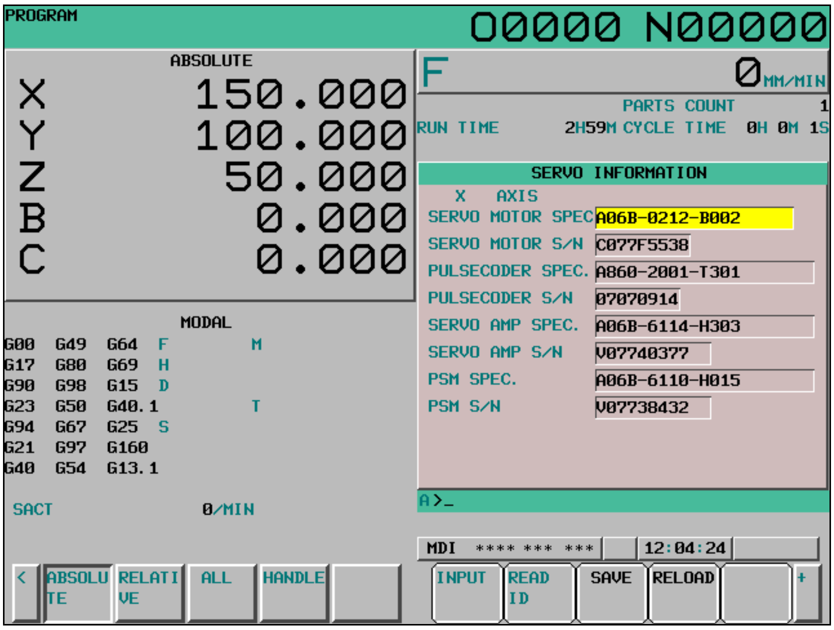
SERVO AMP SPEC. A06B-6114-H303

SERVO AMP S/N V07740377

PSM SPEC. A06B-6110-H015

PSM S/N V07738432

- Editing the servo information screen



SERVO INFORMATION

X AXIS

SERVO MOTOR SPEC A06B-0212-B002

SERVO MOTOR S/N C077F5538

PULSECODER SPEC. A860-2001-T301

PULSECODER S/N 07070914

SERVO AMP SPEC. A06B-6114-H303

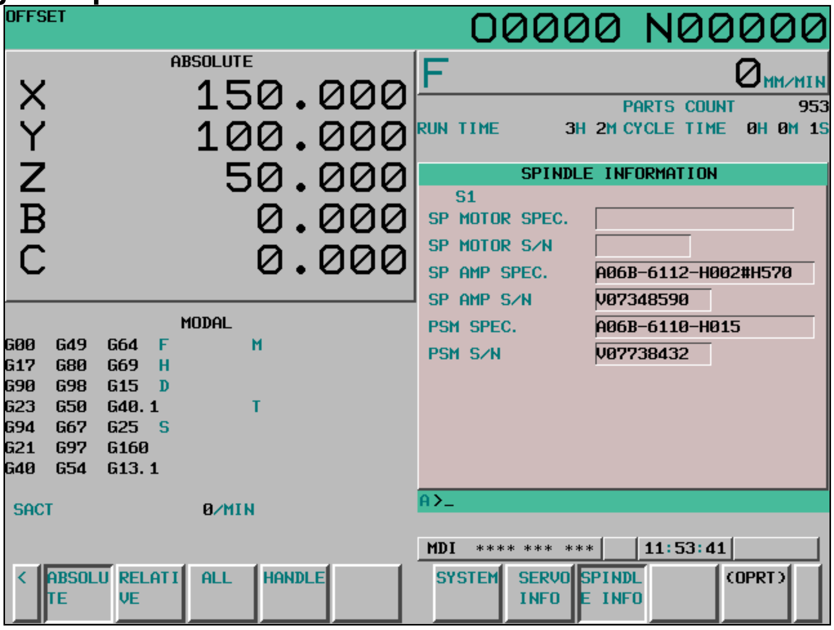
SERVO AMP S/N V07740377

PSM SPEC. A06B-6110-H015

PSM S/N V07738432

12.32.2 Spindle Information Screen

- Displaying the spindle information screen



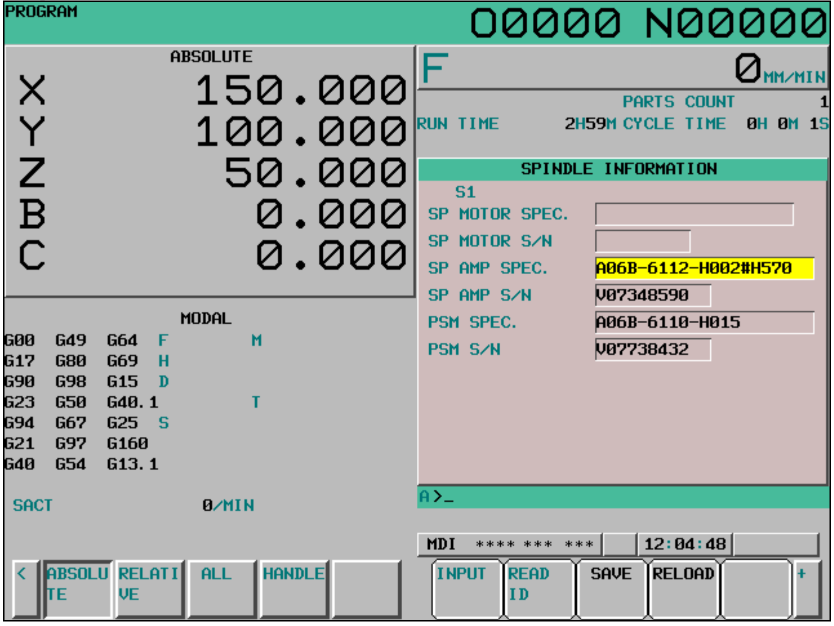
OFFSET O0000 N00000

ABSOLUTE
X 150.000
Y 100.000
Z 50.000
B 0.000
C 0.000

F 0 MM/MIN
PARTS COUNT 953
RUN TIME 3H 2M CYCLE TIME 0H 0M 1S

SPINDLE INFORMATION
S1
SP MOTOR SPEC.
SP MOTOR S/N
SP AMP SPEC. A06B-6112-H002#H570
SP AMP S/N V07348590
PSM SPEC. A06B-6110-H015
PSM S/N V07738432

- Editing the spindle information screen



PROGRAM O0000 N00000

ABSOLUTE
X 150.000
Y 100.000
Z 50.000
B 0.000
C 0.000

F 0 MM/MIN
PARTS COUNT 1
RUN TIME 2H59M CYCLE TIME 0H 0M 1S

SPINDLE INFORMATION
S1
SP MOTOR SPEC.
SP MOTOR S/N
SP AMP SPEC. A06B-6112-H002#H570
SP AMP S/N V07348590
PSM SPEC. A06B-6110-H015
PSM S/N V07738432

12.33 GRAPHIC DISPLAY

The tool path of a program during machining can be drawn. So, the progress of machining and the current tool position can be checked.
The following functions are available:

- The current tool position in the workpiece coordinate system is displayed.
- Graphic coordinates can be set freely.
- Rapid traverse and cutting feed can be drawn using a different color for each.
- The values of F, S, and T in the program during drawing are displayed.
- Graphic enlargement or reduction is possible.

12.34 DYNAMIC GRAPHIC DISPLAY

M

The dynamic graphic display function has two features:

- Path Drawing
  The path of coordinates specified in a program is drawn on the screen.
  By displaying a travel path on the screen, the path can be checked easily before performing machining actually.
- Animation
  The figure of a workpiece to be machined by a programmed tool movement is drawn.
  By drawing the three-dimensional figure of a workpiece to be machined in an animation-like way, the intermediate machining process and final figure can be grasped easily.

When machining is performed according to a program, this function can draw a tool path with another program.
This function performs drawing much faster than the graphic display function based on automatic operation, so that a program check can be made more swiftly.

T

The dynamic graphic display function can display the travel path of machining without operating the machine actually.
(It is not necessary to operate the machine actually as in the graphic display function, but path drawing needs to be performed with the conditions under which the machine can be operated actually provided, such as selection of the MEM mode with the mode switching switch on the machine operator's panel.)

12.35 TOUCH PANEL CONTROL

A display unit with a touch panel enables you to operate soft keys by touching the screen.

Moreover, an application using a touch panel can be created with the C language executor.

- Remarks

- RS-232C serial port 2 (JD36B) is occupied.
- Touch panel pressing information is read at intervals of 32 msec.
- A positional precision of ±2.5 mm is provided.

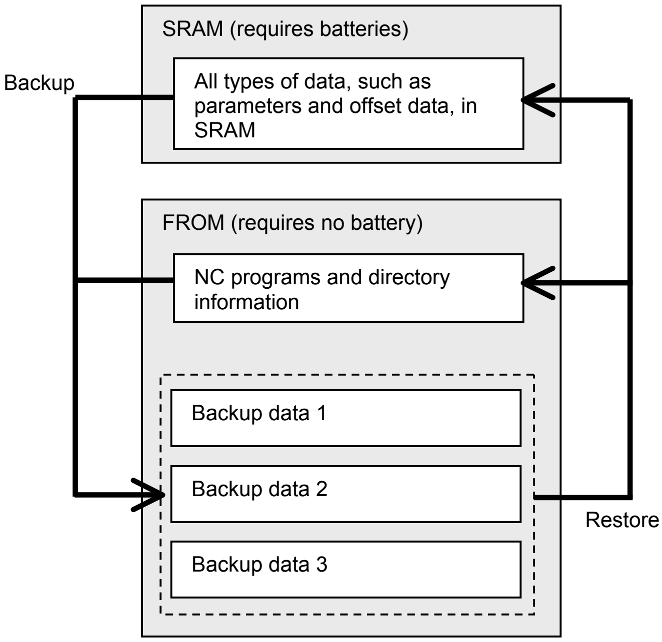
12.36 EXTERNAL TOUCH PANEL INTERFACE

External Touch Panel(called "ETP" below) of SNP-X protocol can be connected with CNC.
ETP has functions that can read out/ write in from/to PMC such control signals as input signal(X), output signal(Y), internal relay(R), keep relay(K), data table(D), extra relay (E), timer(T), counter(C), and the function is almost the same as operating panel of machine.

The remarkable function of ETP is drawing function.
Assignment between drawing and address(signal) can be specified freely.
For example, the data in data table can be set with the switch on the screen which is designed to assign the setting of data table.

12.37 AUTOMATIC DATA BACKUP

It is possible to back up data held in the CNC's FROM/SRAM by storing it automatically in the FROM, which requires no battery and to restore the baked-up data as required. If data is lost from the CNC due to unforeseen circumstances, this function can be used to restore the data easily.
Also, it is possible to hold up to three occurrences of backup data. With this function, the CNC data can be quickly switched to a post-machine adjustment state or an arbitrary backup state.



12.38 SPEED DISPLAY FUNCTION OF A MILLING TOOL WITH SERVO MOTOR

Any servo motor axis can be selected to display its speed considering gear ratio.

13 DATA INPUT/OUTPUT

Chapter 13, "DATA INPUT/OUTPUT", consists of the following sections:

13.1 READER/PUNCHER INTERFACE

The data shown below can be input/output through reader/puncher interface.

- Program
- Tool compensation value
- Parameter
- Pitch error compensation data
- Custom macro common variable
- Workpiece coordinate system setting data

Usually, the screen is switched according to the type of data to be input from or output to an external device; for example, a parameter screen is used for parameter input/output, and a program screen is used for program input/output. However, a single ALL I/O screen can be used to input and output programs, parameters, offset data, and macro variables.

13.2 FAST DATA SERVER

The fast data server performs machining by storing NC programs in the built-in ATA flash card.

- Machining programs can be transferred between the personal computer and data server.
- Memory operation using macro statements and sub programs called from the ATA flash card can be performed. DNC operation from the personal computer is also possible.
- Programs stored in the ATA flash card can be edited.

While operation is performed using the data server operation, other Ethernet functions can be used.

13.3 EXTERNAL DATA INPUT

The external data input function sends data to the CNC from an external unit such as a machine to perform desired operation.
The external data input function includes the following capabilities:

- External tool compensation
- External program number search
- External workpiece coordinate system shift
- External machine zero point shift

- Extended external machine zero point shift
- External alarm message
- External operator message
- Assignment of machined parts count and required parts count data

13.3.1 External Tool Offset

These signals provide for changing the tool compensation value via the PMC. When the offset number is specified by a part program, data input from the PMC is added to the offset value. The offset value can also be used as input data itself by specifying the input signal.
If the tool compensation value is externally input when offset number 0 is specified in a part program (an offset cancel) in the lathe turning machine, the workpiece coordinate system shifts by the entered quantity.
The external tool offset range is ±79999999.
However, the unit and setting range are the same as the Tool offset.

13.3.2 External Program Number Search

A program number (1 to 9999) is specified from an extended source and is selected in the CNC memory.
Data for the external program number search is accepted regardless of CNC mode, but the search execution can be made only in the reset state in MEM mode.

13.3.3 External Workpiece Coordinate System Shift

The external workpiece coordinate system shift function shifts the external workpiece coordinate system using a shift amount input externally.
Each axis has this shift value (setting the parameter), and this shift value is added to all the workpiece coordinate systems in common.
The shift value is not lost by cutting off the power supply.
The shift value can not only be added to the current work coordinate system shift but also be substituted for the current value.
The value range is 0 to ±79999999.
The unit and setting range are the same as the unit and setting range of the tool offset.

13.3.4 External Machine Zero Point Shift

The machine coordinate system can be externally shifted by inputting a shift value.
When the shift value is input, compensation is immediately applied to the corresponding axis and the axis moves. The position accuracy can be improved by combining this function with sensors.
The specification to shift the axis is the same as the external workpiece coordinate system shift.
The compensation value is specified in signals ED0 to ED31 using a binary code ranging from 0 to ±999999999. This compensation value must be specified in absolute value. The value which the machine actually moves at input is the difference from the previously stored value.
When a large value of compensation is applied at one time, an alarm such as "excessive error on stop" may occur. In this case, input the compensation in several smaller increments.

13.3.5 Extended External Machine Zero Point Shift

The conventional external machine zero point shift value function cannot make shifts on multiple axes simultaneously.
With this extended function, external machine zero point shifts can be performed on all controlled axes.
An external machine zero point shift value is to be set in a parameter-set R area. A shift value must be specified using a binary code, and the absolute value of a number from -32767 to 32767 must be specified.

13.3.6 External Alarm Message

By sending alarm number from PMC, the CNC is brought to an alarm status; an alarm message is sent to the CNC, and the message is displayed on the screen of the CNC.
Reset of alarm status is also done with external data.
Up to 4 alarm numbers and messages can be sent at a same time. Up to 32 characters can be sent in an alarm message.
Also the alarm is set separately for one.
It is available to select the range of alarm number and the display form by setting the parameter.

- When the parameter is set to 0.
  Alarm number 0 to 999 can be sent. To distinguish these alarms from other alarms, the CNC displays them by adding 1000 to an alarm number.
- When the parameter is set to 1.
  Alarm number 0 to 4095 can be sent. The CNC displays them with prefix characters “EX” to an alarm number for display.

13.3.7 External Operator Message

The external operator message function allows transfer of messages for the operator from the outside to the CNC to display them on the CNC screen.
The operator messages can be cleared by external data.
Up to 4 message numbers can be sent at a same time. Up to 256 characters can be sent in an operator message.
It is available to select the range of message number and the display form by setting the parameter.

- When the parameter is set to 0.
  The message numbers 0 to 999 can be sent.
  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 an alarm number.
  When a message from 100 to 999 is displayed, the message number is not displayed; only its text is displayed.
- When the parameter is set to 1.
  The message numbers 0 to 4095 can be sent.
  The message numbers 0 to 99 are displayed along with the message.
  The CNC displays them with prefix characters "EX" to an alarm for display.
  When a message number from 100 to 4095 is displayed, the message number is not displayed; only its text is displayed.

13.3.8 Assignment of Machined Parts Count and Required Parts Count

Each of the required parts count and machined parts count can be externally preset to a value in the range 0 to 9999.

13.4 EXTERNAL KEY INPUT (KEY INPUT FROM THE PMC)

By turning input signals from the PMC to CNC on and off, code signals corresponding to keys on the MDI panel are input from the PMC to CNC. This produces the same effect as when keys on the MDI panel are operated.
For example, the following application is found:
To store a machining position as a programmed command after moving the tool to that position by using the playback function, you had to type keys such as X, Y, Z, and <INSERT>. These key operations can be replaced by just one press of a switch on the machine operator's panel. Therefore, when the switch is

pressed, the PMC can input code signals corresponding to the keys such as X, Y, Z, and <INSERT> to the CNC to produce the same effect as when key operations take place.

13.5 EXTERNAL WORKPIECE NUMBER SEARCH

By specifying a workpiece number from 0011 to 9999 through an external unit such as a machine, the program corresponding to the workpiece number can be selected.
A workpiece number is used as a program number without modification. For example, when workpiece number 12 is specified, program number O0012 is selected.

13.6 MEMORY CARD INPUT/OUTPUT

By using the memory card interface located on the left side of the LCD display unit, information written on the memory card can be read into the CNC, or data can be written to the memory card.
The following types of data can be input/output:

- Program
- Offset data
- Parameter
- Pitch error compensation data
- Custom macro common variable
- Workpiece coordinate system setting data
- Operation history data and so on

If an attempt is made to use an existing file name when NC data, such as programs and parameters, are written to the memory card, it is possible to select whether to write the NC data to the existing file or cancel the attempt.
Whether to enable this function can be specified using the parameter.
For detailed explanations about how to output data, refer to the OPERATOR’S MANUAL (B-64304EN).

13.7 SCREEN HARD COPY

The information displayed on the screen of the display unit can be output as bit-mapped format data to the memory card. Note, however, that only freeze-screen information can be output.
Bit-mapped format data created with this function can be referenced using a device such as a personal computer.

13.8 POWER MATE CNC MANAGER

When the β amplifier (I/O Link option) is used as an additional (slave) axis of the CNC, the power mate CNC manager enables the display and setting of data from the CNC.
Up to eight slaves can be connected per I/O Link channel.
The power mate CNC manager supports the following functions:

- Current position display (absolute/machine coordinate)
- Parameter display, setting, I/O (memory card, program area on the CNC)
- Diagnosis display
- System configuration screen display
- Alarm display

13.9 ONE TOUCH MACRO CALL

By pressing a switch on the machine, the following three operations can be performed with minimum ladder modifications:

- Switching to the MEM mode
- Execution of macro programs stored in memory
- Return to the mode before execution. The program selected before execution is automatically selected.

This function is enabled only in the reset state. This means that this function cannot be used during automatic operation (during automatic operation halt and automatic operation stop periods as well).

13.10 EXTERNAL I/O DEVICE CONTROL

The registration or punch of a program can be specified externally.

- Registration
  External read start signal EXRD can be used to register a program in the part program memory from an external input device with the background edit function.
- Punch
  External punch start signal EXWT can be used to output all programs registered in part program memory to an external output device with the background edit function.

14 INTERFACE FUNCTION

Chapter 14, "INTERFACE FUNCTION", consists of the following sections:

14.1 EMBEDDED ETHERNET

The 100-Mbps Embedded Ethernet is supported as standard. By connecting the CNC with a personal computer, NC programs can be transferred, the machine can be controlled, machine operation status can be monitored, and the machine can be adjusted and maintained.

- NC program transfer using the FTP transfer function
  By CNC screen operations, NC programs can be transferred. The personal computer uses FTP server software, so that NC programs can be transferred to and from a host computer operating not in the Windows environment.

- Machine control and monitoring using FOCAS2/Ethernet
  Application software for controlling and monitoring the machine can be created by using FOCAS2/Ethernet function.

- Machine adjustment, maintenance ladder program maintenance, and servo motor adjustment using FANUC LADDER-III and SERVO GUIDE can be performed online.

14.2 FAST ETHERNET / FAST DATA SERVER

- Using FOCAS2/Ethernet in controlling and monitoring machines
  Application software for controlling and monitoring the machine can be created by using FOCAS2/Ethernet function.
  In addition, the unsolicited messaging function enables the CNC to send messages (CNC/PMC data) voluntarily to an application on the PC in response to a command from an NC program or a ladder program.
  Fast Ethernet uses a dedicated CPU on the option board to perform communication processing, so it enables high-speed and simultaneous data transfer to and from more than one personal computer.
  This function is suitable for building a production system that exchanges information with the machining lines and host computer of a plant.

- Data Server function
  A large program which can not be stored on the CNC memory can be stored on the built-in memory card of the Fast Data Server to allow high-speed machining. While the data server is operating, another Ethernet function can be used at the same time.

  High-speed transfer
    Machining programs can be transferred between a personal computer and the data server at a high speed. CNC parameters and tool information files can also be sent and received.

  Memory operation
    Memory operation that calls macro statements and subprograms from the built-in memory card of the Fast Data Server can be performed.

DNC operation

The DNC operation of an NC program stored on the built-in memory card of the Fast Data Server can be performed. In addition, the DNC operation can also be performed while NC programs are being FTP-transferred from the PC.

Program editing

NC programs stored on the built-in memory card of the Fast Data Server can be edited.

14.2.1 Functional differences between the embedded Ethernet function and the Ethernet function based on the option board

The table below indicates the differences between the embedded Ethernet function and the Ethernet function based on the option board.

	Embedded Ethernet	**Option board**
Data Server function	Not available	Available
FTP file transfer function	Available	Available
DNC operation	Not available	Available
FOCAS2/Ethernet function NOTE1	Available	Available
CNC screen display function	Not available	Available
Machine Remote diagnosis function	Not available	Available
CNC Unsolicited message function	Not available	Available
DNS/DHCP client function	Available	Available

NOTE

1 Compared with the option board, the embedded Ethernet function allows a smaller number of FOCAS2/Ethernet clients to be connected simultaneously.

	Embedded Ethernet	**Option board**
Number of clients that can be connected simultaneously	5 clients maximum	20 clients maximum
Number of personal computers that can be connected simultaneously	1 unit (recommended)	20 units maximum

2 Communications using the embedded Ethernet function is processed by the CPU of the CNC. This means that the operation state of the CNC can affect the performance of communication based on the embedded Ethernet function, and communication based on the embedded Ethernet function can affect the processing of the CNC.
The embedded Ethernet function has lower priority than axis-by-axis processing such as automatic operation processing and manual operation, So, when automatic operation is being performed or many controlled axes are involved, communication may become slower.
On the contrary, the embedded Ethernet function has higher priority over CNC screen display processing, C language executor processing (excluding execution macros). So, communication based on the embedded Ethernet function can decrease the performance of such processing.

3 Note that when the embedded Ethernet function is connected to an intranet that handles large volumes of broadcast data, for example, the processing of broadcast data can take a longer time, resulting in a decrease in performance of processing such as CNC screen display processing.

14.3 FIELD NETWORKS

The option board supports the field networks listed below, and DI/DO signals assigned to PMC addresses can be transferred to other CNCs and other vendors' devices that comply with the same communication standards.

- PROFIBUS-DP functions (master/slave)
  PROFIBUS-DP is a communication function defined by PROFIBUS Organization.
  PROFIBUS-DP has a master function and a slave function, and the CNC supports both functions.
  DI/DO signals can be transferred between the CNC and a device provided with PROFIBUS-DP.

NOTE

The PROFIBUS Setting Tool (drawing number: A08B-9510-J530) is provided as a personal computer tool related to the PROFIBUS-DP master function. This tool enables PROFIBUS-DP master function parameters to be created on your personal computer. Those who use the PROFIBUS-DP master function for the first time or are unfamiliar with PROFIBUS-DP master function setting are recommended to purchase a copy of this tool.

- DeviceNet functions (master/slave)
  DeviceNet is a communication function defined by Open DeviceNet Vendor Association, Inc. (ODVA).
  DeviceNet has a master function and a slave function, and the CNC can support the both functions. DI/DO signals can be transferred between the CNC and a device provided with the DeviceNet function.

- FL-net functions
  FL-net is a communication function defined by Japan Electrical Manufacturers’ Association (JEMA). In FL-net, there is no distinction like the master function and the slave function. DI/DO signals can be transferred between the CNC and a device provided with the FL-net function.

NOTE

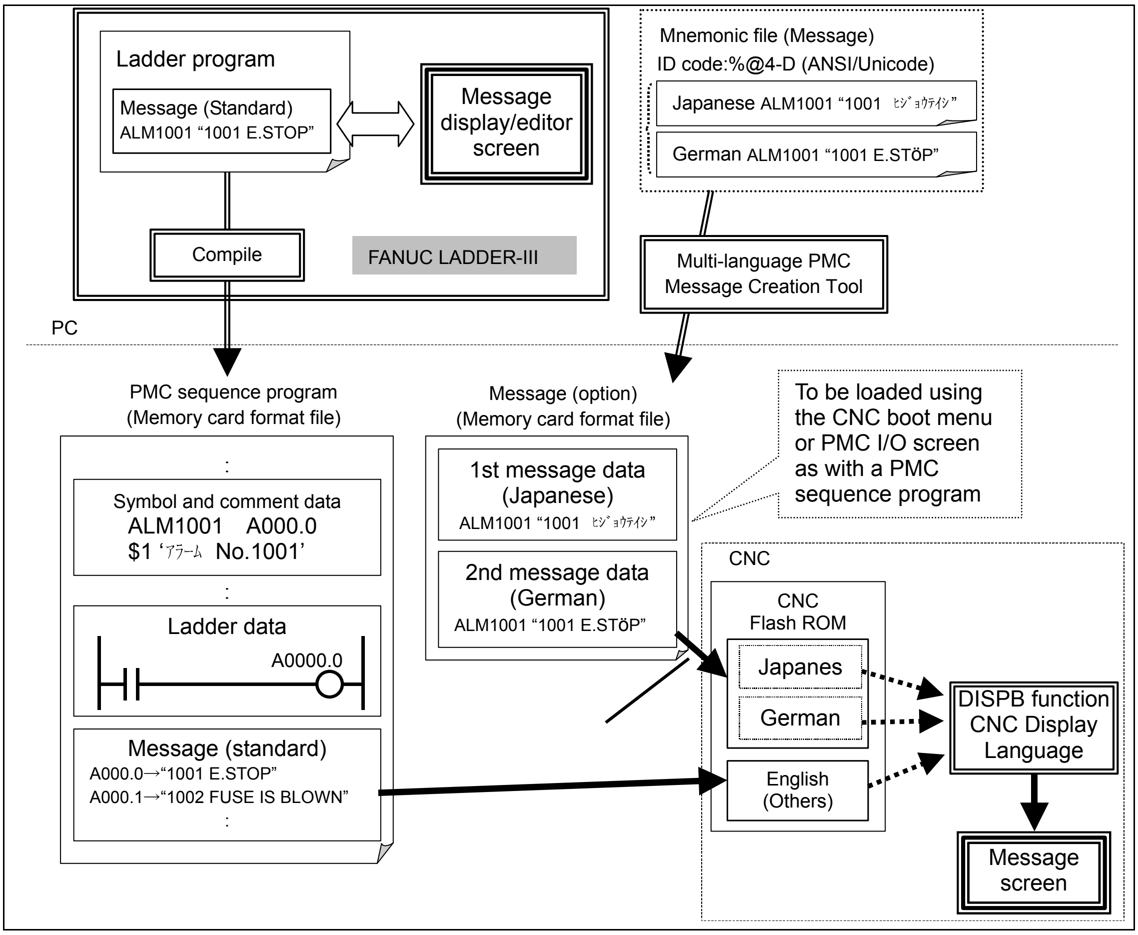
The FL-net setting tool (drawing number: A08B-9510-J536) is provided as a personal computer tool related to the FL-net function. The FL-net setting tool allows you to easily create FL-net function parameters on your personal computer. Those who use the FL-net function for the first time or are unfamiliar with FL-net function setting are recommended to purchase a copy of this tool.

15 PMC

Chapter 15, "PMC", consists of the following sections:

15.1 PMC MESSAGE MULTI-LANGUAGE DISPLAY FUNCTION

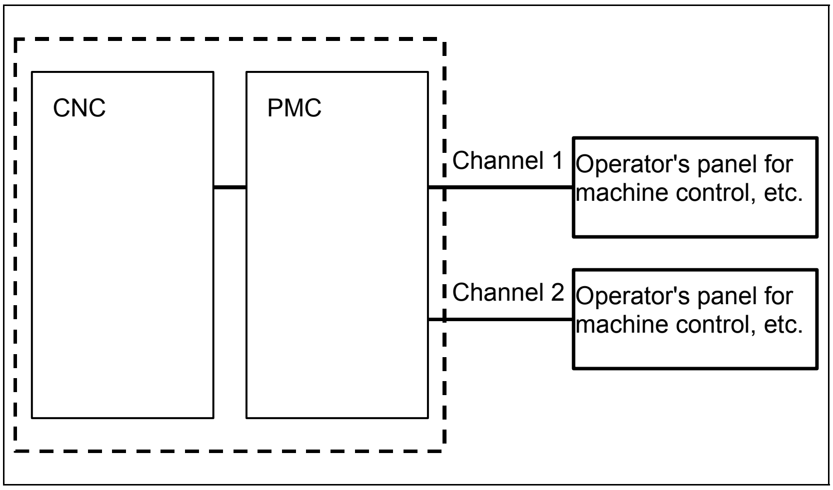
The PMC message multi-language display function manages the language of alarm message and operator message, switching the language according to the language setting of CNC using message data defined in various languages. The message data for this function is stored into a memory card format data, separated from the data of ladder program, and loaded into CNC individually. Up to 2000 messages can be registered for each language.



15.2 I/O Link EXPANSION SECOND/THIRD/FOURTH CHANNEL

The number of DI/DO signal points that can be input/output on the I/O Link is extended.
If channel 2 is added, it is possible to increase the number of DI/DO signals that can be input/output by 1024 points.

PMC addresses to and from which signals are transferred on the I/O Link can be allocated to each channel by parameter setting.



NOTE
The maximum number of DI/DO points per channel is 1024 points/1024 points.

16 OTHERS

Chapter 16, "OTHERS", consists of the following sections:

16.1 STATUS OUTPUT SIGNAL

16.1.1 NC Ready Signal

This signal is sent to the machine side when CNC power is on and control becomes possible. Sending of this signal will be stopped when CNC power is turned off.

16.1.2 Servo Ready Signal

This signal is sent to the machine side when the servo system becomes operatable. Axes necessary to be braked must be braked when this signal is not sent.

16.1.3 Automatic Operation Signal

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

16.1.4 Automatic Operation Start Lamp Signal

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

16.1.5 Feed Hold Signal

This signal is sent out when a stop is made by feed hold or the like (when commands to be executed remain).

16.1.6 Reset Signal

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

16.1.7 NC Alarm Signal

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

16.1.8 Distribution End Signal

This signal is sent out when pulse distribution of the M, S, T, or second auxiliary function has ended, so that they can be used after move of the commanded block ends.

16.1.9 Rewinding Signal

This signal shows that main program in program memory is rewinding.

16.1.10 Inch Input Signal

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

16.1.11 Cutting Signal

This signal shows that the move command is done under cutting.

16.1.12 In-position Signal

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

16.1.13 Threading Signal

This signal shows that the threading mode or the threading cycle for turning is in progress.

16.1.14 Tapping Signal

This signal is output to show that the machine is under tapping mode (G63) or tapping cycle (G74 or G84 for M series), (G84 or G88 for T series) in the canned cycle is under operation.

16.1.15 Axis Moving Signal

This signal shows that an axis is moving. This signal is sent out for each 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.

16.1.16 Axis Moving Direction Signal

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

16.1.17 Rapid Traverse Signal

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

16.1.18 Overtravel Alarm Signal

Indicates whether the tool was about to enter the parameter-specified forbidden area (stored stroke limits).

16.1.19 Constant Surface Speed Signal

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

16.1.20 DI Status Output Signal

To inform the exterior of the states of software operator’s panel, which are set via MDI panel, and machine operator’s panel, following DI state output signals are sent.

- Mode-select check signal
- Single-block check signal
- Manual absolute check signal
- Dry-run check signal
- Machine-lock check signal
- Display-lock check signal
- Auxiliary-function-lock check signal
- Optional block-skip check signal
- Mirror-image check signal

APPENDIX

A RANGE OF COMMAND VALUE

- In case of millimeter input, feed screw is millimeter

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (mm)	0.01	0.001	0.0001
Least command increment (mm)	0.01	0.001	0.0001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (mm/min)*1	999,000	999,000	100,000
Feedrate range (mm/min)*1	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (mm)*2	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- In case of inch input, feed screw is millimeter

	Increment system		
	IS-A	IS-B	IS-C
Least input increment （inch）	0.001	0.0001	0.00001
Least command increment （inch）	0.001	0.0001	0.00001
Max. programmable dimension (inch)	±39,370.078	±39,370.0787	±3,937.00787
Max. rapid traverse (mm/min)*1	999,000	999,000	100,000
Feedrate range （inch/min)*1	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01
Tool compensation amount (inch)*2	0 to ±999.999	0 to ±999.9999	0 to ±999.99999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- In case of inch input, feed screw is inch

	Increment system		
	IS-A	IS-B	IS-C
Least input increment （inch）	0.001	0.0001	0.00001
Least command increment （inch）	0.001	0.0001	0.00001
Max. programmable dimension (inch)	±99,999.999	±99,999.9999	±9,999.99999
Max. rapid traverse (inch/min)*1	96,000	9,600	4,000
Feedrate range （inch/min） *1	0.001 to 96,000	0.0001 to 9,600	0.00001 to 4,000
Incremental feed (inch/step)	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1	0.00001 0.0001 0.001 0.01
Tool compensation amount (inch)*4	0 to ±999.999	0 to ±999.9999	0 to ±999.99999
Backlash compensation amount (pulses)*3	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)*4	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- In case of millimeter input, feed screw is inch

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (mm)	0.01	0.001	0.0001
Least command increment (mm)	0.01	0.001	0.0001
Max. programmable dimension (mm)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (inch/min)[*1]	96,000	9,600	4,000
Feedrate range (mm/min)[*1]	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (mm/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (mm)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

- Rotary axis

	Increment system		
	IS-A	IS-B	IS-C
Least input increment (deg)	0.01	0.001	0.0001
Least command increment (deg)	0.01	0.001	0.0001
Max. programmable dimension (deg)	±999,999.99	±999,999.999	±99,999.9999
Max. rapid traverse (deg/min)[*1]	999,000	999,000	100,000
Feedrate range (deg/min)[*1]	0.01 to 999,000	0.001 to 999,000	0.0001 to 100,000
Incremental feed (deg/step)	0.01 0.1 1.0 10.0	0.001 0.01 0.1 1.0	0.0001 0.001 0.01 0.1
Tool compensation amount (deg)[*2]	0 to ±9,999.99	0 to ±9,999.999	0 to ±9,999.9999
Backlash compensation amount (pulses)[*3]	0 to ±9,999	0 to ±9,999	0 to ±9,999
Dwell (sec)[*4]	0 to 999,999.99	0 to 999,999.999	0 to 99,999.9999

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.

*2 If the mode of input is switched between inch input and metric input, the maximum compensation value that can be set at inch input time is (maximum compensation value) × 1/25.4. If a value exceeding this value is specified at inch input time, the compensation value is not converted to a metric value correctly when the mode of input is switched to metric input.

*3 The unit is the detection unit.

*4 Depends on the increment system of the axis at in address X.

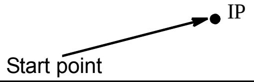
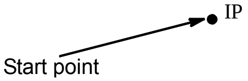
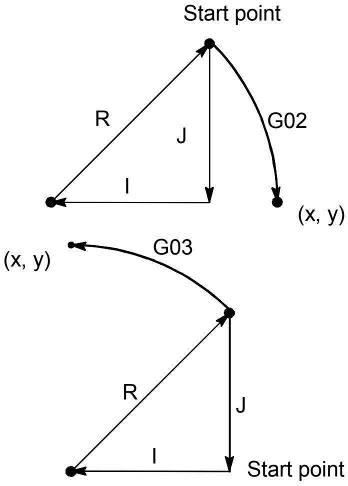
B LIST OF FUNCTIONS AND PROGRAM FORMAT

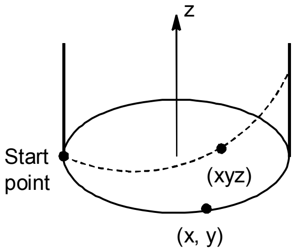
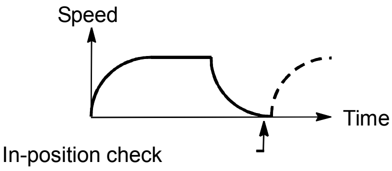
With some functions, the format used for specification on the M series differs from the format used for specification on the T series. Some functions are supported only for either M series or T series.
Some functions cannot be added as options depending on the model.
For details of command formats, see the relevant sections or subsections.

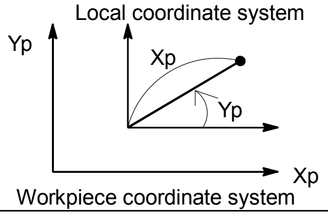
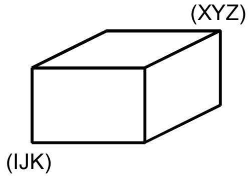
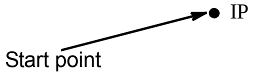
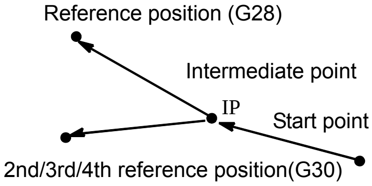
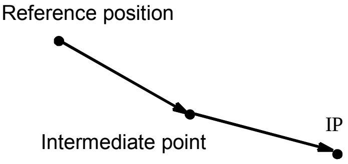
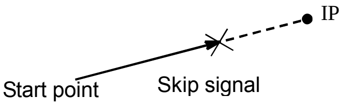
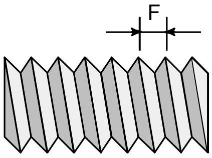
In the list, the following symbols are used:

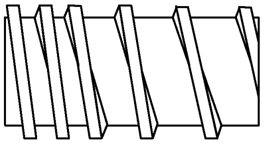
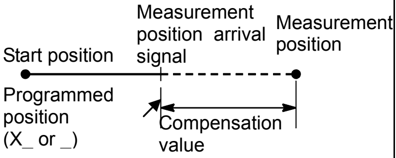
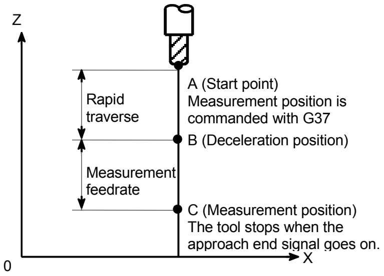
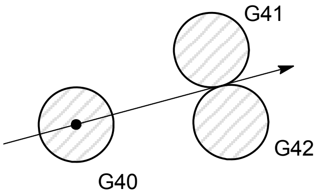
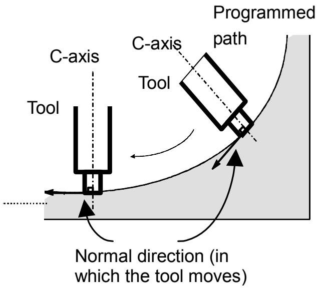
- For M series
  x : 1st basic axis (X),
  y : 2nd basic axis (Y),
  z : 3rd basic axis (Z)
- For T series
  x : 1st basic axis (X),
  z : 2nd basic axis (Z),
  coded using G code system A

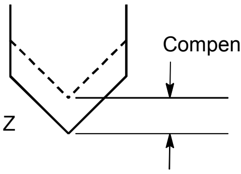
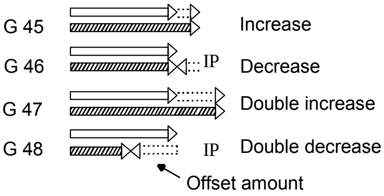
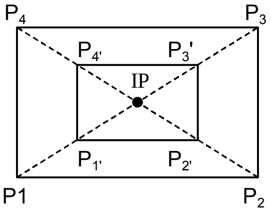
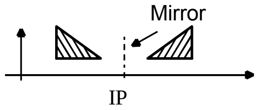
IP_ : presents a combination of arbitrary axis addresses using X, Y, Z, A, B, C, U, V, and W (such as X_ Y_ Z_A_).
α : One of the arbitrary addresses
β : One of the arbitrary addresses
Xp : X-axis or axis parallel to the X-axis
Yp : Y-axis or axis parallel to the Y-axis
Zp : Z-axis or axis parallel to the Z-axis

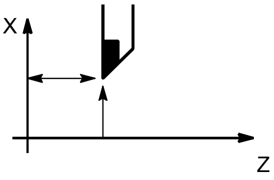
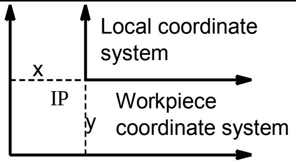
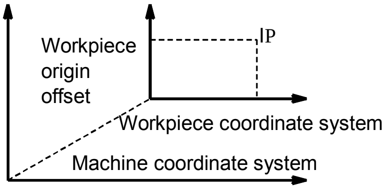
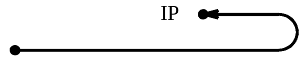
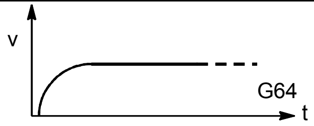
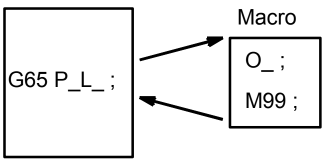
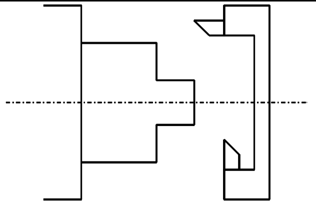
Functions	Illustration	Program format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		G17 $\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}$ X_ Y_ $\begin{Bmatrix}\text{R\_}\\\text{I\_ J\_}\end{Bmatrix}$ F_ ; G18 $\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}$ X_ Z_ $\begin{Bmatrix}\text{R\_}\\\text{I\_ K\_}\end{Bmatrix}$ F_ ; G19 $\begin{Bmatrix}\text{G02}\\\text{G03}\end{Bmatrix}$ Y_ Z_ $\begin{Bmatrix}\text{R\_}\\\text{J\_ K\_}\end{Bmatrix}$ F_ ;

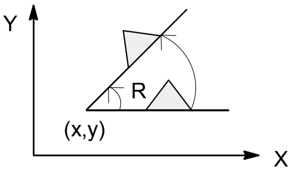
Functions	Illustration	Program format
Helical interpolation (G02, G03)	 In case of G03 on X-Y plane	$G17 \left\{\begin{array}{c}G02\\G03\end{array}\right\} X\_ Y\_ \left\{\begin{array}{c}R\_\\I\_ J\_\end{array}\right\} \alpha\_ F\_ ;$ $G18 \left\{\begin{array}{c}G02\\G03\end{array}\right\} X\_ Z\_ \left\{\begin{array}{c}R\_\\I\_ K\_\end{array}\right\} \alpha\_ F\_ ;$ $G19 \left\{\begin{array}{c}G02\\G03\end{array}\right\} Y\_ Z\_ \left\{\begin{array}{c}R\_\\J\_ K\_\end{array}\right\} \alpha\_ F\_ ;$ α: Arbitrary address except the circular interpolation axis
Dwell (G04)		M $G04 \left\{\begin{array}{c}X\_\\P\_\end{array}\right\} ;$ T $G04 \left\{\begin{array}{c}X\_\\U\_\\P\_\end{array}\right\} ;$
M AI advanced preview control / AI contour control (G05.1)		G05.1 Q1 ; AI advanced preview control / AI contour control mode on G05.1 Q0 ; AI advanced preview control / AI contour control mode off
HRV3 control (G05.4)		G05.4 Q1 ; HRV3 control mode on G05.4 Q0 ; HRV3 control mode off
Cylindrical interpolation (G07.1)		G07 IP_ r_; Cylindrical interpolation mode r : Cylinder radius G07 IP 0; Cylindrical interpolation mode cancel
T Advanced preview control (G08)		G08 P1 ; AI contour control mode on G08 P0 ; AI contour control mode off
Exact stop (G09)		$G09 \left\{\begin{array}{c}G01\\G02\\G03\end{array}\right\} IP\_ ;$
Programmable data input (G10)		M Tool compensation memory A G10 L01 P_ R_ ; Tool compensation memory C G10 L10 P_ R_ ; (Geometry offset amount/H) G10 L11 P_ R_ ; (Wear offset amount/H) G10 L12 P_ R_ ; (Geometry offset amount/D) G10 L13 P_ R_ ; (Wear offset amount/D) T Geometry offset amount G10 P_ X_ Z_ R_ Q_ ; P = 10000 + Geometry offset number Wear offset amount G10 P_ X_ Z_ C_ Q_ ; P = Wear offset number

Functions	Illustration	Program format
T Polar coordinate interpolation (G12.1, G13.1)		G12.1 ; Polar coordinate interpolation mode on G13.1 ; Polar coordinate interpolation cancel
M Polar coordinate command (G15, G16)		G17 G16 Xp_ Yp_ . . . ; G18 G16 Zp_ Xp_ . . . ; G19 G16 Yp_ Zp_ . . . ; G15 ; Cancel
Plane selection (G17, G18, G19)		G17 ; Xp Yp-plane selection G18 ; Zp Xp-plane selection G19 ; Yp Zp-plane selection
Inch/metric conversion (G20, G21)		Inch input G20 ; Metric input G21 ;
Stored stroke check (G22, 23)		G22 X_ Y_ Z_ I_ J_ K_ ; Stored stroke check on G23 ; Stored stroke check off
T Spindle speed fluctuation detection (G25, G26)		G26 P_ Q_ R_ I_ ; Spindle speed fluctuation detection enabled G25 ; Spindle speed fluctuation detection disabled
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd/3rd/4th Reference position return (G30)		G28 IP_ ; Reference position return G30 P2 IP_ ; 2nd reference position return G30 P3 IP_ ; 3rd reference position return G30 P4 IP_ ; 4th reference position return P2 can be omitted.
M Movement from reference position (G29)		G29 IP_ ;
Skip function (G31)		G31 IP_ F_ ;
M Threading (G33) T Threading (G32)		M G33 IP_ F_ ; F : Lead T Equal lead threading G32 IP_ F_ ; F : Lead

Functions	Illustration	Program format
T Variable lead thread cutting (G34)		G34 IP_ F_ K_ ; F : Lead at the start position in the longitudinal direction K : Lead increment/decrement per spindle rotation
T Automatic tool offset (G36,G37)		G36 X_ ; G37 Z_ ;
M Automatic tool length measurement (G37)	 Compensation value = (Current compensation value) + [(Coordinates of the point at which the tool is stopped) - (Coordinates of the programmed measurement position)]	G92 IP_ ; Workpiece coordinate system setting (This can also be set with G54 to G59) HOO ; Offset number specified for tool length compensation G90 G37 IP_ ; Absolute programming IP_ : Measurement position on X_, Y_, Z_, or 4th axis
Tool radius/tool nose radius compensation, (G39, G40 to G42)		M $\left\{\begin{array}{l}G17\\G18\\G19\end{array}\right\} \left\{\begin{array}{l}G41\\G42\end{array}\right\}$ IP_ D_ ; D : Tool compensation number G40 : Cancel T $\left\{\begin{array}{l}G17\\G18\\G19\end{array}\right\} \left\{\begin{array}{l}G41\\G42\end{array}\right\}$ IP_ ; G40 : Cancel
M Normal direction control (G40.1, G41.1, G42.1)		G41.1 ; Normal direction control on : right G42.1 ; Normal direction control on : left G40.1 ; Normal direction control cancel

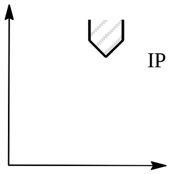
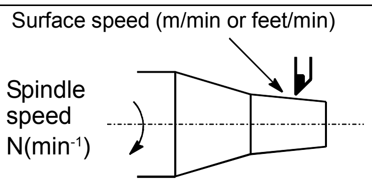
Functions	Illustration	Program format
M Tool length compensation (G43, G44, G49)		$\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ Z_ H_ ; $\left\{\begin{matrix}G17\\G18\\G19\end{matrix}\right\}\left\{\begin{matrix}G43\\G44\end{matrix}\right\}\left\{\begin{matrix}Z\\Y\\X\end{matrix}\right\}$ H_ ; $\left\{\begin{matrix}G43\\G44\end{matrix}\right\}$ IP_ H_ ; H : Tool compensation number G49 : Cancel
Tool offset (G45 to G48)		M $\left\{\begin{matrix}G45\\G46\\G47\\G48\end{matrix}\right\}$ IP_ D_ ; D : Tool offset number
M 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 : Control position of scaling G50 : Cancel
M Programmable mirror image (G50.1, G51.1)		G51.1 IP_ ; Setting IP_: Command for the symmetric axis of the mirror image G50.1 IP_ ; Cancel IP_: Any command for the symmetric axis of the mirror image
T Polygon turning (G50.2, G51.2) (G250, G251)		G51.2 (G251) P_ Q_ ; Polygon turning on P_ Q_ : Rotation ratio between the spindle and rotation axis G50.2 (G250) ; Polygon turning cancel
T Synchronous, composite, and superimposed control by program command (G50.4, G51.4, G50.5, G51.5, G50.6, G51.6)		G51.4 P_Q_(L_) ; Start synchronous control (L_ can be omitted.) G50.4 Q_ ; Cancel synchronous control. P : Number to identify synchronous master axis Q : Number to identify synchronous slave axis L : Parking start command G51.5 P_Q_ ; Start composite control G50.5 P_Q_ ; Cancel composite control P : Number to identify composite axis 1 Q : Number to identify composite axis 2 G51.6 P_Q_ ; Start superimposed control G50.6 Q_ ; Cancel superimposed control P : Number to identify superimposed master axis Q : Number to identify superimposed slave axis

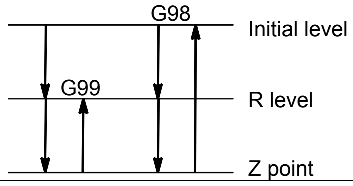
Functions	Illustration	Program format
T Coordinate system setting or Maximum spindle speed clamp (G50)		G50 IP_ ; (Coordinate system setting) G50 S_ ; (Maximum spindle speed clamp)
Local coordinate system setting (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Selection of workpiece coordinate system (G54 to G59)		$\left\{\begin{array}{c}\text{G54}\\ \vdots\\ \text{G59}\end{array}\right\}$ IP_ ;
M Selection of additional workpiece coordinate system (G54.1,G54)	Example) G54.1 P12 ; Select the additional workpiece coordinate system 12.	G54.1 Pn ; (n=1 to 48) G54 Pn ; (n=1 to 48)
M Single direction positioning (G60)		G60 IP_ ;
Cutting mode (G64) Exact stop mode (G61) Tapping mode (G63)		G64_ ; Cutting mode G61_ ; Exact stop mode G63_ ; Tapping mode
M Automatic corner override (G62)		G62_ ; Automatic corner override
Custom macro (G65, G66, G67)		One-shot call G65 P_ L_ <Argument assignment> ; P : Program number L : Number of repetition Modal call G66 P_ L_ <Argument assignment> ; Call after the move command G67 ; Cancel
T Mirror image for double turret (G68, G69)		G68 : Mirror image for double turret G69 : Mirror image cancel

<table>
<tr><th>Functions</th><th>Illustration</th><th>Program format</th></tr>
<tr><td>M
Coordinate system rotation
(G68, G69)</td><td>
In case of X-Y plane</td><td>$G68\left\{\begin{array}{l}G17\ X\_\ Y\_\\ G18\ Z\_\ X\_\\ G19\ Y\_\ Z\_\end{array}\right\}R\ ;$
R: Angle of rotation in a counterclockwise direction

G69 ; Cancel</td></tr>
<tr><td>Canned cycle for drilling
M
(G73, G74, G76, G80 to G89)
T
(G80 to G89)</td><td></td><td>G80 ; Cancel
M
$\left.\begin{array}{l}G73\\ G74\\ G76\\ G81\\ :\\ G89\end{array}\right\}$ X_Y_Z_P_Q_R_F_K_ ;</td></tr>
<tr><td>T
Canned cycle for turning
Multiple repetitive canned cycle
(G70 to G76)
Canned cycle
(G90, G92, G94)</td><td></td><td>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 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 R_ ;
G76 X(u)_ Z(w)_ P_ Q_ R_ F_ ;
$\left\{\begin{array}{l}G90\\ G92\end{array}\right\}$ X_ Z_ I_ F_ ;
G94 X_ Z_ I_ F_ ;</td></tr>
<tr><td>Canned grinding cycle (for grinding machine)
M
(G75 to G79)
T
(G71 to G74)</td><td></td><td>M
G75 I_ J_ K_ α_ R_ F_ P_ L_ ;
G77 I_ J_ K_ α_ R_ F_ P_ L_ ;
G78 I_ (J_) K_ α_ F_ P_ L_ ;
G79 I_ J_ K_ α_ R_ F_ P_ L_ ;
α : Arbitrary axis address of the grinding axis

T
G71 A_ B_ W_ U_ I_ K_ H_ ;
G72 P_ A_ B_ W_ U_ I_ K_ H_ ;
G73 A_ (B_) W_ U_ K_ H_ ;
G74 P_ A_ (B_) W_ U_ K_ H_ ;</td></tr>
<tr><td>M
Electronic gear box
(G81,G80)
(G81.4,G80.4)</td><td></td><td><table>
<tr><td rowspan="2"></td><td colspan="2">Parameter EFX
（No.7731#0）</td></tr>
<tr><td>0</td><td>0</td></tr>
<tr><td>Start of synchronization</td><td>G81 T_ (L_) (Q_ P_) ;</td><td>G81 T_ (L_) (Q_ P_) ;</td></tr>
<tr><td>Cancellation of synchronization</td><td>G80 ;</td><td>G80 ;</td></tr>
</table>T : Number of teeth
L : Number of hob threads
Q : Module or diametral pitch
P : Gear helix angle</td></tr>
</table>

Functions	Illustration	Program format
Absolute/incremental programming (G90/G91)		M G90_ ; Absolute programming G91_ ; Incremental programming : G90_ . . . G91_ ; Programming in both modes T For G code system A X_ Z_ C_ : Absolute programming U_ W_ H_ :Incremental programming For G code system B/C G90_ ; Absolute programming G91_ ; Incremental programming : G90_ ⋯ G91_ ; Programming in both modes
Maximum incremental command value check (G91.1)		G91.1 IP_ ; IP_ ; Maximum incremental value Set 0 to cancel maximum incremental value check.
Change of workpiece coordinate system or Maximum spindle speed clamp (G92)		M G92 IP_ ; Change of workpiece coordinate system G92 S_ ; Constant surface speed control : Maximum spindle speed clamp
Workpiece coordinate system preset M (G92.1) T (G50.3)		M G92.1 IP 0 ; T G50.3 IP 0 ;
M Inverse time feed (G93)		G93 ; Inverse time setting mode
Feed per minute, Feed per revolution M (G94, G95) T (G98, G99)	mm/min inch/min mm/rev inch/rev	M G94 F_ ; Feed per minute G95 F_ ; Feed per revolution T G98 F_ ; Feed per minute G99 F_ ; Feed per revolution
Constant surface speed control (G96, G97)		G96 S_ ; Constant surface speed control on (surface speed specification) G97 S_ ; Constant surface speed control off (spindle speed specification)

Functions	Illustration	Program format
T Speed display function of a milling tool with servo motor (G96.1,G96.2,G96.3,G96.4)		G96.1 P_R_; The next block starts operating upon completion of spindle indexing (the SV speed control mode is off). G96.2 P_R_; The next block starts operating without waiting for spindle indexing to complete. G96.3 P_; The next block starts operating after confirming the completion of spindle indexing (the SV speed control mode is off). G96.4 P_; The SV speed control mode is on.
Canned cycle initial level return/R level return (G98, G99)		G98_ ; Canned cycle initial level return G99_ ; Canned cycle R level return T They can be used only in the G code system B/C.
M In-feed control (for grinding machine) (G160, G161)		G161 R_ ; Figure program (G01, G02, G03) G160 ;

C PROGRAM CODE LIST

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Number 0	0	30	0	20			*
Number 1	1	B1	1	01			*
Number 2	2	B2	2	02			*
Number 3	3	33	3	13			*
Number 4	4	B4	4	04			*
Number 5	5	35	5	15			*
Number 6	6	36	6	16			*
Number 7	7	B7	7	07			*
Number 8	8	B8	8	08			*
Number 9	9	39	9	19			*
Address A	A	41	a	61			*
Address B	B	42	b	62			*
Address C	C	C3	c	73			*
Address D	D	44	d	64			*
Address E	E	C5	e	75			*
Address F	F	C6	f	76			*
Address G	G	47	g	67			*
Address H	H	48	h	68			*
Address I	I	C9	i	79			*
Address J	J	CA	j	51			*
Address K	K	4B	k	52			*
Address L	L	CC	l	43			*
Address M	M	4D	m	54			*
Address N	N	4E	n	45			*
Address O	O	CF	o	46			*
Address P	P	50	p	57			*
Address Q	Q	D1	q	58			*
Address R	R	D2	r	49			*
Address S	S	53	s	32			*
Address T	T	D4	t	23			*
Address U	U	55	u	34			*
Address V	V	56	v	25			*
Address W	W	D7	w	26			*
Address X	X	D8	x	37			*
Address Y	Y	59	y	38			*
Address Z	Z	5A	z	29			*
Delete	DEL	FF	Del	7F	×	×	
Back space	BS	88	BS	2A	×	×	
Tabulator	HT	09	Tab	2E	×	×	
End of block	LF or NL	0A	CR or EOB	80			
Carriage return	CR	8D			×	×	
Space	SP	A0	SP	10	□	□	
Absolute rewind stop	%	A5	ER	0B			
Control out (start of comment)	(	28	(2-4-5)	1A			
Control in (end of comment)	)	A9	(2-4-7)	4A			

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Plus sign	+	2B	+	70			*
Minus sign	-	2D	-	40			*
Colon (address O)	:	3A					
Optional block skip	/	AF	/	31			
Period (decimal point)	.	2E	.	6B			*
Sharp	#	A3	Parameter (No.6012)				
Dollar sign	$	24			□	□	
Ampersand	&	A6	&	0E			
Apostrophe	’	27			□	□	
Asterisk	*	AA	Parameter (No.6010)				
Comma	,	AC	,	3B			
Semicolon	;	FB			□	□	
Left angle bracket	<	2C					
Equal sign	=	BD	Parameter (No.6011)				
Right angle bracket	>	BE					
Question mark	?	3F			△	○	
Commercial at mark	@	C0			△	△	
Quotation mark	“	22			□	□	
Left square bracket	[	DB	Parameter (No.6013)		△		
Right square bracket	]	DD	Parameter (No.6014)		△		
Underscore	_	6F	Parameter (No.6018)		△	△	*
Lowercase letter a	a	E1			△	△	*
Lowercase letter b	b	E2			△	△	*
Lowercase letter c	c	63			△	△	*
Lowercase letter d	d	E4			△	△	*
Lowercase letter e	e	65			△	△	*
Lowercase letter f	f	66			△	△	*
Lowercase letter g	g	E7			△	△	*
Lowercase letter h	h	E8			△	△	*
Lowercase letter i	i	69			△	△	*
Lowercase letter j	j	6A			△	△	*
Lowercase letter k	k	EB			△	△	*
Lowercase letter l	l	6C			△	△	*
Lowercase letter m	m	ED			△	△	*
Lowercase letter n	n	EE			△	△	*
Lowercase letter o	o	6F			△	△	*
Lowercase letter p	p	F0			△	△	*
Lowercase letter q	q	71			△	△	*
Lowercase letter r	r	72			△	△	*
Lowercase letter s	s	F3			△	△	*
Lowercase letter t	t	74			△	△	*
Lowercase letter u	u	F5			△	△	*
Lowercase letter v	v	F6			△	△	*
Lowercase letter w	w	77			△	△	*
Lowercase letter x	x	78			△	△	*

Character name	ISO code		EIA code		Custom macro		Usable as file name
	Character	Code (hexadecimal)	Character	Code (hexadecimal)	without custom macro	with custom macro	
Lowercase letter y	y	F9			△	△	*
Lowercase letter z	z	FA			△	△	*

NOTE

1 The symbols used in the "Custom macro" column have the following meanings.
(Space) : The character will be registered in memory and has a specific meaning. It is used incorrectly in a statement other than a comment, an alarm occurs.
× : The character will not be registered in memory and will be ignored.
△ : The character will be registered in memory, but will be ignored during program execution. If the character is indicated with “*” in the “Usable as file name” column and used in a file name, however, it will not be ignored.
○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
□ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2 The symbol used in the column "Usable as file name" has the following meaning:
*: Capable of being coded between "<" and ">" as a file name.

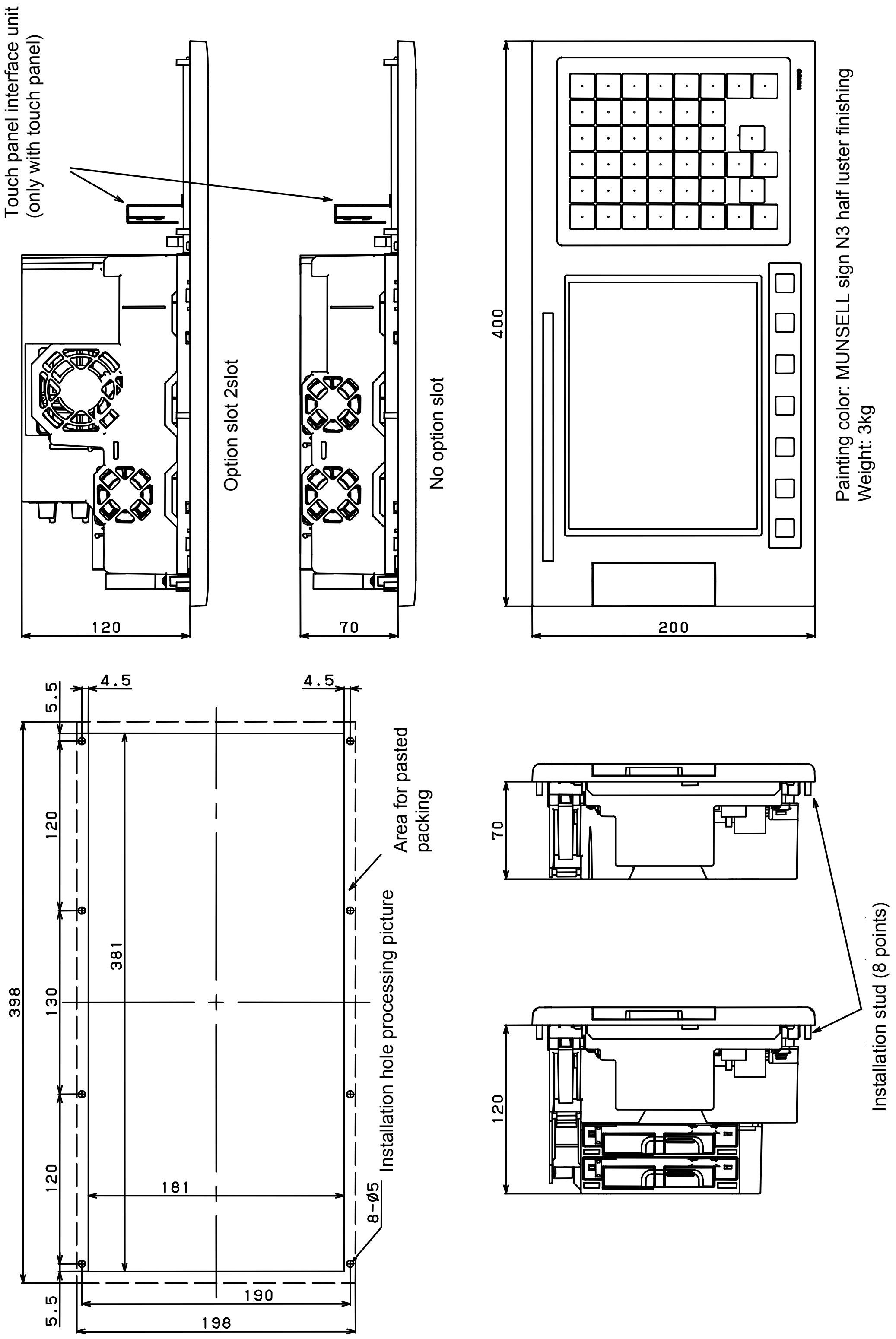
3 Codes not in this table are ignored if their parity is correct.

4 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

D EXTERNAL DIMENSIONS OF EACH UNIT

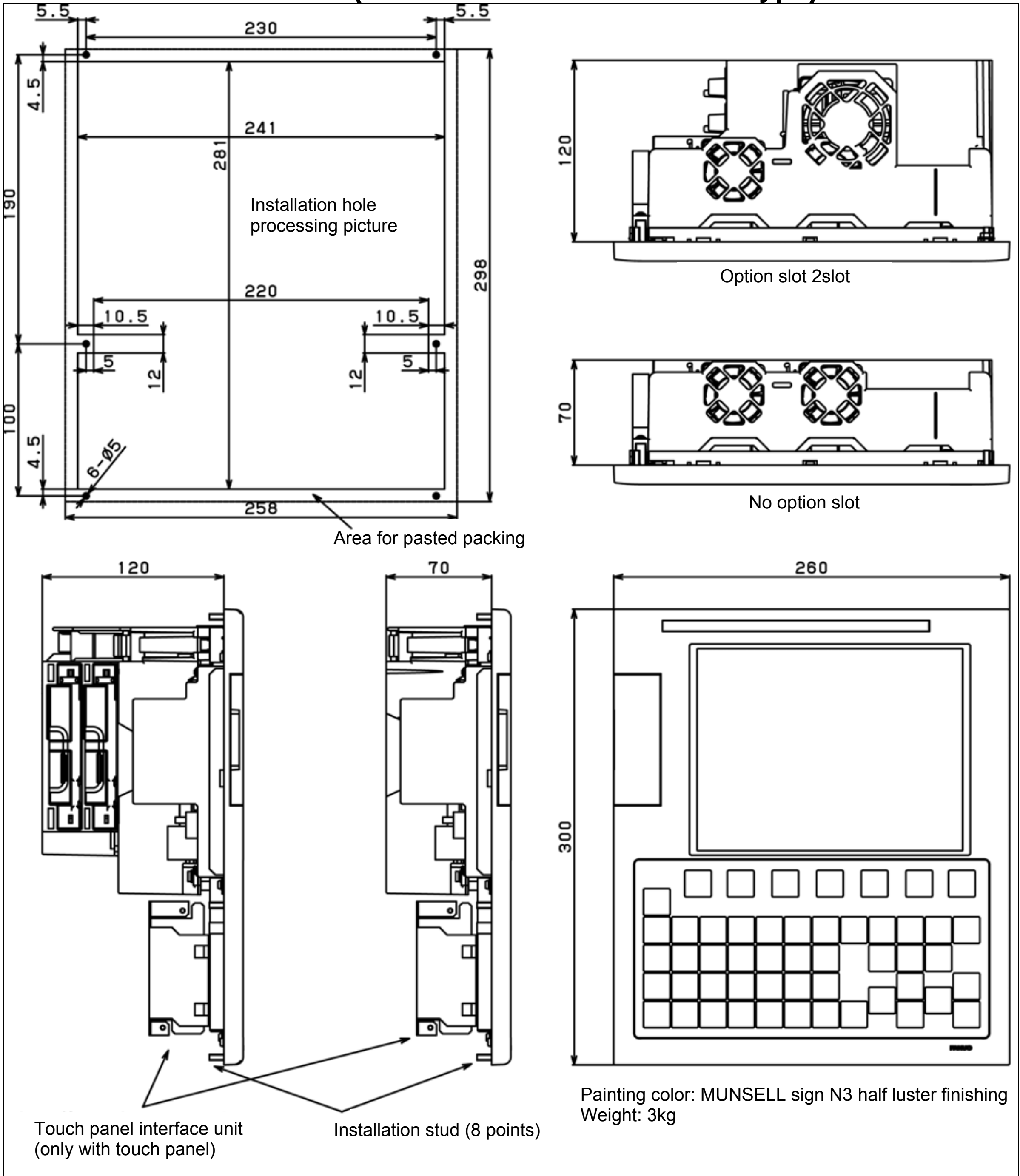
Outline drawing name		Specification drawing number	Fig.
CNC control unit (8.4” color LCD/MDI horizontal type)			Fig. U1
CNC control unit (8.4” color LCD/MDI vertical type)			Fig. U2
CNC control unit (10.4” color LCD)			Fig. U3
MDI unit	Horizontal type	A02B-0319-C125#T, A02B-0319-C125#M	Fig. U4 (a)
	Vertical type	A02B-0319-C126#T, A02B-0319-C126#M	Fig. U4 (b)
	Small type	A02B-0303-C120#T, A02B-0303-C120#M	Fig. U4 (c)
Portion in which each CNC control unit is installed			Fig. U6
I/O unit for 0*i*		A02B-0309-C001	Fig. U7
αi position coder	10000 min^{-1}	A860-2109-T302	Fig. U17
Manual pulse generator		A860-0203-T001	Fig. U18
Pendant type manual pulse generator		A860-0203-T004	Fig. U19
		A860-0203-T005	
		A860-0203-T007	
		A860-0203-T010	
		A860-0203-T012	
		A860-0203-T013	
Separate detector interface unit		A02B-0303-C205 A02B-0236-C204	Fig. U20
Battery case for separate detector interface unit (ABS)		A06B-6050-K060	Fig. U21
CNC battery unit for external installation		A02B-0236-C282	Fig. U22
Punch panel Narrow width type	Cable length: 1m	A02B-0236-C191	Fig. U24
	Cable length: 2m	A02B-0236-C192	
	Cable length: 5m	A02B-0236-C193	
Machine operator's panel: Main panel A		A02B-0319-C242	Fig. U25
Machine operator's panel: Main panel B		A02B-0319-C243	Fig. U26
Machine operator's panel: Sub panel A		A02B-0236-C232	Fig. U27
Machine operator's panel: Sub panel B1		A02B-0236-C235	Fig. U28
Screw cap (for small MDI unit)		A02B-0303-K190, A02B-0303-K191	Fig. U29
Stand-alone type control unit			Fig. S1
Punch Panel (for Stand-alone Type Control Unit)	Cable length: 1m	A02B-0120-C191	Fig. S2
	Cable length: 2m	A02B-0120-C192	
	Cable length: 5m	A02B-0120-C193	

Fig. U1 CNC control unit (8.4” color LCD/MDI horizontal type)



NOTE The unit is installed from outside cabinet.
Fix with M4 nut from the reverse side. The tightening torque is 2.0 N·m.

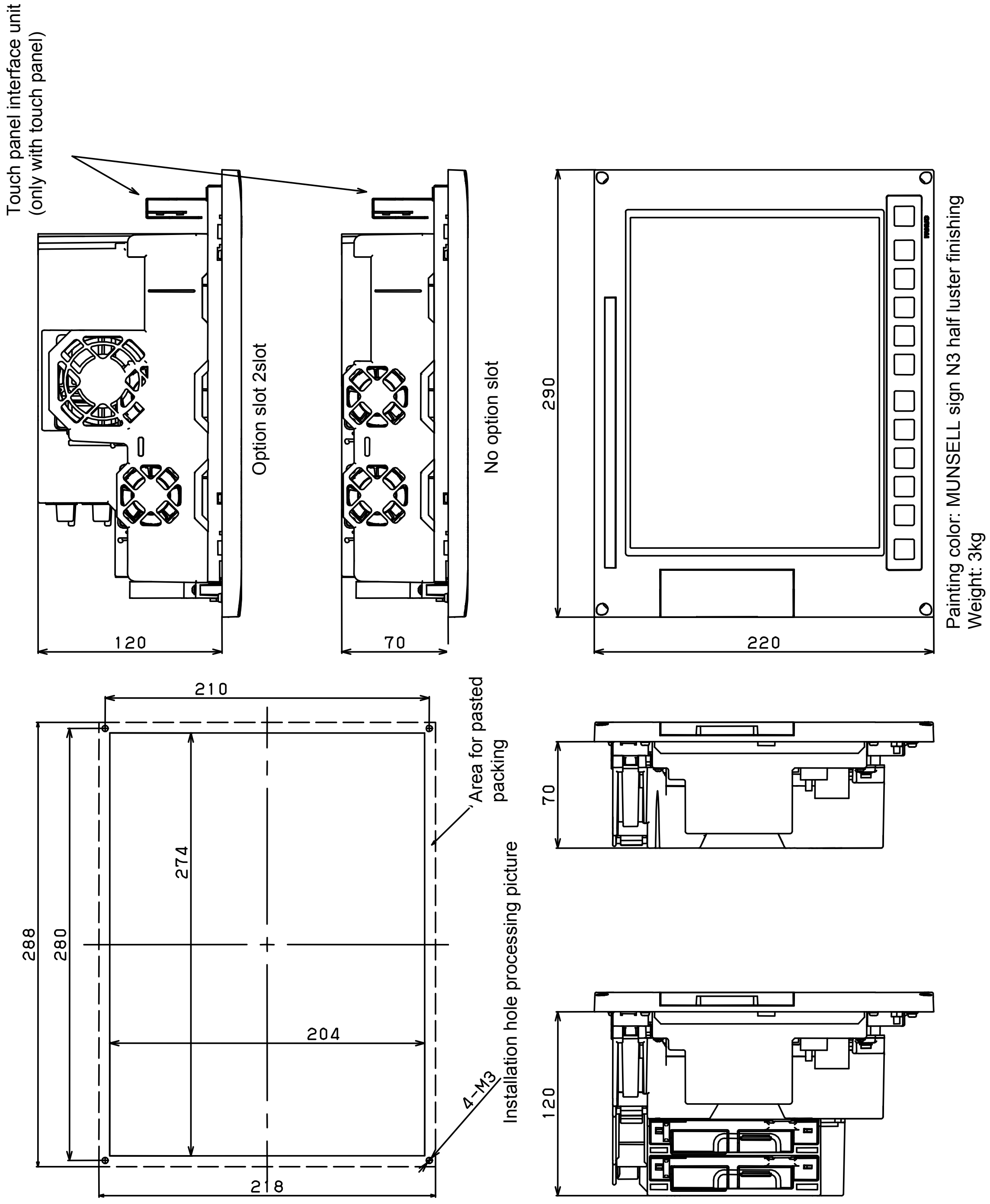
Fig. U2 CNC control unit (8.4” color LCD/MDI vertical type)



Painting color: MUNSELL sign N3 half luster finishing
Weight: 3kg

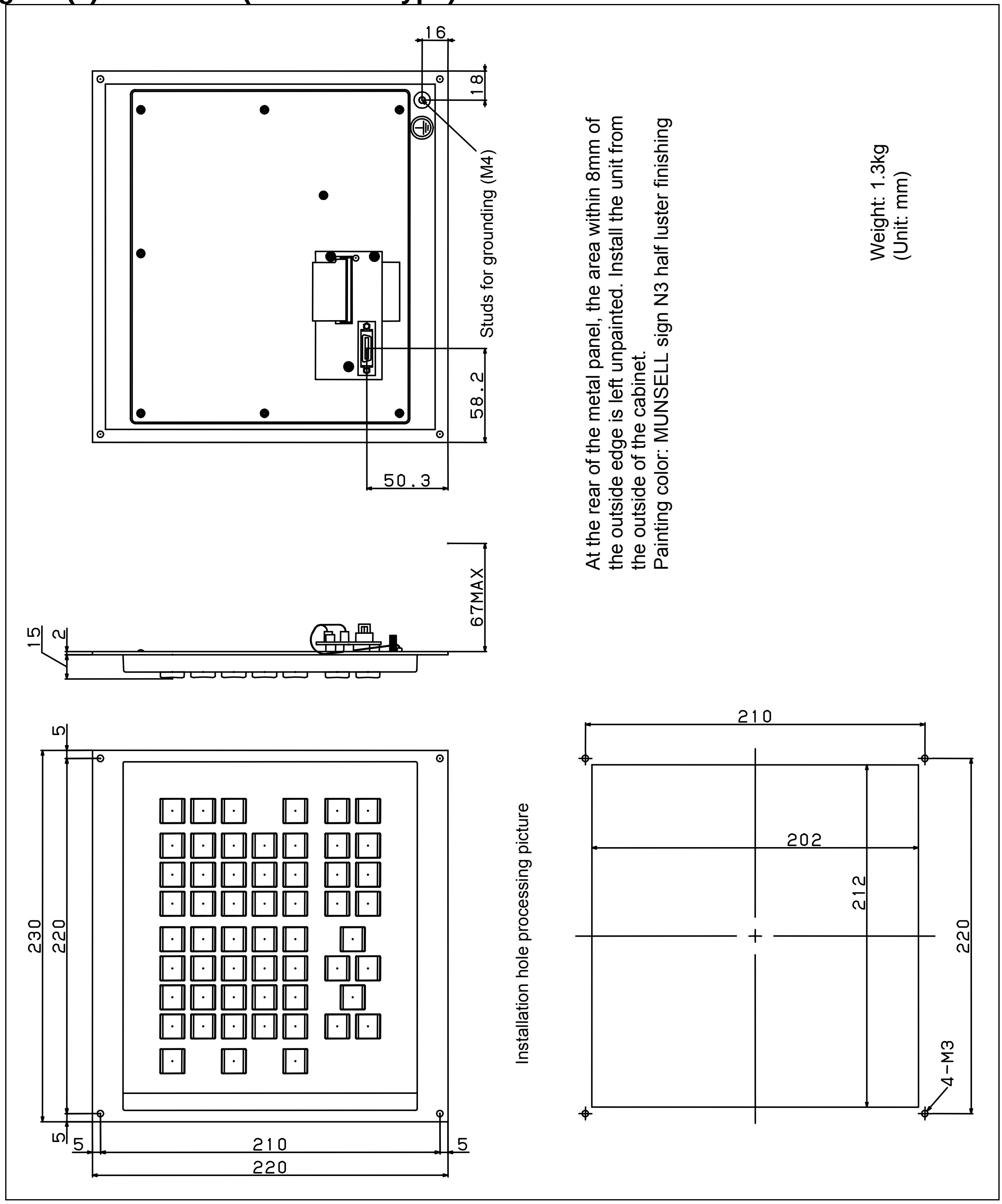
NOTE The unit is installed from outside cabinet.
Fix with M4 nut from the reverse side. The tightening torque is 2.0 N·m.

Fig. U3 CNC control unit (10.4” color LCD)



NOTE The unit is stopped with the screw from outside of the cabinet, then install the screw cap.
The unit with a touch panel has no soft keys.

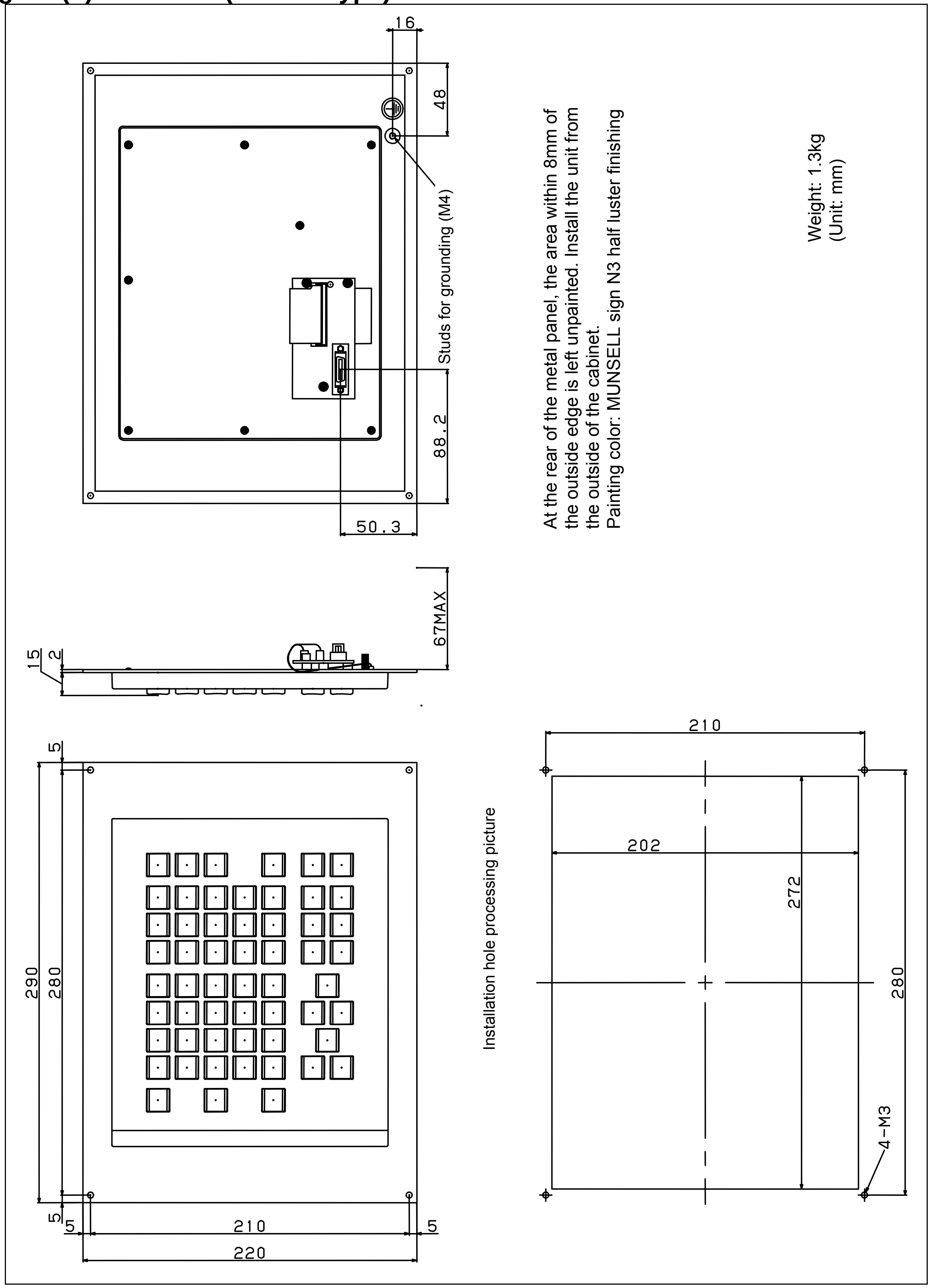
Fig. U4(a) MDI unit (horizontal type)



At the rear of the metal panel, the area within 8mm of the outside edge is left unpainted. Install the unit from the outside of the cabinet.
Painting color: MUNSELL sign N3 half luster finishing

Weight: 1.3kg
(Unit: mm)

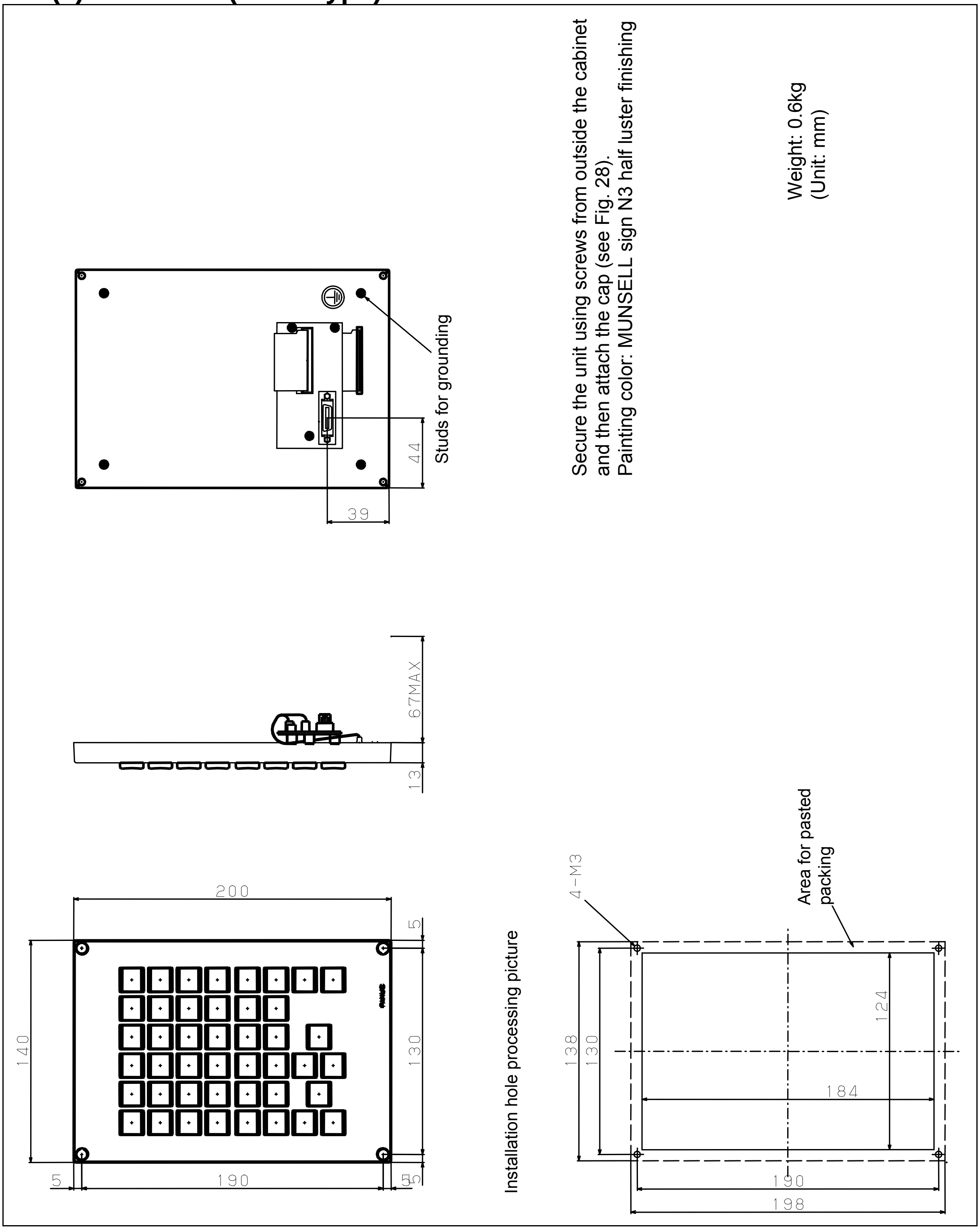
Fig. U4(b) MDI unit (vertical type)



At the rear of the metal panel, the area within 8mm of the outside edge is left unpainted. Install the unit from the outside of the cabinet.
Painting color: MUNSELL sign N3 half luster finishing

Weight: 1.3kg
(Unit: mm)

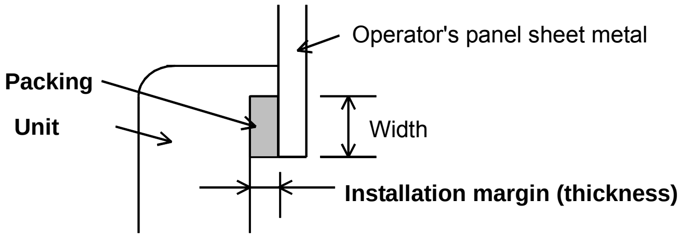
Fig. U4(c) MDI unit (small type)



Secure the unit using screws from outside the cabinet and then attach the cap (see Fig. 28).
Painting color: MUNSELL sign N3 half luster finishing

Weight: 0.6kg
(Unit: mm)

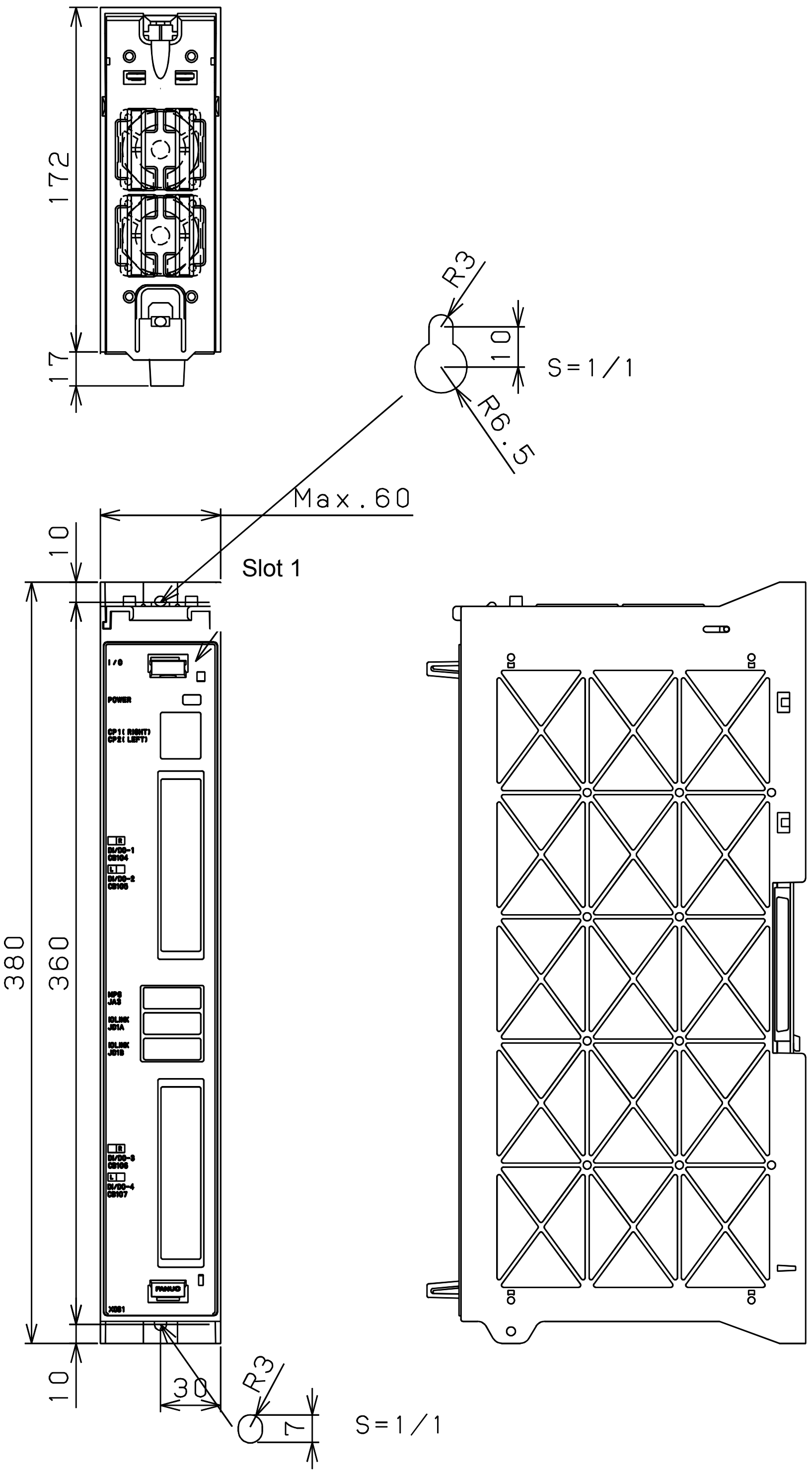
Fig. U6 Portion in which each CNC control unit is installed



CAUTION
When installing the control unit, be sure to seal between the unit and the operator's panel with packing.
For the installation margin (thickness) and width, see the table below.

Display unit type	Installation margin (thickness)	Width
8.4" color LCD/MDI (horizontal type)	0.8mm	8.5mm
8.4" color LCD/MDI (vertical type)		
10.4" color LCD	0.8mm	7mm

Fig. U7 I/O unit for 0i



Weight: 1.2kg

Fig. U17 αi position coder

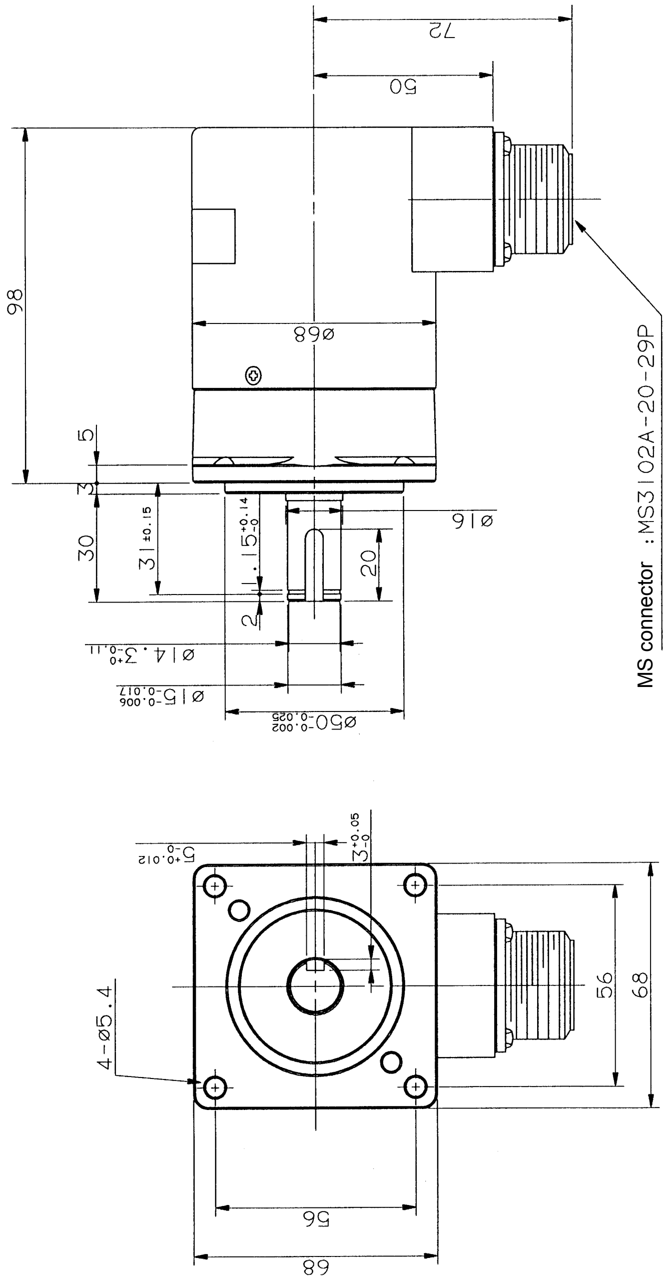


Fig. U18 Manual pulse generator

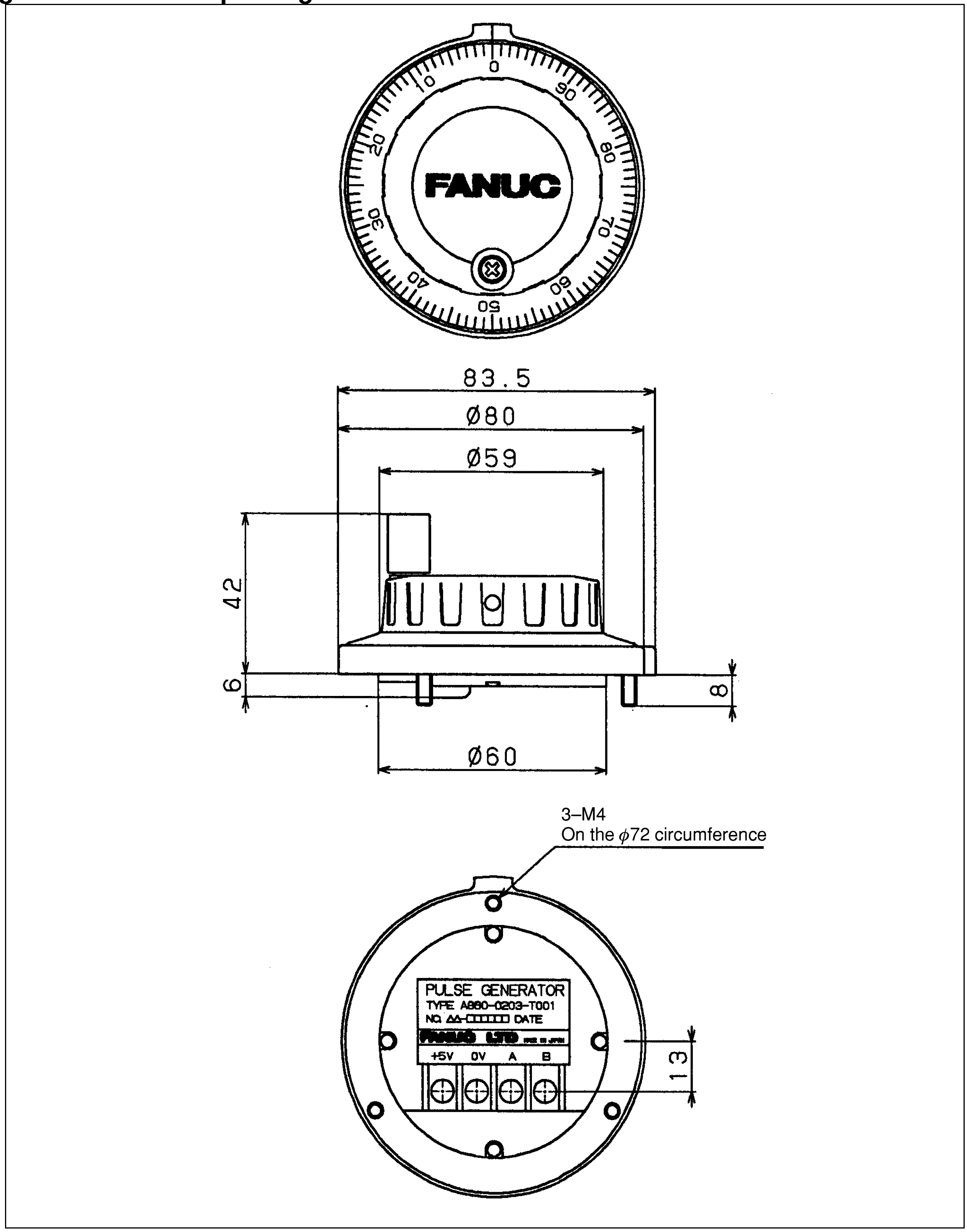


Fig. U19 Pendant type manual pulse generator

A860−0203−T010~T015

Weight: 0.4kg
(Unit: mm)

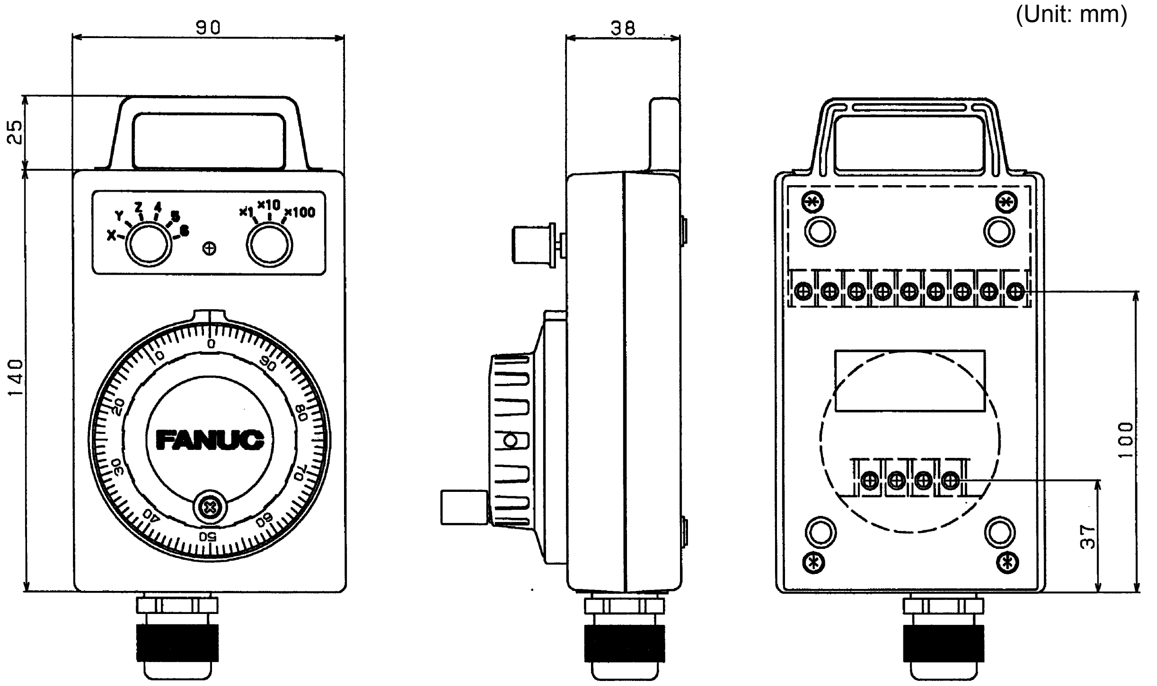
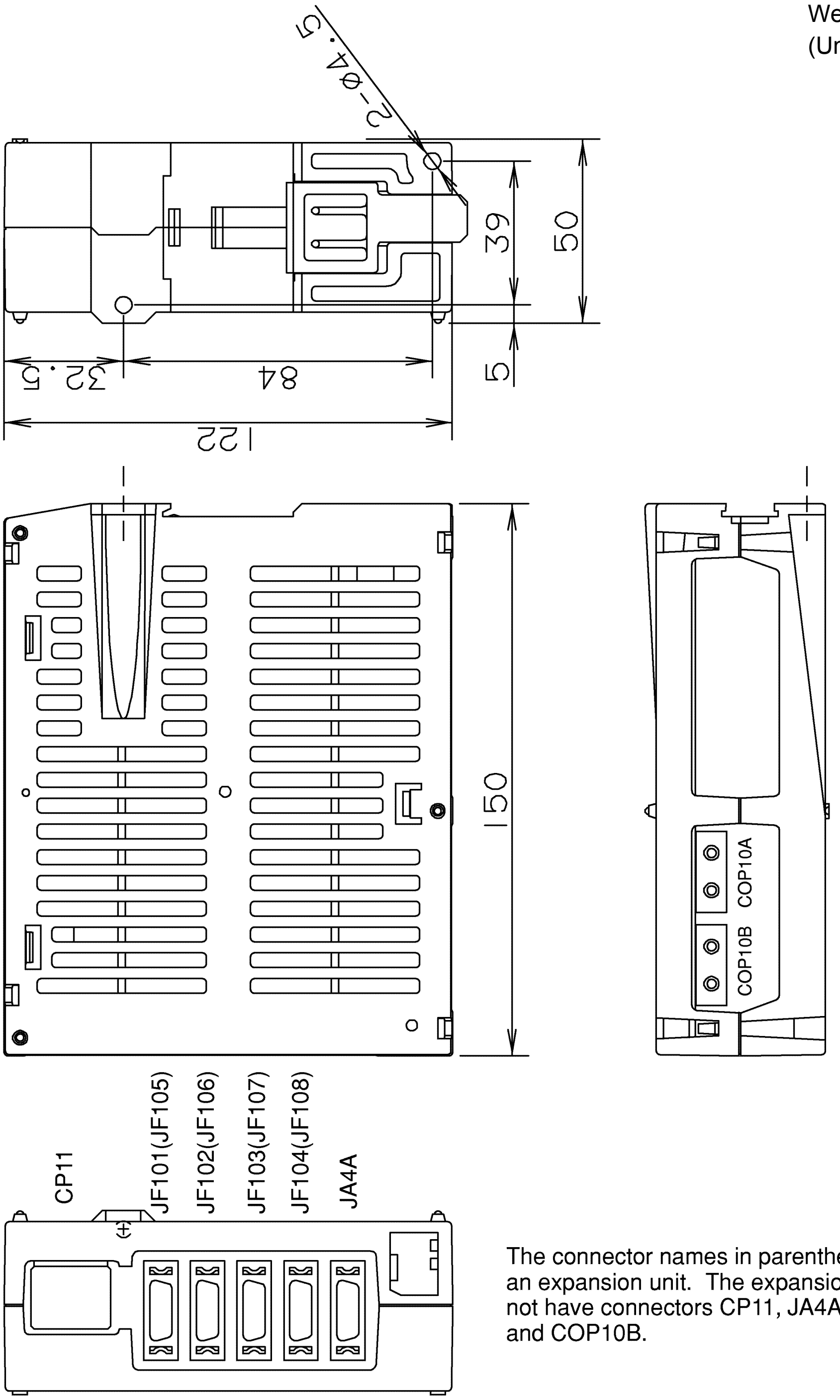


Fig. U20 Separate detector interface unit

Weight: 0.4kg
(Unit: mm)



The connector names in parentheses are for an expansion unit. The expansion unit does not have connectors CP11, JA4A, COP10A, and COP10B.

Fig. U21 Battery case for separate detector interface unit (ABS)

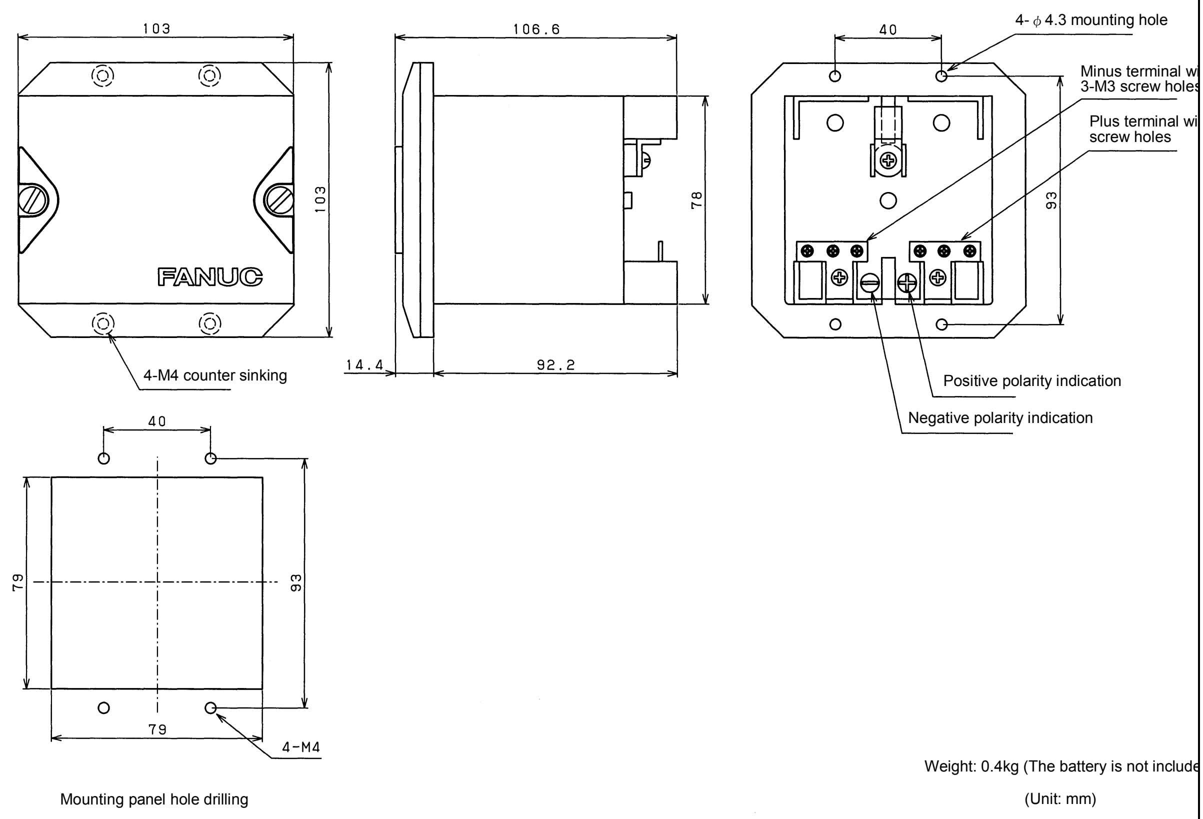
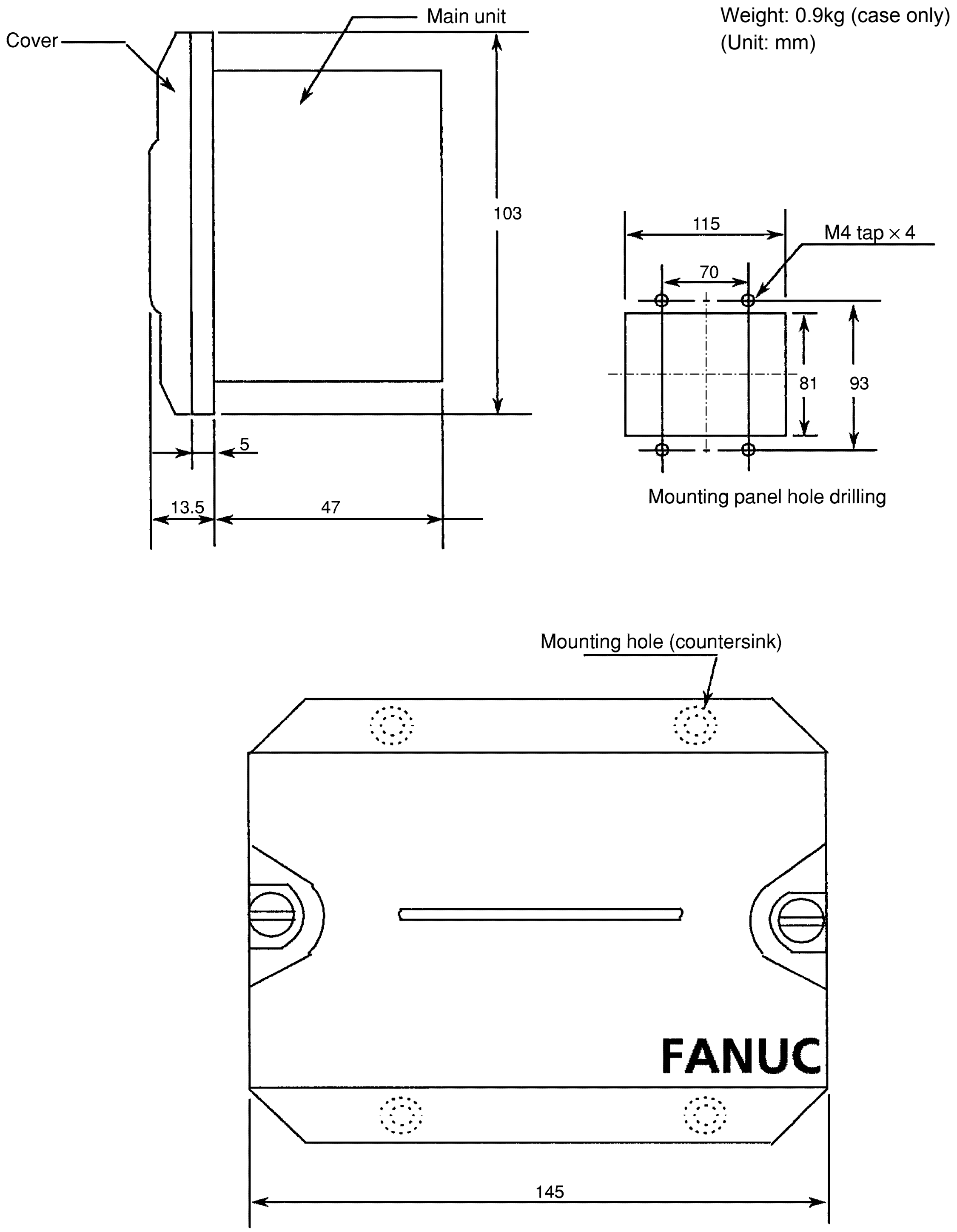
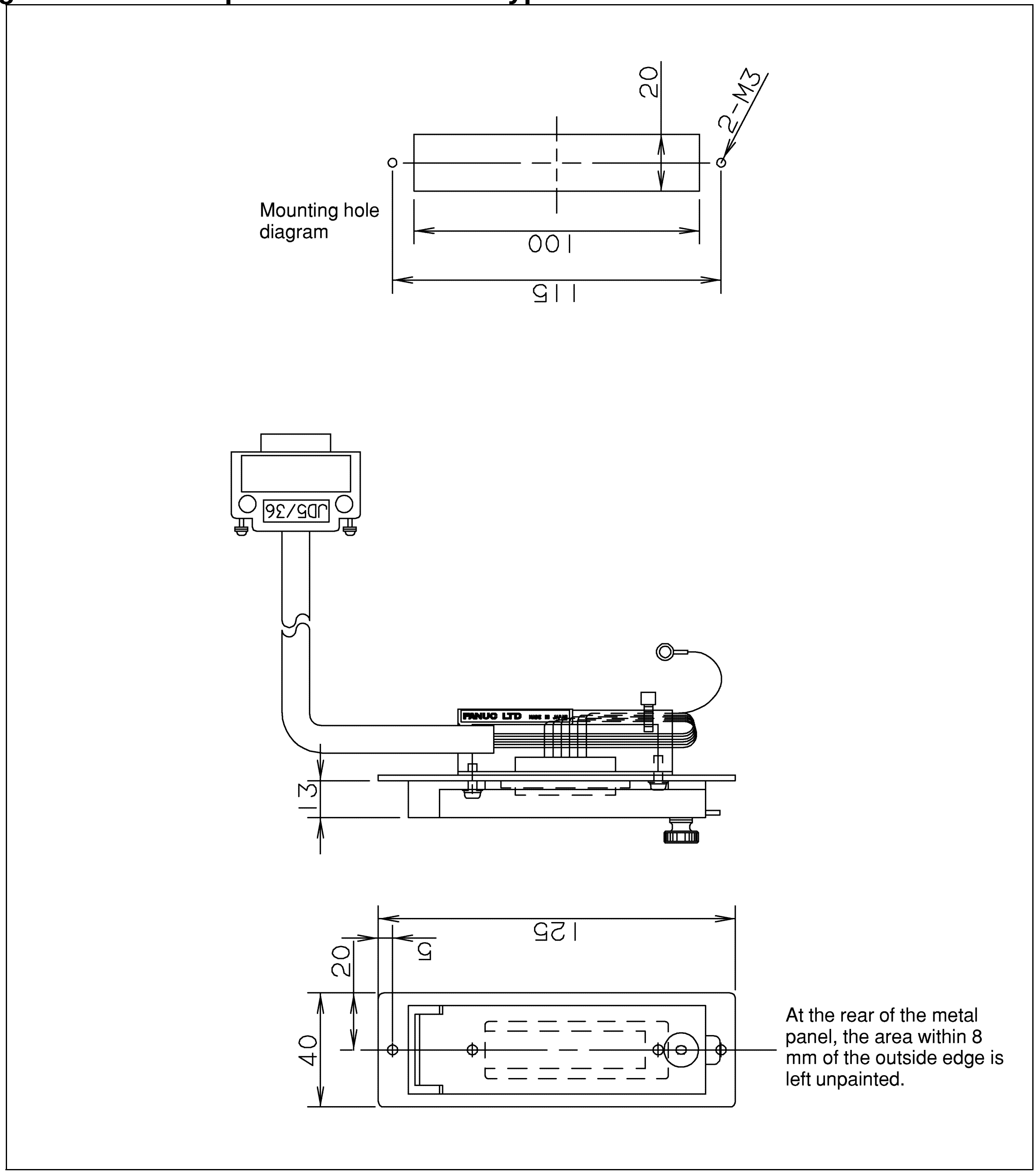


Fig. U22 CNC battery unit for external installation



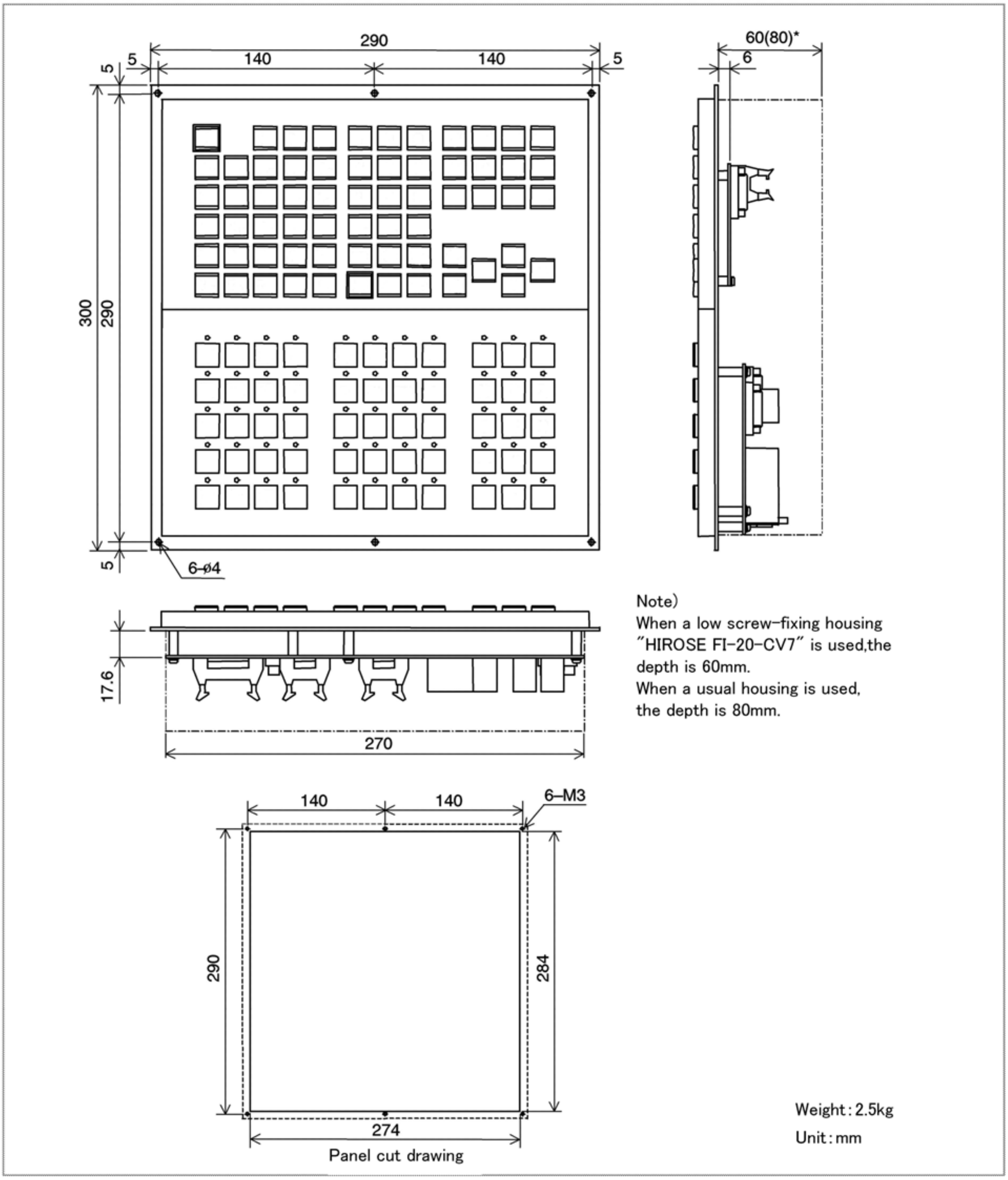
The battery unit is fitted with a 14–m battery cable.

Fig. U24 Punch panel Narrow width type



At the rear of the metal panel, the area within 8 mm of the outside edge is left unpainted.

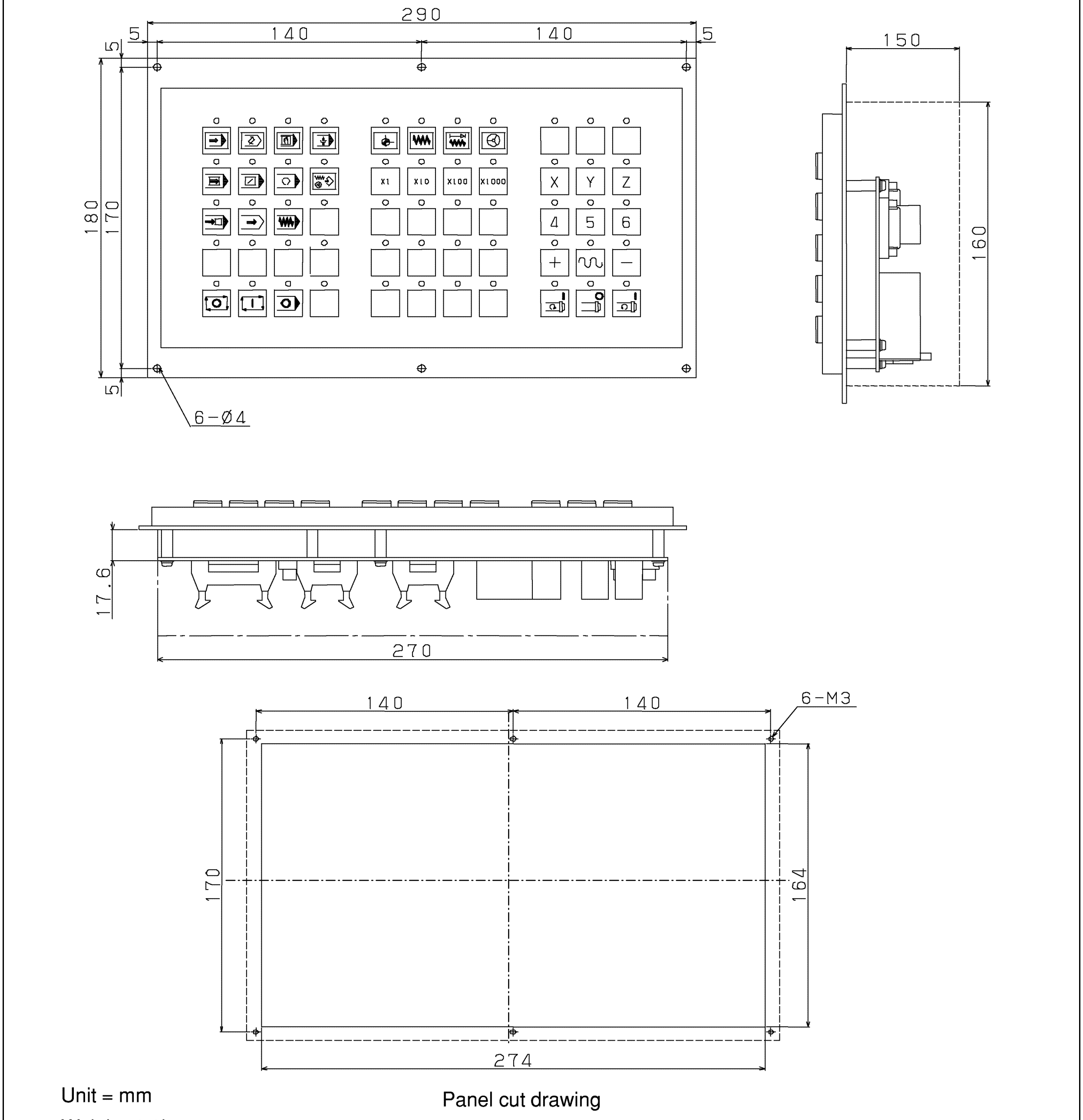
Fig. U25 Machine operator's panel: Main panel A



Note)
When a low screw-fixing housing "HIROSE FI-20-CV7" is used,the depth is 60mm.
When a usual housing is used, the depth is 80mm.

Weight : 2.5kg
Unit : mm

Fig. U26 Machine operator's panel: Main panel B

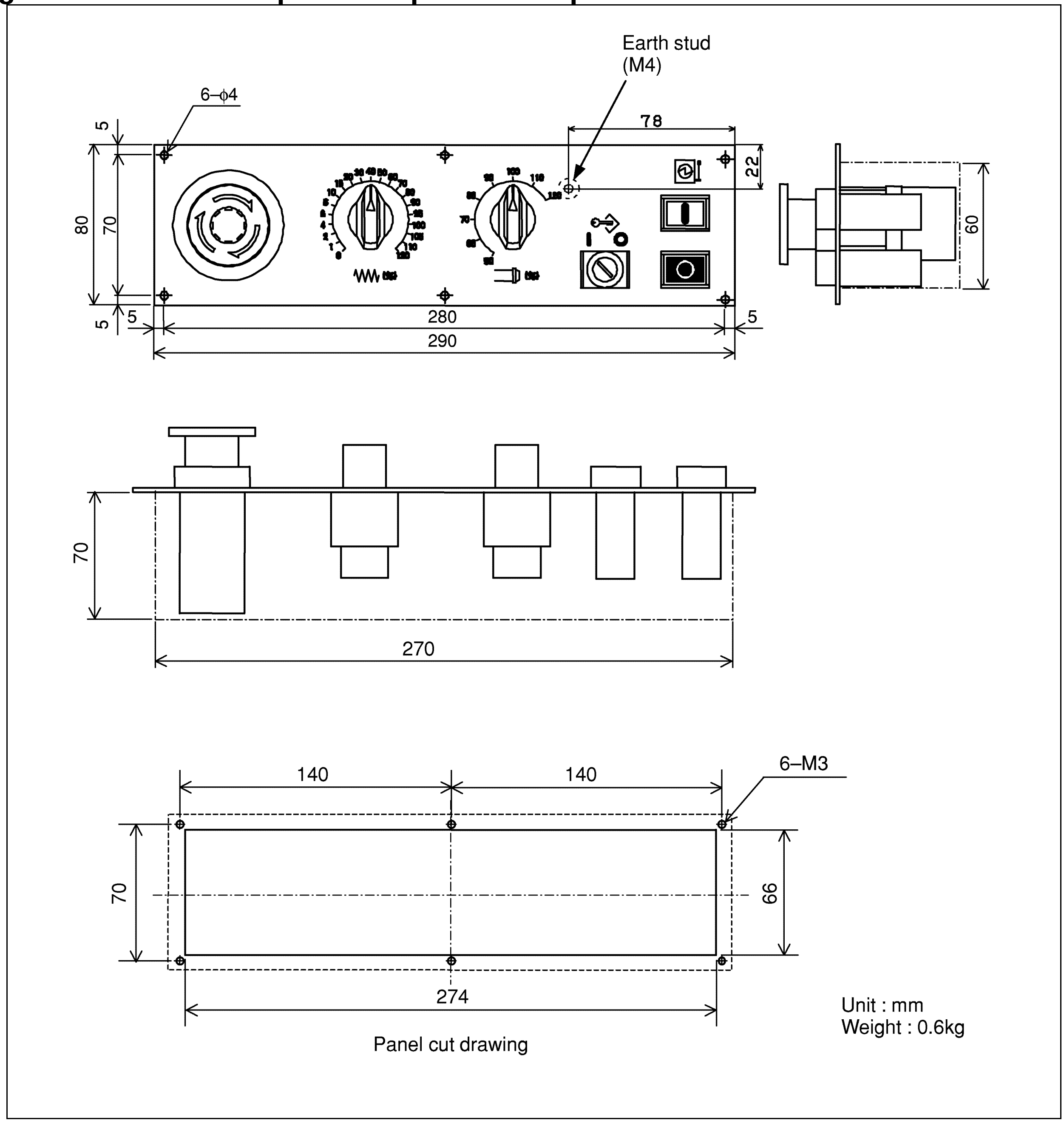


Unit = mm

Weight: 1.6kg

Panel cut drawing

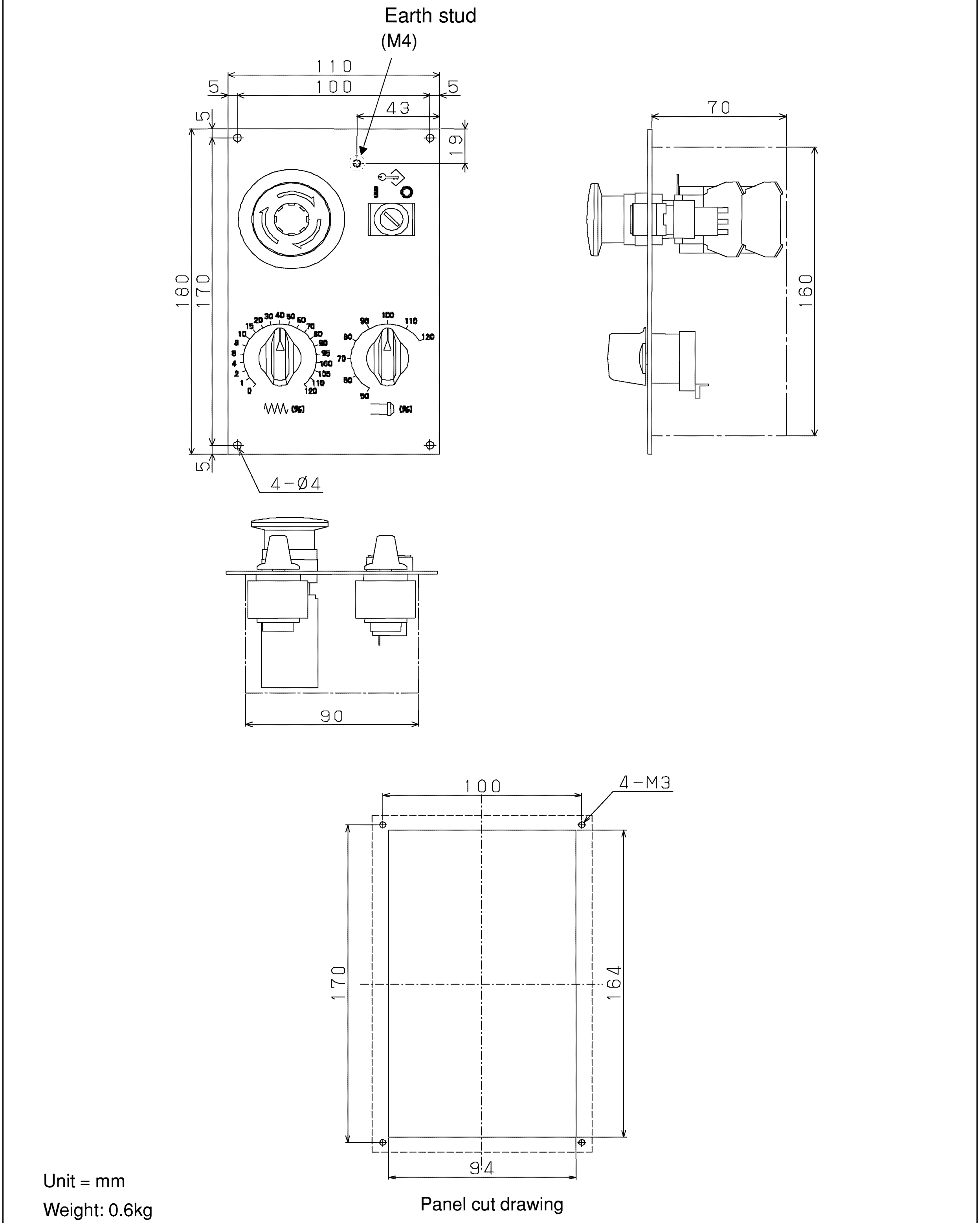
Fig. U27 Machine operator's panel: Sub panel A



Panel cut drawing

Unit : mm
Weight : 0.6kg

Fig. U28 Machine operator's panel: Sub panel B1

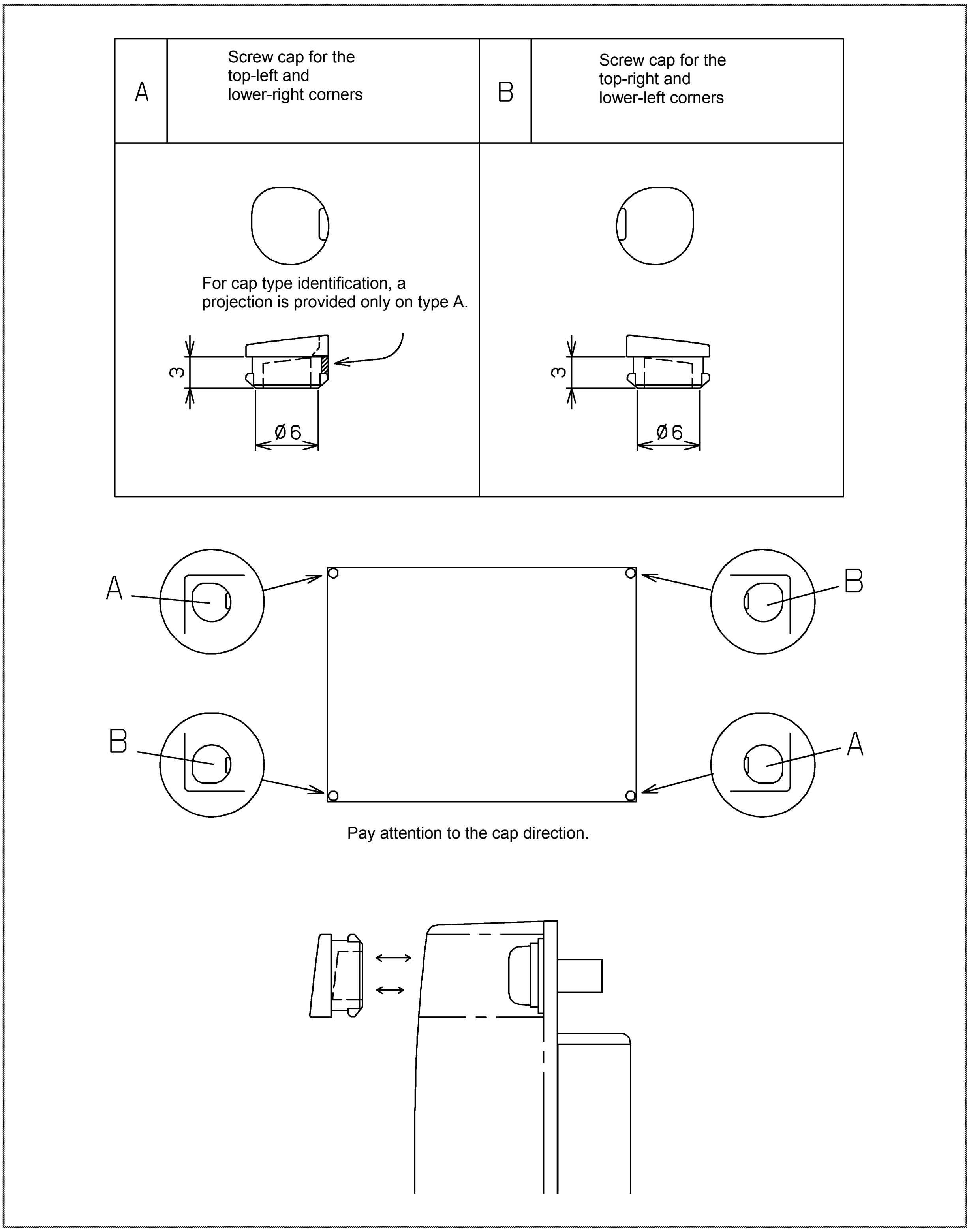


Unit = mm

Weight: 0.6kg

Panel cut drawing

Fig. U29 Screw cap (for small MDI unit)



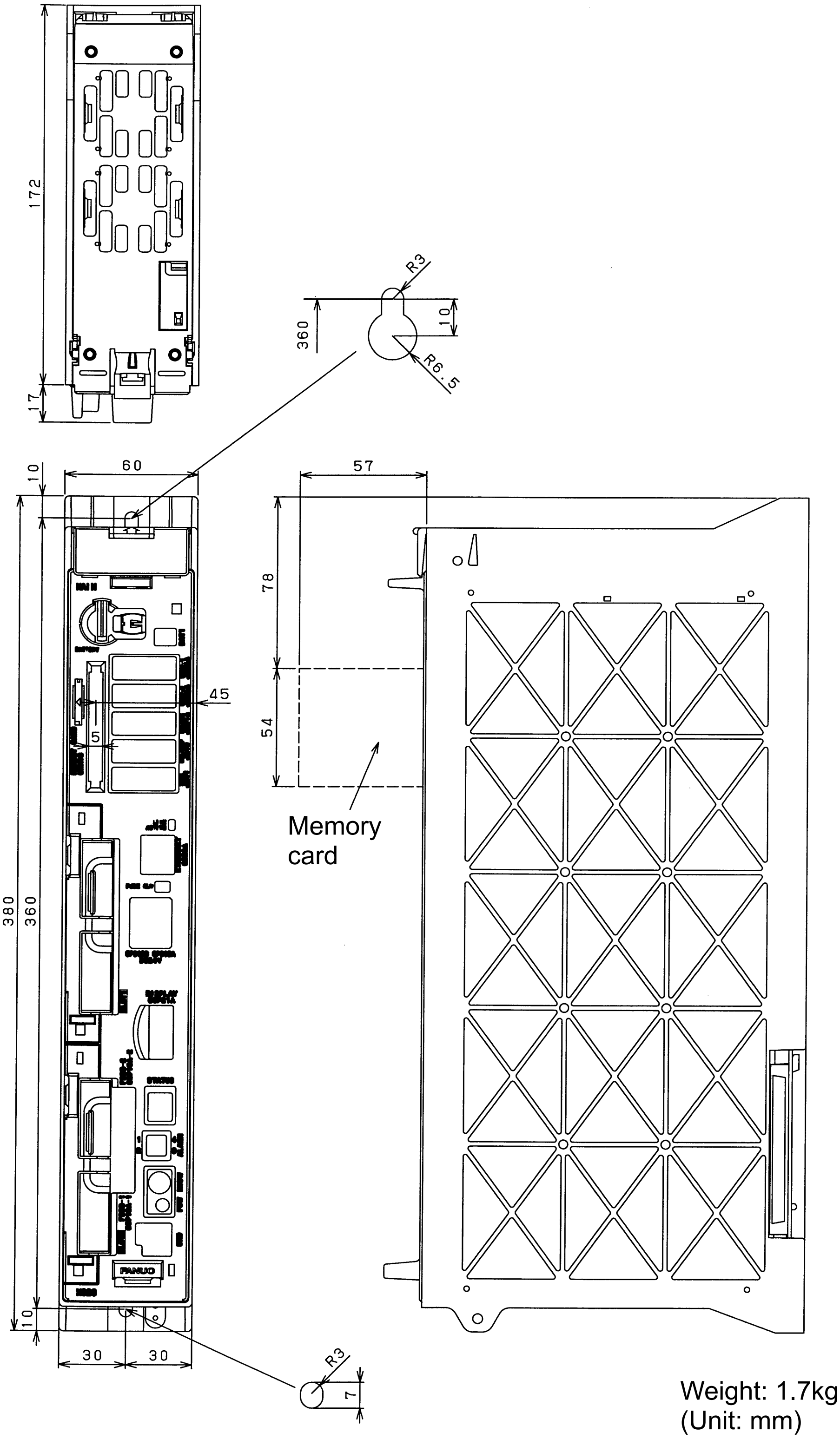
A Screw cap for the top-left and lower-right corners

B Screw cap for the top-right and lower-left corners

For cap type identification, a projection is provided only on type A.

Pay attention to the cap direction.

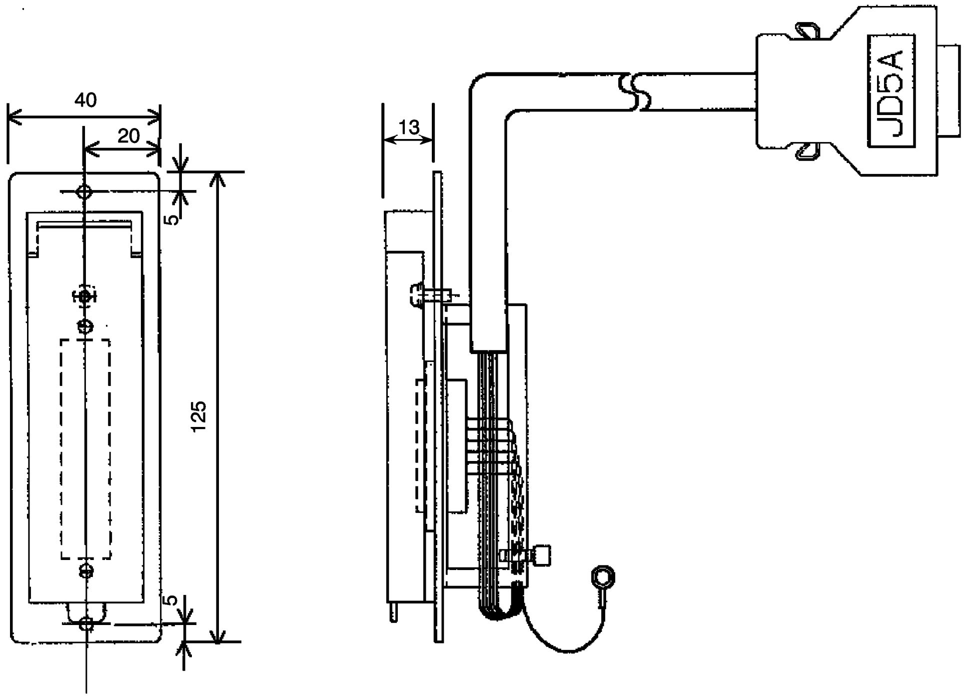
Fig. S1 Stand-alone type control unit



Weight: 1.7kg
(Unit: mm)

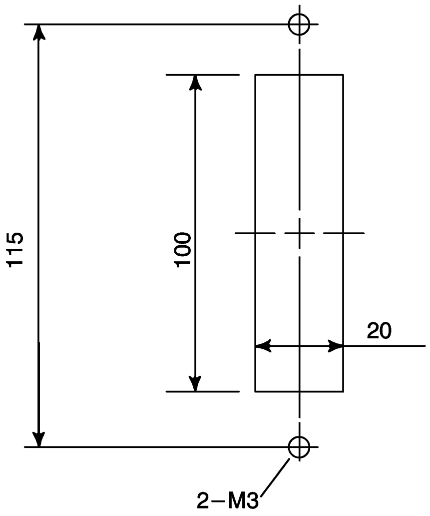
Fig. S2 Punch Panel (for Stand-alone Type Control Unit)

Specification: A02B-0120-C191,C192,C193



A paint masking zone 8 mm wide is provided on the periphery of the rear of the plate.

The panel cut dimensions of this punch panel are indicated below.



> **NOTE**
> This punch panel cannot used for the PANEL *i*.

INDEX

Revision Record

FANUC Series 0*i*-MODEL D/Series 0*i* Mate-MODEL D DESCRIPTIONS (B-64302EN)

Edition	Date	Contents	Edition	Date	Contents
01	Jun., 2008	—			

B-64302EN/01

* B - 6 4 3 0 2 E N / 0 1 . 0 1 *